Creator

Online Help Summary

Version 2.6
April 2003



MultiGen-Paradigm
VISUALIZE REALITY

Creator Online Help Summary, Version 2.6

© 2003 MultiGen-Paradigm, Inc.
All trademarks, trade names, service marks and logos referenced herein belong to their respective companies. All rights reserved.

MultiGen-Paradigm, Inc. (MultiGen-Paradigm) PROVIDES THIS MATERIAL AS IS, WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MultiGen-Paradigm, Inc. may make improvements and changes to the product described in this document at any time without notice. MultiGen-Paradigm assumes no responsibility for the use of the product or this document except as expressly set forth in the applicable MultiGen-Paradigm agreement or agreements and subject to terms and conditions set forth therein and applicable MultiGen-Paradigm policies and procedures. This document may contain technical inaccuracies or typographical errors. Periodic changes may be made to the information contained herein. If necessary, these changes will be incorporated in new editions of the document.

MultiGen-Paradigm, Inc. is the owner of all intellectual property rights in an to this document and any proprietary software that accompanies this documentation, including but not limited to, copyrights in and to this document and any derivative works therefrom. Use of this document is subject to the terms and conditions of the MultiGen-Paradigm Software License Agreement included with this product.

No part of this publication may be stored in a data retrieval system, transmitted, distributed or reproduced, in whole or in part, in any way, including, but not limited to, photocopy, photograph, magnetic, or other record, without the prior written permission of MultiGen-Paradigm, Inc.

Use, duplication, or disclosure by the government is subject to restrictions set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause and DFARS 52.227-7013 and in similar clauses in the FAR and NASA FAR Supplement.

Printed in the U.S.A.
Part Number: 1-55395-939-6
April 29, 2003

Product Updates

It is the policy of MultiGen-Paradigm, Inc. to improve products as new technology, software, hardware, and firmware become available. MultiGen-Paradigm, therefore, reserves the right to change specifications without prior notice. Some features, functions, and operations described herein may not be available worldwide.

Contact Information

The following table contains MultiGen-Paradigm, Inc. contact information for product support in Addison.

Contact Method	For Product Support:
Email	vega@multigen-paradigm.com
Website	http://www.multigen-paradigm.com
Customer Support and Licensing	877-289-5670 (toll free)
Mail	MultiGen-Paradigm, Inc. 5465 Legacy Drive Plano TX 75024 USA

The following table contains MultiGen-Paradigm, Inc. contact information for product support in San Jose.

Contact Method	For Product Support:
Email	creator@multigen-paradigm.com
Website	http://www.multigen-paradigm.com
Customer Support and Licensing	877-289-5670 (toll free)
Mail	MultiGen-Paradigm, Inc. 2044 Concourse Drive San Jose CA 95131 USA

Contents

Chapter 1 Getting Started 33

System Requirements 33
Starting Creator 34
Exiting Creator 35
Version Information 35
License Information 35
- Working With Network License Configurations 36
- Listing Other License Configuration Users 36

Creator Extras 37
- Model Libraries and the Partners CD 37

Learning About Creator 37
Document Conventions 38
Online Help 39
- Context-Sensitive Help 40
- Tool Tips 40

Getting Technical Support 41

Chapter 2 Customizing Creator 43

Environment Variables 43
- To Set Environment Variables in Windows NT 45
- To Set Environment Variables in Windows 2000 46
- To Set Environment Variables in Windows XP Professional 46
- To Set Environment Variables in IRIX 47

Preferences 49
- State Preferences 49
- Editing Preferences 53
- Grid Preferences 56
- Color Preferences 56
- BSP Preferences 58
- Texture Preferences 59
- Flight Preferences 60
- Nintendo Preferences 64

Plug-ins 66
- Using Help on Plugins 66
- Making Plug-ins 67
- The Plug-In Wizard 67

Creator Options and Product Levels 67

Chapter 3 Windows and Views 69

The Application Window 69
The Database Window 70
Toolbar Buttons 71
- New File 71
- Open File 71
- Save File 72
- Cut 72
- Copy 72
- Paste 72
- Undo 72
- Rotate View 72
- Pan View 72
- Zoom View 73
- Pick Mode 73
- Mode Select List 73
- Fence Touch On 73
- Fit on Pick (Ctrl+F) 73
- Fit on Select 73
- Reset View (U) 74
- Fit on Grid (Ctrl+Shift+V) 74
- View Panel 74
- Coordinate Panel 74
- Select Panel 74
- Primary Color 74
- Alternate Color 75
- Current IR Color 75
- Current Material 75
- Current Light Source 75
- Current Texture 75
- Current Texture Mapping 75
- Sound Palette 75

Control Bar Features 76
- Eyepoint Menu 76
- Eyepoint Clocks 76
- Eyepoint Coordinates 77
- Tracking Plane Menu 77
- Freeze Flags 77

Managing Database Windows 77
Switching Views 78
Zoom View 78
CAD View 79
Reset CAD View 79

Standard View 80
Special Controls 80
Clocks 80
Toolbox Tear-off Windows 81
Current Buttons 81
Statistics 81
Error Messages 82
Window Panes 83

Chapter 4 Toolboxes 85

Face Tools 85
Polygon Tool 85
Rectangle Tool 86
Rectangle Plus Tool 87
Circle Tool 91
Circle Plus Tool 92
Geometry Tools 96
Wall Tool 96
Wall Plus Tool 98
Peak Tool 103
Sphere Tool 105
Revolve About Edge Tool 106
Strip Face Tool 108
Rounded Strip Face Tool 111
Loft Tool 115
Curve Tool 116
3D Text Tool 119
Extrusion Tool 121
Flip Book Tool 128
Billboard Wizard 132
Tree Wizard 132
Building Wizard 132
Roof Tool 133
True Type Text Tool 139
Maneuver Tools 144
Translate Tool 144
Rotate About Point Tool 146
Rotate About Edge Tool 148
Scale Tool 149
Rotate - Scale to Point Tool 151
Put Tool 154
Insert Transformation Matrix Tool 157

Duplicate Tools 159
Duplicate Tool 160
Replicate Tool 162
Populate Tool 164
Scatter Tool 168
Modify Geometry Tools 170
Mirror Tool 170
Project Tool 172
Plant Tool 173
Linear Bend Tool 173
Slice Tool 174
Cookie Cutter Tool 176
Lattice Deform Tool 177
Bend Deform Tool 178
Twist Deform Tool 180
Taper Deform Tool 181
Bulldoze Tool 182
Modify Face Tools 184
Reverse Face Tool 184
Split Face Tool 185
Add Vertex Tool 186
Triangulate Tool 188
Combine Faces Tool 189
Cut Subfaces Tool 190
Edge Turn Tool 191
Tessellate Tool 193
Divide Quads Tool 194
Orient Faces Tool 196
T-Vertex Killer Tool 197
Modify Vertex Tools 200
Modify Vertex Tool 200
Match Vertices Tool 202
Average Vertices Tool 204
Dimmer Tool 204
Properties Tools 205
Insert Line Style Tool 206
Get Line Style From Face Tool 206
Paste GeoFeature Attributes Tool 207
Copy GeoFeature Attributes Tool 208
Edge Tools 209
Edge From Mouse Tool 209
Parallel to Edge Tool 210
Perpendicular to Edge Tool 212

Perpendicular to Trackplane Tool 214
Centerline Tool 216
Intersection of Two Planes Tool 217
Construction Curve Tool 218
Vertex Tools 220
Construct Vertex From Mouse Tool 220
Construct Average Vertex Tool 221
Intersection of Two Edges Tool 222
Closest Point on Line Tool 222
Interval Between Two Vertices Tool 223
Closest Point on Plane Tool 224
Saving and Restoring Construction Vertices and Edges 224
Create Tools 225
Create Tool 226
Create Group Tool 226
Create Object Tool 227
Create LOD Tool 227
Create DOF Tool 227
Create Sound Tool 227
Create Light Point Tool 227
Create Light Point System Tool 227
Create Light Source Tool 228
Create BSP Tool 228
Create Switch Tool 228
Create External Reference Tool 228
Create Instance Tool 228
Create Clip Node Tool 228
Create Text Node Tool 229
Hierarchy Tools 229
Insert Level Above Tool 229
Delete Level Tool 229
Attach Tool 230
Detach Tool 230
Shift Left Tool 230
Shift Far Left Tool 230
Shift Right Tool 230
Shift Far Right Tool 231
Toggle Display Tool 231
Reorganize Hierarchy Tool 231
Expand Hierarchy Tool 231
Contract Hierarchy Tool 232
Collect Tool 232

Mode Tools 233
Group Mode Tool 233
LOD Mode Tool 234
Object Mode Tool 234
Face Mode Tool 234
Edge Mode Tool 234
Vertex Mode Tool 234
DOF Mode Tool 235
Sound Mode Tool 235
Switch Mode Tool 235
Light Mode Tool 235
External Reference Mode Tool 235
BSP Mode Tool 236
Curve Mode Tool 236
Clip Mode Tool 236
Light Point Mode Tool 236
Light Point System Mode Tool 236
Road Mode Tool 236
Grid Mode Tool 237
Helper Mode Tool 237
Surface Mode Tool 237
Torsion Mode Tool 237
Text Mode Tool 237
Adaptive Mode Tool 237
Mesh Mode Tool 237
Custom Toolbox 238
Tessellation Tools 239

Chapter 5 Working with Database Files 241

Opening Database Files 242
Opening Recent Files 243
Creating New Database Files 244
Undoing Work 244
Saving Database Files 244
To Save a New File 245
To Save and Back Up Files Automatically 245
Saving With a New Name 247
Saving Portions of a Database 248
Saving Checkpoint Files 248
Using Snapshot 248
To Take a Snapshot 248
Closing a Database 248

Chapter 6 Working in the Graphics View 251

Adjusting The Eyepoint 251
Changing the Eyepoint Position 251
Rotating the Eyepoint 255
The Yaw, Pitch, and Roll Clocks 257
Changing the Center of Rotation 257
Understanding the Default Eyepoint Position 262
Understanding the Viewing Volume 263
Controlling the Viewing Volume 264
Controlling the Graphics View Appearance 267
Creating Fog 267
Setting Sky Color 269
Drawing Controls 271
Changing the Database View 273
View/Graphics 273
Select/Isolate 273
View/Full View 273
Using Vega Viewer 274
Using Interceptor 3D Viewer 276

Chapter 7 Working in the Hierarchy View 279

Expanding and Collapsing Hierarchy Levels 281
To Expand Specific Nodes 282
To Expand The Database By Levels 282
To Collapse Specific Nodes 282
To Collapse The Database By Levels 283
Resetting the View 283
Changing the Hierarchy View Color 284
Selecting Nodes 284
Viewing Selected Nodes 285
Zooming in the Hierarchy View 285
Panning in the Hierarchy View 286
Editing the Hierarchy Structure 287
Drag-and-Drop Editing 287
Priority and Shifting Nodes 288
Database Identification Names 289
To Rename a Node 289
Database Parent 290
Setting the Parent 291
Default Parenting 291
Database Node Types 292

Database Header 292
Groups 293
Objects 293
Faces 294
Edges 295
Vertices 295
Saving Selected Nodes 296
To Save With Write Select 296
Designing An Efficient Database 299
Database Traversal 299
Database Hierarchy Reorganization 300
To Reorganize a Database Hierarchy 300

Chapter 8 The Tracking Plane and Grid 303

Defining the Grid 303
Setting Grid Preferences 304
Setting Default Grid Colors 305
Grid and Trackplane Controls 306
Placing a Coordinate on the Tracking Plane 312
Storing Custom Tracking Planes 313
To Store a Tracking Plane 313
To Recall a Tracking Plane 313
To Store and Recall Both Eyepoint and Tracking Plane 314

Chapter 9 The View Panel 315

Lights On/Off (L) 316
Draw Vertex Normals 316
Draw Degree of Freedom Axes 316
Antialias Lines 316
Infrared View 316
Ortho/Perspective View 317
Draw Vertex Large 317
Draw Lights Large 317
Draw Wire Backface 317
Fog On/Off 317
Shrink Mode 318
Draw Vertex Numbers 318
Draw Control Points 318
Dynamic Wireframe 318
Background Image 318

Draw Bounding Volumes 319
Display Update Rate 319
Draw Select Outline 319
Draw Constructions 319
Draw Clip Nodes 319
Draw Grid Nodes 320
Draw Helper Nodes 320
Draw Torsion Nodes 320
Draw Light Sources 320
Draw Sounds 321
Texture Layers 321
Draw Mode 321
Grid On/Off (G) 322
Grid Snap 322
Grid Larger 322
Grid Smaller 323
Rotate Grid 323
Coordinate Plane Buttons (XY, XZ, YZ) 323
Trackplane From Face 323
Trackplane Center 323
Trackplane From Vertex 323
Perpendicular Tracking Plane (P) 324
Trackplane From View 324
Grid Type 324
Offset 324
Draw Grid 325
Dimension 325
Grid Axes 325
Quadrants On/Off 326

Chapter 10 Selecting 327

Fence Selection 328
Customizing Selection Highlights 328
Controlling Highlights in the Graphics View 329
Using the Select List 329
To Add Items to the Select List 330
To Replace the Contents of the Select list 330
To Remove Items From the Select List 330
Displaying Attributes in the Select List 331
Selecting With Arrow Keys 331
Using The Select Menu 331
Isolate 332

Select From ID 333
Select All 333
Deselect All 333
Select All Lights 334
Select Empty Nodes 334
Select Concurrent Vertices 334
Select Faces with Coaligned Vertices 334
Select Nonplanar Faces 334
Select Backfacing Faces 335
Select Thin Faces 335
Select Concave Faces 335
Select Concave Objects 335
Select Adjacent Faces 336
Select Connected Faces 337
Select Non-Manifold Faces 338
Select Concurrent Faces 339
Change Mode 339
Fence Touching 340
Select List 340

Chapter 11 The Coordinate Panel 341
Reading the Coordinate Panel 341
Using the Coordinate Panel With Tools 342

Chapter 12 Modeling Tips and Techniques 345
Working With the Graphics Buffer 345
Viewing the Contents of the Graphics Buffer 345
Copying and Pasting 346
Replace Graphics 347
Centering a Model at the Origin 352
Referencing a Vertex or Edge 355
To Reference a Vertex 355
To Reference an Edge 356
To Snap a Vertex to the Tracking Plane 356
Changing the Modeling Mode 357
Correcting Non-coplanar Vertices 357
Using Quick Palettes 358
To Select an Item From a Quick Palette 359
To Add an Item to a Quick Palette 359
To Delete an Item From a Quick Palette 360

To Remove or Restore Quick Palette Items 360
To Update Quick Palettes 361
Using Background Images 361
To Load a Background Image 361
To Zoom or Move a Background Image 362
To Align a Model to the Background Image 363
To Model with a Background Image 363
Working With Local Coordinate Systems 365
To Define a Local Coordinate System 366
To Define a Local Coordinate System With a Transformation 367
To Remove a Local Coordinate System 367
To Add Geometry to a Local Coordinate System 367
To Remove Geometry from a Local Coordinate System 367
Instances 368
To Create an Instance 369
To Make Copies of an Instance 370
To Change an Instance into Geometry 370
External References 370
Working with External References 371
To Create an External Reference 372
External References in the Graphics View 373
Reading and Re-Reading External References 373
Converting External References to Instances 374
Listing and Editing External References 375
Documenting External References 376
External Reference Attributes 379
Billboards 380
To Create a Billboard 380
Positioning a Billboard 381
Meshes 383
Using Meshes With Terrain 383
Using Generate Mesh 384
Editing a Mesh 385
Decomposing a Mesh 385

Chapter 13 Materials 387

Material Preferences 387
The Material Palette 387
Material Palette Menus 388
Creating New Material Definitions 389
Modifying Material Properties 390
Saving and Loading Material Palettes 393

Applying Material Properties 394
Insert Material Tool 395
To Remove a Material From a Set of Faces 395
Get Material From Face Tool 395
Exercise: Apply Material Properties 396

Chapter 14 Color 399

Color Palette Window 399
Primary Color and Alternate Color 401
Current Color Buttons 401
Color Attributes 401
Color Names 402
Color Palette Index 405
Color Intensity 406
RGB Mode 408
Inserting and Getting Color from Face 409
Insert Color Tool 409
Get Color From Face Tool 410
Editing Colors 410
Editing Colors in Windows 411
Editing Colors in IRIX 412
Resetting the Color Palette 413
Saving and Loading Color Palettes 413
Saving a Color Palette 413
Loading a Color Palette 414
Creating a Default Color Palette 414
Color Matching 414
Infrared Color 415
Infrared Codes 415
Infrared View 415
Infrared Palette 416
Infrared Modeling Tools 417
Exercise: Adding Color to a Model 418

Chapter 15 Texture 421

Texture Palettes 422
To Add Textures to the Palette 424
Current Texture 425
The Texture Palette Menus 425
Organizing Texture Palettes 434

Texture Palette Files 441
Applying Texture 444
Texture Tools 444
Texture Tools in the Properties Toolbox 461
Removing a Texture 462
Using Chop Texture to Reduce Texture Size 464
Calculating Memory Requirements 466
Tips for Working with Textures 467
Exercise: Applying a Texture 468
MultiTexture 470
To Apply Multiple Textures 472
Texture Mapping with MultiTexture 473
Copying Texture Layers 474
Reordering Texture Layers 476
Deleting Texture Layers 477
Subtexture 478
Creating a Texture Collage for Subtextures 479
Defining and Applying Subtextures 479
Modify Texture Tools 481
Translate Texture Tool 481
Rotate Texture Tool 484
Scale Texture Tool 486
Modify Mapping Tool 488
Eight Point Warp Texture Tool 488
Modify UVs Tool 490
Texture Composer Tool 499
The Texture Mapping Palette 516
To Apply the Current Texture Mapping 517
To Change the Current Texture Mapping 518
To Create a New Texture Mapping 518
To Save a Texture Mapping Palette File 519
To Load a Texture Mapping Palette File 519
Texture Mapping Palette Menus 520
Displaying Texture in the Database 521
Draw Texture 521
Toggle Alpha Blending 522
Dynamic Texture 523
Texture Attributes 523
Format Attributes 524
Wrap Method Attribute 526
Size, Up Vector, and Pivot Attributes 527
Minification/Magnification Attributes 527
Detail Texture 530

Nintendo 64 Texture Attributes 540
Real 3d Pro 1000 Texture Attributes 540
Texture Variables and Preferences 541
The MPI_CREATOR_TXTPATH Environment Variable 541
Texture Preferences 541
Editing Texture Patterns 545
Installing Adobe Photoshop Plug-ins (Windows Only) 545
Setting an Image Editor (Windows) 545
Setting an Image Editor (IRIX) 547
The Texture Editor 548
Starting the Texture Editor 549
Reading the Texel Display 550
Controlling the Eyepoint 550
Supported Image Formats 551
Texture Editor Menus 552
Using the Draw Tools 558
Selecting Texels 563
Defining the Pen Tip 567
Choosing a Paint Mode 570
Exercise: Drawing With a Pen Tip 575
Displaying and Protecting Texels 577
Using Texture Operations Tools 578
Exercise: Selecting and Flipping Texels 583
Using Texture Maneuvering Tools 584
The Texture Compositor 586
The Texture Sequencer 589
The Fractal Texture 594

Chapter 16 Light Sources and Database Shading 599

Light Sources 599
Using the Light Source Palette 600
Creating Light Sources 602
Modifying Light Source Definitions 603
Light Source Tips 610
Enabling Light Sources 611
Modeling Lights 611
Light Source Nodes 612
Placing and Viewing Light Source Nodes 613
Modifying Light Source Nodes 615
Saving/Loading Light Source Palettes 617
To Save a Light Source Palette to Disk 617
To Load a Saved Light Source Palette 617

To Save a Default Light Source Palette 618
To Copy a Light Source 618
Setting Lighting Preferences 618
Database Shading 620
Shading Preferences 621
Shading the Database 622
Modifying Shade Characteristics 625
Shade Attributes 625
Vertex Normals 626

Chapter 17 Light Points and Light Strings 631

Creating Light Points or Strings 631
Light Point/String Attributes 634
Light Point Attributes 634
Light Point Color and Percentages Parameters 637
Light Point LOD Attributes 638
Light Point Directionality 639
Light Point Palette 640
Using the Light Point Palette 641
Light Point Palette Attributes 644
General Attributes 644
Directional Attributes 646
SGI Performer Attributes 648
Calligraphic Display Attributes 651
Vega Prime Appearance Attributes 652
Animation Attributes 654
Vega Prime Animation Attributes 656
Creating a Sequence Animation 657
Sequence Animation Attributes 658
Rotating Animation Attributes 658
Strobe Animation Attributes 659
Morse Code Animation Attributes 659
Light Point Tools 660
Create Light Point Tool 660
Light Point Mode Tool 660
Light Point System Mode Tool 660
Draw Lights Large Tool 661
Draw Vertex Normals Tool 661
Modifying Light Point Attributes 661
Modifying Light Point System Attributes 662
Modifying Light Point Colors 663
To Change the Light Point Front Color 664

To Change the Light Point Back Color 664
To Change Both Front and Back Light Point Colors 664
Modifying Light Point/String Placement 665
Modifying Individual Light Points, Polygons, and Grids 665
Modifying Light Strings 666
Deleting Light Points 667

Chapter 18 Audio Modeling 669

The Sound Palette 669
Loading Sounds 670
Editing Sounds 671
Deleting Sounds 671
Loading and Saving Sound Palettes 671
Sound Nodes 672
To Create Sound Nodes 672
To Hide or Display Sound Nodes 673
To Select Sound Nodes 674
To Insert Sounds 674
To Remove Sounds 674
To Position Sound Nodes 675
To Play Sounds in the Database 675
Sound Attributes 675
Exercise: Adding Sound to the Database 678

Chapter 19 Realtime Considerations 681

Realtime Overview 681
Frame Rate 681
Bounding Volumes and Collision Detection 682
Draw Order 682
Choosing a Draw Order 683
Fixed List Drawing 685
Binary Separating Plane (BSP) Drawing 686
Z-buffer Drawing 686
Database Traversal Order 688
Relative Priority 688
The Viewing Volume 689
Changing the Clipping Planes 689
Adjusting the Eyepoint Scale 691
Multiple Eyepoint Viewing Volumes 692
Bounding Volumes 694

Displaying Bounding Volumes 695
Changing the Bounding Volume 696
Translating Bounding Volumes 698
Exercise: Draw Order 700

Chapter 20 Levels of Detail 707

Creating LODs 707
Level-of-Detail Hierarchy 709
Nested LODs 710
Independent LODs 710
Additive LODs 711
Selecting LODs 711
Displaying and Modifying LOD Attributes 712
Switching LODs in Creator 714
More Detail (Ctrl+M) 714
Most Detail (Ctrl+Shift+M) 714
Less Detail (Ctrl+L) 715
Least Detail (Ctrl+Shift+L) 715
Set LOD Distance 715
Auto LOD 715
Eyepoint LOD 715
Morphing Between LODs 716
How Morphing Works 716
Working With Morph Vertices 717
Generating an LOD Automatically 720
To Generate an LOD 721
Generate LOD Parameters 723
Tips for Working With Generate LOD 725
Exercise: Working With LODs 725
View Two LODs 726
View LOD Switch Attributes 728
Set Up Morphing Between LODs 729
Generate a Lower Detail Model 729

Chapter 21 Making Things Move 735

Degrees of Freedom 735
Overview: Creating Degrees of Freedom 736
Creating DOF Nodes 737
Displaying DOFs in the Graphics View 737
Selecting DOFs 737

Positioning DOF Axes 737
Setting DOF Limits 739
Using the DOF Viewer 745
Exercise: Creating a DOF 746
Animation 759
Overview: Animating a Model 759
Setting the Animation Attributes 760
Controlling Animation 761
Exercise: Creating an Animation 762

Chapter 22 Instrumentation 769

Creating a Dial Gauge 770
Defining the Dial Background 772
Adding Labels to the Dial 775
Adding Arcs 779
Completing the Gauge 780
Creating a Vertical Gauge 784
Defining the Gauge Background 786
Adding Labels to the Gauge 788
Completing the Gauge 790
Creating a Cylindrical Gauge 790
Defining the Gauge Structure 792
Adding Labels to the Gauge 794
Completing the Gauge 795
Creating a Pitch Ladder 795
Defining the Pitch Ladder Background 798
Adding Degree Marks 800
Adding Labels to the Gauge 801
Completing the Gauge 802
Clipping Nodes and Planes 802
Creating Clipping Nodes 803
Clipping Node Attributes 806
Nesting Clipping Nodes 807
Displaying Clipping Polygons 807
Using the Font Editor 807
Editing a Font 807
Placing 2D Text in the Database 812
Creating Dynamic Text 814
Creating Text Nodes 815
Text Node Attributes 818

Line Styles 821
Line Style Palette 821
Assigning a Line Style 823

Chapter 23 OpenFlight Nodes and Attributes 825

Opening an Attributes Window 826
Closing an Attributes Window 827
Reflect Selection 827
Modify All 827
DB Navigator Panel 828
Comments Panel 830
Header Attributes 831
Header Attributes: Preferences Panel 831
Header Attributes: Projection Panel 833
Group Attributes 835
Group Attributes: Group Panel 835
Group Attributes: Bounding Volume Panel 836
Object Attributes 839
Object Attributes: Object Panel 839
Face Attributes 840
Face Attributes: Colors Panel 841
Face Attributes: Textures Panel 843
Face Attributes: Drawing Panel 845
Face Attributes: Other Panel 847
Vertex Attributes 848
Vertex Attributes: Vertex Panel 849
Vertex Attributes: Vega Prime Light Point Back Color Panel 851
Curve Attributes 851
Mesh Attributes 852
Mesh Attributes: Colors Panel 853
Mesh Attributes: Textures Panel 855
Mesh Attributes: Drawing Panel 856
Mesh Attributes: Other Panel 858
Mesh Attributes: Statistics Panel 860
LOD Attributes 861
Switch Nodes and Switch Attributes 861
To Create a Switch Node 861
Switch Attributes 862
BSP Attributes 864
DOF Attributes 865
DOF Attributes: Translate Panel 866
DOF Attributes: Rotate Panel 867

DOF Attributes: Scale Panel 867
Sound Attributes 868
Light Source Attributes 869
Generic Attributes 869
Attribute Searching 869
Performing an Attribute Search 869
Reopening an Attribute Search 872
Attribute Reports 872
To Generate a Report 872
Attribute Resolver 880
To Resolve Palette Inconsistencies 880
Exercise: Using Attribute Windows 882

Chapter 24 Preference and Default Files 889

Preference and Shortcut Files 889
Palette Files 889
Terrain and Feature Preferences 891
Road Definition File 893

Chapter 25 Keyboard Shortcuts 895

Function Key Definitions 896
File Menu Shortcuts 896
Edit Menu Shortcuts 896
View Menu Shortcuts 897
Info Menu Shortcuts 897
Select Menu Shortcuts 898
Attributes Menu Shortcuts 898
LOD Menu Shortcuts 898
Palettes Menu Shortcuts 899
Window Menu Shortcuts 899
Eyepoint and Tracking Plane Control Shortcuts 899
View Control Shortcuts 900
Texture Mapping Shortcuts 900
Tool Shortcuts 901
Graphics View Shortcuts 902
Hierarchy View Shortcuts 902
Customizing Keyboard Shortcuts 902
Creating Keyboard Shortcuts 903

Chapter 26 Binary Separating Planes 907

Designing Separable Databases 907
What Makes a Database Separable? 907
Combining BSPs and Fixed Listing 908
Understanding BSP Trees 908
Tips for Creating a Separable Database 909
Enabling BSP Draw Order 912
Automatically Separating a Database 913
Setting BSP Preferences 913
Using Check Separability 915
Removing Temporary Planes and Hulls 916
Using Separate Selected 916
Using Separate Children 919
Using Recursive Separation 920
Manually Separating a Database 922
Creating a BSP Node 922
Inserting a BSP in the Database 922
Inserting a BSP Manually 923
Displaying Binary Separating Planes 925
Using Display Planes 926
Using Select Planes 926
Collapsing a BSP Structure 926
Correcting Inseparable Databases 926
Using Diagnosis 927
Correcting Interpenetrating Hulls 928
Using Auto Slice to Insert BSPs 928
Separating Concave Objects 929
BSP Separation and Isolated Views 930
Exercise: Separating a Database 931
Using Separate Selected 932
Using Separate Children 936
Using Recursive Separation 937

Chapter 27 Basic Terrain Conversion 939

Importing Elevation Data 939
Selecting a Gaming Area 941
Choosing Triangulation Parameters 941
Changing Contours and Their Properties 947
Assigning Properties to Contours 947
Adjusting Contours 949
Changing the Fixed Elevation Contour 951

Interpolating a Palette 951
Creating Terrain LODs 952
About Quad Tree LODs 952
Setting LOD Switch Ranges 955
Design Tips for Building Terrain 956
Balancing Culling and Drawing 957
Texture Considerations 958
Choosing LOD Ranges 958
Modify Delaunay 960
Exercise: Using Modify Delaunay 960
Modify Delaunay Requirements 968
To Modify a Delaunay Triangulation 968
The Modify Delaunay Window 969
Exercise: Generating a Terrain Database 971
Import a DED File 971
Change Contours and Properties 974
Create LODs 976
Generate a Test Terrain 977
Set Triangulation Parameters 979
View the Terrain 980

Chapter 28 Pro Terrain Conversion 983

Terrain Process Flowchart 984
Creating Projects 986
To Create a New Project 987
To Load a Project 988
The Terrain Window 988
Opening the Terrain Window 989
Selecting an Area for Processing 990
Changing Contours and Their Properties 993
Setting Terrain Levels of Detail 998
Setting Panel Preferences 1000
The Project Panel 1000
The Map Panel 1003
Map Projection Types 1006
The Triangle Panel - Choosing an Algorithm 1009
Polymesh Terrain Conversion 1011
Delaunay Conversion 1018
Terrain Culture Triangulation 1028
Continuous Adaptive Terrain Conversion 1035
The Texture Panel 1046
The Batch Panel 1048

The Feature Panel 1053
Generating the Terrain 1055
Creating a Terrain 1056
Terrain Tips 1057
Modifying a Database 1057
Modify Delaunay 1057
Lat/Long to X/Y 1058
Convert Database Projection 1059

Chapter 29 Reading and Importing Elevation Data 1063

About Terrain Source Data 1063
DED File Format Description 1064
Raster To DED 1067
Preparing to Use Raster To DED 1067
How to Use the Raster To DED Dialog 1068
Converting Raw Elevation Data 1084
To Convert DTED Files From Hard Disk 1087
To Convert USGS Data to DED Format 1088
To Convert Float Data to DED Format 1089

Chapter 30 Importing and Creating New Features 1091

Converting Feature Data 1091
Importing Feature Data Into Creator 1093
To Import DFD Files 1093
Creating New Features 1099
Creating a New Feature Layer 1099
To Create a Point Feature 1100
To Create an Areal Feature 1102
To Create a Linear Feature 1104
Creating Features From an Image 1106
About Feature Types 1107
Thinning and Reducing Feature Data 1109
Modifying and Correcting Feature Data 1110
To Change Feature Attributes 1110
Using Modeling Tools to Edit Features 1115
Scatter Points in Areals 1118
To Reduce Vertices 1119
To Smooth Features 1121
Displaying and Selecting Features 1122
To Perform a Feature Attribute Search 1123

Showing and Hiding Features 1124
Using Toggle Linears 1125
Saving Feature Data 1125
To Export All Features to DFD 1125
To Export Selected Features to DFD 1126

Chapter 31 Adding Features to Terrain 1127

Types of Feature Projection 1128
Manual Post-Projection 1129
Automatic Post-Projection 1129
Automatic Pre-Projection 1129
Apply Rules to Files 1130
Setting Feature Preferences 1130
To Set Up Feature Preferences 1131
Valid SMCs and Face Attributes 1133
Light Point Preferences 1137
Feature Projection Preferences 1138
Library Model Format 1142
Transferring Attribute Values 1142
Saving and Loading Feature Preferences 1143
Setting Global Projection Preferences 1144
Creating Rules and Actions 1145
Creating Actions 1146
Creating Rules 1153
Assigning Rules to LODs 1156
Document Rules and Actions 1157
Projecting Features Onto Existing Terrain 1157
Projecting Features Into Terrain 1158
Performing Library Substitution 1159
Pre-Projected Features 1163
Cutout Files 1163
Footprint Files 1166
Pathfinder Features in TCT 1166
Batch Culture For Existing Terrain 1169
To Add Features to Terrain Databases 1169

Chapter 32 Batch Processing 1171

Batch Processing Basics 1171
Zooming in Batch Mode 1172
Understanding Master Files 1173

Working with Large Area Databases 1173
Area Block Naming Conventions 1174
To Create a Batch Terrain 1175
Batch Terrain With Feature Projection 1177
To Create Terrain With Features 1177
Projecting Features Without Terrain 1179
To Project Features Without Terrain 1179

Chapter 33 Applying Texture to Terrain 1183

Applying Texture With the Contour Palette 1183
Using Indirect Texture 1185
To Create an Indirect Texture 1186
To Apply an Indirect Texture 1187
Applying a Geospecific Texture 1191
Setting Geospecific Information 1192
Applying Texture in the Terrain Window 1196
The Batch GeoPut Utility 1197
The GeoPut Texture Tool 1202
Creator Terrain Studio 1204

Chapter 34 Conversion Utilities and Filters 1205

Using the Conversion Utilities On Windows Systems 1206
Using the Conversion Utilities on IRIX Systems 1207
The readdma Utility 1207
The readusgs Utility 1209
The float2ded Utility 1210
To Convert Float Data into DED Format 1211
The catdma Utility 1212
The mgattredit Utility 1213
The mgutm2geo Utility 1213
Convert Database to DED 1214
Converting Databases to DED Files 1214
The DED Builder 1219
DED File Cells 1220
Viewing DED Headers 1225
Import Filters 1226
3D Studio Import Filter 1226
AutoCAD Import Filter 1228
Wavefront Object Import Filter 1228
Stereolithography Import Filter 1228

Export Filters 1228
AutoCAD Export Filter 1229
VRML Export Filter 1229
Stereolithography Export Filter 1229

Chapter 35 Road Tools 1231

Road Tools Overview 1231
Constructing Road Sections 1232
Using the Construction Tool 1233
Creating Curves 1244
Special Curves 1245
Creating Hills 1247
Special Hills 1249
Creating Straight Sections 1250
Modifying Roads 1250
Tips for Good Road Design 1251
Superelevation 1251
Spiral Transitions 1251
Arc Radius 1252
Road Tessellation 1252
Selecting Road Sections 1253
Using the Tessellation Tool 1253
Defining the Road Style 1256
Defining Levels of Detail for Roads 1259
Lofting a Road 1265
Adding Features to a Road Definition 1267
Applying and Saving a Road Definition 1272
Tips for Tessellating Roads 1274
Driving the Road 1275
To Drive Along a Roadway 1275
Generating Path Data 1276
Defining the Centerline 1277
Defining a Road Scenario 1280
Defining Scenario Lanes 1283
Creating a Scenario Lane File 1286
Viewing Road Attributes 1289
Road Attributes 1289
Path Attributes 1289
Road LOD Attributes 1290
Road Face Attributes 1291

Chapter 36 Creator Wizards 1293

Billboard Wizard Tutorial 1293
Preparing to Use the Billboard Wizard 1293
Running the Billboard Wizard 1296
Tree Wizard Tutorial 1304
Preparing to Use the Tree Wizard 1305
Using the Tree Wizard 1310
Building Wizard Tutorial 1324
Preparing to Use the Building Wizard 1325
Running the Building Wizard 1332

Chapter 37 Glossary 1347

A 1348
B 1349
C 1351
D 1354
E 1356
F 1357
G 1359
H 1361
I 1361
L 1362
M 1363
N 1365
O 1365
P 1366
R 1368
S 1368
T 1370
U 1371
V 1372
W 1373
X 1373
Y 1373
Z 1373

1 Getting Started

MultiGen® Creator produces realistic three-dimensional models and terrain for use in realtime applications. Creator includes an integrated set of powerful tools for building hierarchical visual databases in a “what you see is what you get” (WYSIWYG) environment. These databases conform to the *OpenFlight*® format for general purpose modeling.

Creator’s comprehensive set of tools is organized to focus on both creating and editing models and databases. Once a database is constructed, its hierarchy can be quickly rearranged and whole groups of elements can be modified. Individual elements can be isolated for precise editing. Elements such as lights and textures are grouped and saved in palettes for easy reuse. External conversion utilities are included or available through third-party vendors so you can import models saved in other file formats, such as those used by popular animation and CAD applications.

System Requirements

Creator runs on standard PC workstations with Windows 2000 Professional, or Windows XP Professional operating system, and on SGI workstations running IRIX 6.5 or later. The following lists the system configurations for each operating system.

Windows

- Pentium III or IV PC workstation
- 500 MHz CPU clock speed or higher
- Windows 2000 Professional, Windows XP Professional
- 128 MB memory or higher
- 4 GB or greater hard drive
- OpenGL-compatible 3D accelerator card with 64 MB texture RAM or higher
- 1024 x 768 or higher screen resolution
- CD-ROM drive

SGI IRIX

- Any OpenGL-based SGI IRIX workstation
- IRIX 6.5 or higher (Indigo2 IMPACT, O2, Octane, Onyx2 or later)

- 250 MHz MIPS CPU clock speed or higher
- 128 MB memory or higher
- 4 GB or greater hard drive
- 1024 x 768 or higher screen resolution
- CD-ROM drive

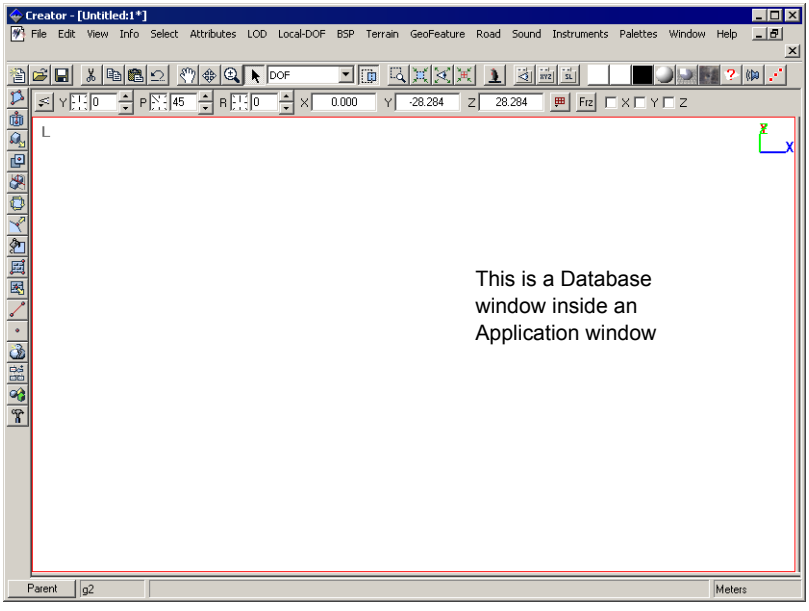
Starting Creator

You must install Creator on your hard disk before you can run it. You cannot run Creator from a CD-ROM. See the *Installation Guide* for information about installing Creator.

To start Creator in Windows, choose **Start/Programs/Multigen-Paradigm/Creator 2.6/Creator**. To start Creator on IRIX, open a shell window and type `creator`.

Creator searches for the `creator1.prefs` file in the Creator directory. This file contains Creator's default window settings

An application window opens with a new Database window.



Exiting Creator

To exit Creator, choose **File/Exit** (Ctrl+Q). For each open database file containing unsaved changes, a dialog box appears to prompt you to save your work.

Windows only: You can also exit Creator by clicking the Application window's standard **Close** button or by double-clicking its **Program** menu icon. (Program menu is available on Windows systems only.)



Version Information

To see Creator's version number and other information, choose **Help/On Version**.

License Information

On Windows, Creator runs with either an individual or a network license; individual licenses require dongles. On IRIX systems, Creator runs with either an individual or a network license.

To see which features are available with your current license, choose **Info/License**. If no license is available, Creator will run in read-only mode. For complete information on setting up the Creator license on your platform, see the *Getting Started Guide* included with Creator.

Working With Network License Configurations

When you start Creator with a network license, it retrieves—or checks out—a particular license configuration from the license manager on the network. To see which configurations are available on your network, choose **Info/License** to open the *License Manager* window.

The *License Manager* window shows a list of the license configurations available on your network and the maximum number of users who can check out each configuration simultaneously. The configurations are listed in descending order of priority, from your first choice at the top to your last choice at the bottom. You can change a configuration's priority by selecting it in the list and using the **Increase** and **Decrease** buttons to move it up or down. A new top-priority license does not take effect until you restart Creator. License configuration priority lists are saved with your preferences.

When you start Creator, the software attempts to check out your highest-priority configuration. If that is unavailable, one of three things happens:

- If **Use First Available** is selected in the *License Manager* window, then a dialog box appears informing you that the next available configuration will be used.
- If **Prompt When First Choice is Not Available** is selected in the *License Manager* window, then you are prompted to select an available configuration from the *License Manager* window's list.
- If no configurations are available, then Creator opens in read-only mode.

Listing Other License Configuration Users

To see a list of users on the network that have a license configuration checked out, choose **Info/Display Users**. Select an entry in the list to see which license configuration that user has checked out.

Creator Extras

Ways to extend Creator include the following:

- Using the Creator file conversion utilities to exchange image files with other applications. The utilities are located in the directory where the Creator executable file is installed. See “Conversion Utilities and Filters” on page 1205 for a list of conversion utilities and instructions for using them.
- Adding options such as Terrain or Culture processing.
- Using the Creator *application programming interface* (API) to create plug-in utilities that can be accessed from the Creator application.
- Installing plug-ins to add new tools and capabilities to Creator.

Contact MultiGen-Paradigm for more information about adding options to Creator.

Model Libraries and the Partners CD

The Creator CDROM provides sample OpenFlight database files, textures, and 3D fonts for your use. These files are installed in `\MultiGen-Paradigm\resources\gallery` (Windows) or `/usr/local/MultiGen-Paradigm/resources/gallery` (IRIX) when you choose the default installation.

MultiGen-Paradigm also provides a number of tools created by third parties on a Partners CD. Most tools contain document or readme files. Note that items on the Partners CD are not tested or supported by MultiGen-Paradigm, and you should contact the vendor if you are experiencing problems.

Learning About Creator

For quick reference while using Creator, press F1 and then click the tool, window, or dialog box you are using to get context-sensitive online help.

Creator also includes three printed books: *Creating Models for Simulations* presents general concepts that are important for you to know when building models for realtime display, *Creating Terrain for Simulations* describes the additional information you need to create terrain databases, and *The Desktop Tutor* provides step-by-step lessons in modeling and terrain generation. These books are also available in Acrobat (PDF) format on the Creator CD.

Document Conventions

The following sections explain the typographical conventions found in Creator documentation and online Help.

Menus

Paths to menu items use a forward slash (/) to separate menu names from menu items. For example, "Choose **File/Save**" means to click the **File** menu to open it, and then click the item **Save**. Sometimes there are submenus in the path to a menu item. For example, "Choose **View/Hidden Faces/Hide**" means to click the **View** menu to open it, point to the **Hidden Faces** submenu to open it, and click **Hide** in the **Hidden Faces** submenu.

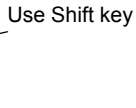
Keyboard Shortcuts

Many Creator functions can be activated either with the mouse or by pressing keyboard shortcuts. In all Creator instructions, the keyboard shortcut that activates a function appears immediately after the function name. For example, "Choose **File/New** (Ctrl+N)" means that you can create a new database file either by choosing **New** from the **File** menu or by holding down the Ctrl key and pressing "N".

For a complete list of keyboard shortcuts, see "Keyboard Shortcuts" on page 895, or choose **Help/Shortcuts**.

Although all keyboard shortcuts are shown in uppercase, do not hold down the Shift key when using keyboard shortcuts unless the Shift key is explicitly included in the shortcut.

Action	Shortcut
Toggle Graphics View/Hierarchy View	S
Reset View	U
Fit On Select	V
Fit On Grid	Ctrl+Shift+V
Center Select	C
Fit On Pick	Ctrl+F



Most keyboard shortcuts only work when the Graphics view has focus. If you make a selection in the Hierarchy view, you can right-click in the Graphics view to give this view focus without deselecting nodes in the Hierarchy view.

Exercises and Tutorials

The *Desktop Tutor* provided with Creator contains lessons that you can follow to learn how to use many of the features of Creator. This online help contains additional exercises. Most of these exercises appear as the last entry in each category on the Help Contents page.

Online Help

Online help is available on most of Creator's controls, menu items, and plug-ins. You can get help from the Help menu, by pressing F1 for context-sensitive help, and by using Tool Tips.

The Help Menu

The Creator **Help** menu contains the following help options:

Help

Starts context-sensitive help by changing the mouse pointer to a pointer-question mark combination. Click the mouse on any part of the interface to display its help topic.

Topics

Opens the *Help Topics* window containing all of the help topics available for Creator. You can click the **Contents** panel to select a section, or book, of the online help, the **Index** panel to locate a topic by its index entry, or the **Find** panel to search for a word in a help topic.

Shortcuts

Displays the keyboard shortcuts available in Creator.

On Plug-Ins

Opens a selection window listing plug-in modules and tools. This menu item is unavailable if Creator does not load any plug-ins.

On Version

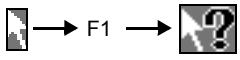
Displays information about the Creator version and license.

About Creator

Displays the splash screen that you see when Creator starts up.

Context-Sensitive Help

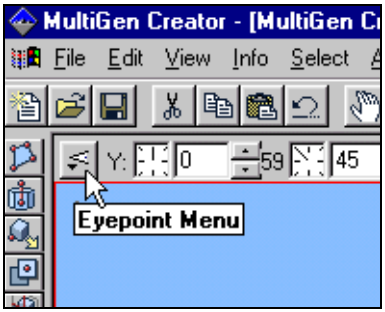
To get context-sensitive help on any Creator control, choose **Help/Help** or press F1 to change the pointer to a pointer-question mark combination, and then click on the control.



To get help on a menu item, click the menu, point to the menu item, and press F1.

Tool Tips

Tool tips are short messages that appear automatically when the mouse pointer comes to rest on a tool or control. Usually, tool tips show the names of the tools or controls that generate them.



Getting Technical Support

MultiGen-Paradigm Inc. provides technical support to customers who are part of our software maintenance program.

Before you contact technical support, make sure you are at your computer and have the following information:

- The Creator version number (choose **Help/On Version** in Creator)
- A brief description of the problem and the steps needed to re-create it
- A brief description of what you are trying to accomplish
- An exact description of any error messages that have appeared
- Your hardware specifications, including PC model, type of CPU, total RAM, size of hard disk, type of graphics card (OpenGL compliant?), and type of monitor
- Your telephone and fax numbers, and your e-mail address

You can reach technical support by telephone or e-mail:

- toll free (877) 289-5670 or (408) 965-8525
- creator@multigen-paradigm.com (e-mail)

The Technical Support Department is open between the hours of 8:00 a.m. and 5:00 p.m. (PST), Monday-Friday.

Answers to many of your questions can also be found by visiting the MultiGen-Paradigm World Wide Web site:

`http://www.multigen-paradigm.com`.

2 Customizing Creator

You can customize Creator to make it work better with your particular needs and constraints by setting environment variables and preferences. Environment variables specify default paths for certain Creator operations. Preferences control the behavior of many features of Creator. The following sections describe Creator's environment variables and preferences, and explains how to change them.

Environment Variables

Environment variables are values that control various aspects of how the system and applications run. For example, some environment variables set the path that an application uses to look for specific types of files.

You can set the following Creator environment variables to control how Creator uses files. Environment variables must be set before starting Creator in order for them to take effect.

MPI_LOCATE_CREATOR

Sets the directory path to the Creator executable files. By default this is set to `/Multigen-Paradigm` (Windows) or `/usr/local/MultiGen-Paradigm` (IRIX). If you installed Creator in a different location you must set this variable.

MPI_CREATOR_PLUGINDIR

Sets a default directory path for plugins. This is the first place Creator will search for plugins. If no plugins are found, Creator looks for a `\plugins` directory in the same directory as the Creator application, and if none is found, it then looks in the `\MultiGen-Paradigm` directory.

MGLMSERVER (IRIX only)

Sets the name of the machine that serves as the Creator license server node.

MGKEYDIR (IRIX only)

Sets the path to the Creator license key directory.

MPI_CREATOR_FLTEXTERNPATH

Sets the default directory path to externally-referenced files.

MPI_CREATOR_FLTPATH

Sets the default directory path for the **File/Open** command. Usually set to the directory where models are stored. You can specify multiple directories for Creator to search.

MPI_CREATOR_TERPATH

Sets the default directory path for the **Terrain/Import Elevation** command. Usually set to the directory where terrain data (.ded files) are stored.

MPI_CREATOR_RESOURCEPATH

Sets the default directory to palette files, such as color and material, that Creator loads when it starts up. If MPI_CREATOR_RESOURCEPATH is undefined, Creator searches the working directory. If no palette files are found, Creator loads standard Color and Material palettes from source definitions.

MPI_CREATOR_TXTPATH

Sets a default directory path to texture files. You can specify more than one directory for Creator to search. (See “Texture Palette Files” on page 441.) This becomes the default directory for the Texture palette.

MPI_CREATOR_FONTPATH

Sets the path to the 3D font directory.

MPI_CREATOR_NOSPLASH

Shows or hides the Creator splash screen on startup. If you are developing and debugging plug-ins, for example, you may want to hide the splash screen when debugging the startup code.

MPI_CREATOR_SOUNDEDITOR (IRIX only)

Associates a third party sound editor with Creator. Choosing **Edit/Edit Sound** automatically launches the sound editor.

GVIEWDFADPREFS

If GView is installed, sets the absolute path and file name of the `dfad.prefs` file that GView should use. GView requires the presence of a `dfad.prefs` file in order to load. The GVIEWDFADPREFS environment variable lets you specify a path to search for this file, and a file name, if it the name is not `dfad.prefs`. See the *GView User's Guide* for more information about the GVIEWDFADPREFS variable.

To Set Environment Variables in Windows NT

Step 1. Log in to Windows as Administrator if you want your environment variable settings to be effective for all users on the system. Log in as a user with administrative privileges if you want the settings to apply only to your user account.

If you do not have Administrator privileges, contact your system administrator for the Administrator password, or for a temporary Administrator account.

Step 2. Choose **Start/Settings/Control Panel** to open the *Control Panel* window, and then double-click the **System** icon to open the *System Properties* window.

Step 3. Click the **Environment** tab.

Step 4. In the **Variable** field, enter the name of the environment variable you want to set. For a list of Creator environment variables, see "Environment Variables" on page 43.

Step 5. In the **Value** field, enter the directory path assigned to that variable.

Step 6. Click **Set**.

Step 7. To define another environment variable, return to Step 4.

Step 8. Click **OK** to close the *System Properties* window when you are finished.

Step 9. Restart Creator.

To Set Environment Variables in Windows 2000

Step 1. Log in to Windows as Administrator if you want your environment variable settings to be effective for all users on the system. Log in as a user with administrative privileges if you want the settings to apply only to your user account.

If you do not have Administrator privileges, contact your system administrator for the Administrator password, or for a temporary Administrator account.

Step 2. Choose **Start/Settings/Control Panel** to open the *Control Panel* window, and then double-click the **System** icon to open the *System Properties* window.

Step 3. Click the **Advanced** tab, and then click the **Environment** button.

Step 4. In the **System Variables** area, click **New**. The New System Variable dialog box appears.

Step 5. In the **Variable Name** field, enter the name of the environment variable you want to set. For a list of Creator environment variables, see "Environment Variables" on page 43.

Step 6. In the **Variable Value** field, enter the directory path assigned to that variable and then click OK.

Step 7. To define another environment variable, return to Step 4.

Step 8. When you are finished, click **OK** to close the Environment Variables window and then click **OK** again to close the *S*ystem Properties window.

Step 9. Restart Creator.

To Set Environment Variables in Windows XP Professional

Step 1. Log in to Windows as Administrator if you want your environment variable settings to be effective for all users on the system. Log in as a user with administrative privileges if you want the settings to apply only to your user account.

If you do not have Administrator privileges, contact your system administrator for the Administrator password, or for a temporary Administrator account.

Step 2. Choose **Start/Settings/Control Panel** to open the *Control Panel* window, and then double-click the **System** icon to open the *System Properties* window.

Step 3. Click the **Advanced** tab, and then click the **Environment Variables** button.

Step 4. In the **System Variables** area, click **New**. The New System Variable dialog box appears.

Step 5. In the **Variable name** field, enter the name of the environment variable you want to set. For a list of Creator environment variables, see "Environment Variables" on page 43.

Step 6. In the **Variable value** field, enter the directory path assigned to that variable and then click OK.

Step 7. To define another environment variable, return to Step 4.

Step 8. When you are finished, click **OK** to close the Environment Variables window and then click **OK** again to close the *S*ystem Properties window.

Step 9. Restart Creator.

To Set Environment Variables in IRIX

Creator for IRIX includes a shell script, `creator.env`, that sets environment variables. You can customize this script and either paste it into your `.login` file (recommended) or source it just before you start up Creator.

To customize `creator.env`:

Step 1. Open a shell window.

Step 2. Change to the Creator home directory (`usually /usr/local/MultiGen-Paradigm/creator`) and open `creator.env` in a text editor.

The following file appears (the example below shows only part of the file):

```
#! /bin/csh
#
#*************************************************
#
#   Creator Environment Variables
#
#   This file contains the environmental variables
#   used by MultiGen Creator on Irix.
#
#*************************************************

# MultiGen Creator Install Root.
# This defines directory where you installed
# MultiGen Creator. Most other variables defined
# in this file depend on this one.  So, if you
# installed Creator in some other location, this
# is the only variable you must change.
#
setenv MGHOME /usr/local/multigen

if ( ! ($?LD_LIBRARYN32_PATH) ) then
setenv LD_LIBRARYN32_PATH $MGHOME/creator/wulib\:$MGHOME/creator
else if ( $LD_LIBRARYN32_PATH !~ *$MGHOME/creator* ) then
setenv LD_LIBRARYN32_PATH $MGHOME/creator/wulib\:$MGHOME/
```

Step 3. Highlight the text in the file

Step 4. Open another shell window.

Step 5. In a text editor, open the `.login` file in your home directory.

Step 6. Place the cursor where you want to insert text, and middle-click to paste the contents of the `creator.env` file into the `.login` file.

Step 7. Save the `.login` file.

Step 8. Log off your user session and then log back on again to reload your `.login` file.

Preferences

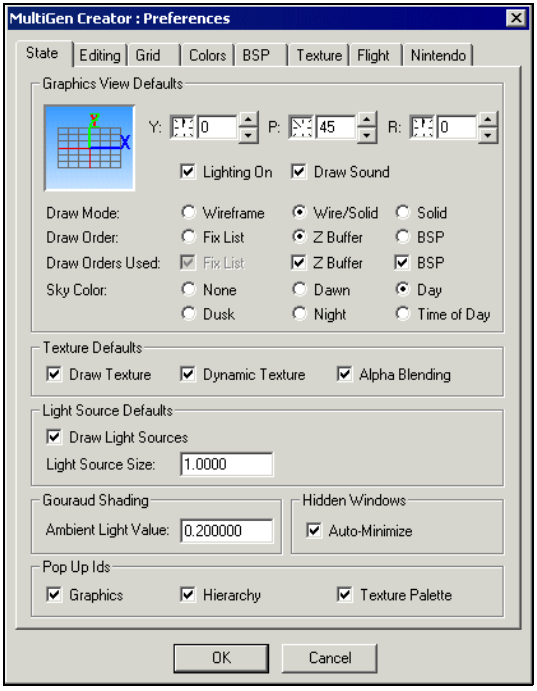
You can customize many aspects of Creator by setting preferences in the Creator *Preferences* window. This window has panels for setting State preferences, Editing preferences, Grid preferences, Color preferences, BSP preferences, Texture preferences, Flight preferences, and Nintendo preferences. Preferences panel settings are saved in the file `creator1.prefs`.

To open the *Preferences* window, choose **Info/Preferences** (Shift+Ctrl+P) and click the tab for a specific preferences panel. For example, click the **Flight** tab to display Flight preferences.

New preference settings are saved and take effect immediately after you click **OK** in the *Preferences* window. New settings are discarded if you click **Cancel** or **Close**.

State Preferences

State preferences (**Info/Preferences** (Shift+Ctrl+P) and click the **State** tab) control the default setting for the Graphics view, including when and how to draw textures, light sources, and Gouraud shading. These settings are applied when you open a new file.



Default Eyepoint



Sets the eyepoint position thatis used when a Graphics view is first opened and when you choose **View /Reset** (U) or click **Reset View** in the Toolbar.

See “Understanding the Default Eyepoint Position” on page 262 for more information.

Lighting On

Controls the default lighting in the Graphics view. Setting this checkbox enables dynamic lighting. See “Setting Lighting Preferences” on page 618.

Draw Mode

Controls the default display of polygons in the Graphics view. In **Wire** mode, only the colored edges of polygons are displayed. In **Wire/Solid** mode, polygons are filled with their assigned color, while edges appear in black. In **Solid** mode, polygons are filled with their assigned color, but edges are not displayed.

See “Draw Mode” on page 321 for more information.

Draw Order

Controls the default drawing algorithm in the Graphics view. See “Draw Order” on page 682.

Draw Orders Used

Sets the sequence of available drawing orders. You can change the drawing order by pressing Z in the Graphics view. See “Draw Mode” on page 321.

Sky Color

Selects the default background sky color in the Graphics view.

See “Setting Sky Color” on page 269 for more information.

Draw Sound

Enables the display of sound icons that indicate the position of sound nodes in the Graphics view. See “Sound Nodes” on page 672.

Draw Texture

Sets the default mode to display applied textures in the Graphics view. See “Displaying Texture in the Database” on page 521.

Dynamic Texture

Sets the default mode to dynamic texture drawing in the Graphics view. In dynamic texture drawing mode, texture drawing remains on when the eyepoint is in motion. (By comparison, with **Draw Texture**, texture drawing is temporarily switched off when the eyepoint is in motion, which reduces the drawing overhead.) See “Displaying Texture in the Database” on page 521.

Alpha Blending

Turns on alpha blending drawing mode by default in the Graphics view. In this mode, textures that contain an alpha (transparency) component are blended with the face that is behind the face to which the texture is applied. See “Displaying Texture in the Database” on page 521.

Draw Light Sources

Enables the display of markers that represent the positions of light source nodes in the Graphics view. See “Setting Lighting Preferences” on page 618.

Light Source Size

Controls the size of the markers that are displayed when **Draw Light Sources** is set. See “Setting Lighting Preferences” on page 618.

Ambient Light Value

Sets a background intensity that affects the amount of ambient light applied to surfaces when Gouraud shading is applied. See “Setting Lighting Preferences” on page 618.

Pop Up Ids

Determines which views display pop-up windows with information about items over which your mouse is placed. Pop Up Ids are displayed in the Graphics and Hierarchy views of the database window, as well as in the Texture Palette, based upon which options you select here.

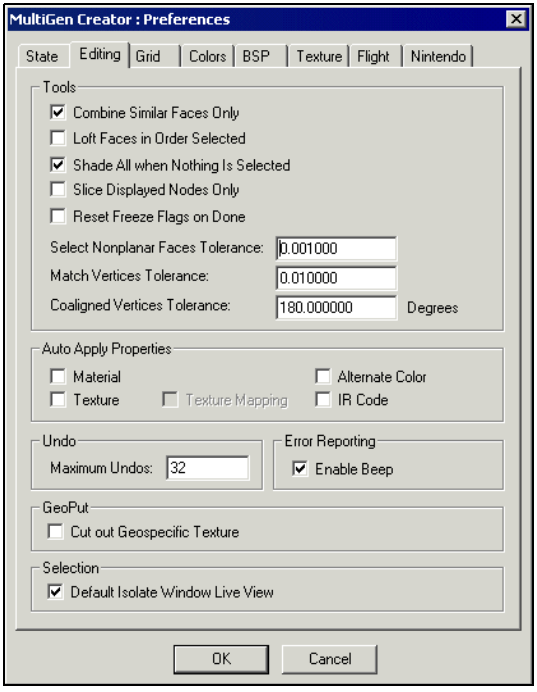
Within the Graphics and Hierarchy views, the pop-up window is displayed when you move the mouse over a node in the scene; it contains the long id of that node. This feature is useful in the Graphics view to confirm which node you are selecting if you click the mouse at its current position. This feature is useful in the Hierarchy view to display the full id of a node when its length is longer than the seven-character limit imposed by the ordinary Hierachy listing; it is also helpful as you zoom out in the Hierarchy view and the ids become smaller and harder to read.

In both the Graphics and Hierarchy views, the long ids of the nodes are displayed, as well as any attributes you have selected for display with this node using the **Attributes/Define ID Arributes** command.

Within the Texture Palette, the pop-up window is displayed when you move the mouse over a texture in the palette; it contains the full name and path of that texture. This feature is useful if you want to know the name of a texture, but do not want to select it.

Editing Preferences

Control the default behavior of certain tools, determine when vertices are coplanar or share the same coordinates, set color and texture defaults for newly-created faces, and specify the number of recent operations that you can Undo. To view Editing preferences, choose **Info/Preferences** (Shift+Ctrl+P) and click the **Editing** tab.



Combine Similar Faces Only

Controls the **Combine Faces** tool. If this checkbox is set, only faces with the same color and texture are combined. Otherwise, faces are combined without regard to color and texture. See “Combine Faces Tool” on page 189.

Loft Faces in Order Selected

Controls the **Loft** tool. If this checkbox is set faces are lofted in the order they appear on the Select list. Otherwise, faces are assumed to be parallel. See “Loft Tool” on page 115.

Shade All When Nothing is Selected

Controls **Calculate Shading**. If this checkbox is set, the entire database is shaded when you choose **Calculate Shading** with nothing selected. Otherwise, **Calculate Shading** has no effect when nothing is selected. See "Database Shading" on page 620.

Slice Displayed Nodes Only

Controls the **Slice** tool. If this checkbox is set, **Slice** only processes nodes that are displayed in the Graphics view. Otherwise, **Slice** processes all selected nodes, regardless of their display. See "Slice Tool" on page 174.

Reset Freeze Flags on Done

Automatically clears freeze flags when an operation is complete. See "Freeze Flags" on page 77.

Select Nonplanar Faces Tolerance

Specifies a maximum distance that is allowed between vertices that are considered to lie in the same plane. You can specify a value between 0 and 1.0. If you specify a value less than 0, all faces (including planar) will be considered non-planar. If you specify a value greater than 1.0, no faces will be considered planar.

Creator calculates a plane from the last three vertices in a face. Other vertices in the same face cannot be any farther from the surface of the plane than the distance set by this preference, or they are considered to be nonplanar. See "Select Nonplanar Faces" on page 334.

The **Combine Faces** tool uses the **Select Nonplanar Faces Tolerance** value to check for coplanarity before merging selected faces that share edges. See "Combine Faces Tool" on page 189.

Match Vertices Tolerance

Specifies the maximum distance that is allowed between two vertices that are considered to share the same coordinate. See "Match Vertices Tool" on page 202.

Coaligned Vertices Tolerance

Specifies the maximum distance that is allowed between three or more vertices that share the same linear sequence.

Auto Apply Current Material

Applies the Current material to each newly created face in the database. If this preference is cleared, a material index of –1 (no material) is assigned to each new face. See “Applying Material Properties” on page 394.

Auto Apply Texture and Mapping

Applies the Current texture pattern and texture mapping to each newly created face in the database. If this preference is cleared, a texture index of -1 (no texture) is assigned to each new face. See “Displaying Texture in the Database” on page 521.

Auto Apply Alternate Color

Applies the Current alternate color to intensity texture patterns in the database. If this preference is cleared, a color index of -1 (no color) is assigned. See “Primary Color and Alternate Color” on page 401.

Auto Apply IR Code

Applies the Current IR code to each newly-created face in the database. If this preference is cleared, an IR code of 0 (black) is assigned to each new face. See “Infrared Color” on page 415.

Max Undos

Specifies the number of edits that are saved in the **Undo** list. See “Undoing Work” on page 244.

Cut out Geospecific Texture

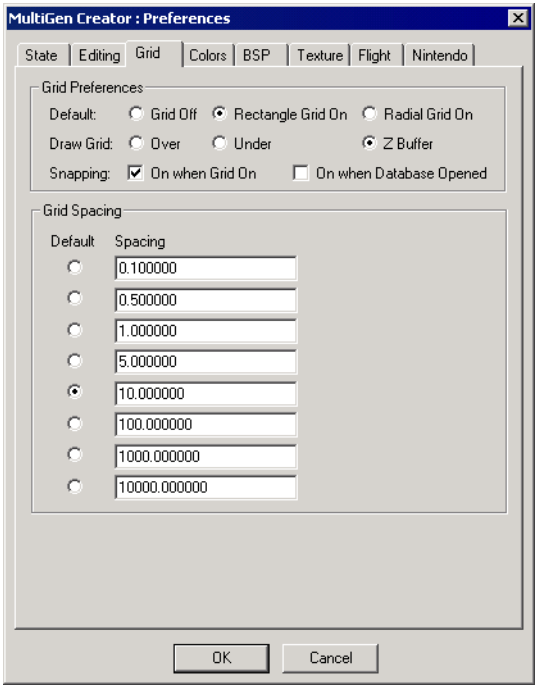
Automatically applies a geospecific texture only over the area determined by its geospecific coordinates. See “Applying a Geospecific Texture” on page 1191 for complete information.

Default Isolate Window Live View

Sets the **Live** button to be on by default each time the Isolate window is active. See “The Live Button” on page 332.

Grid Preferences

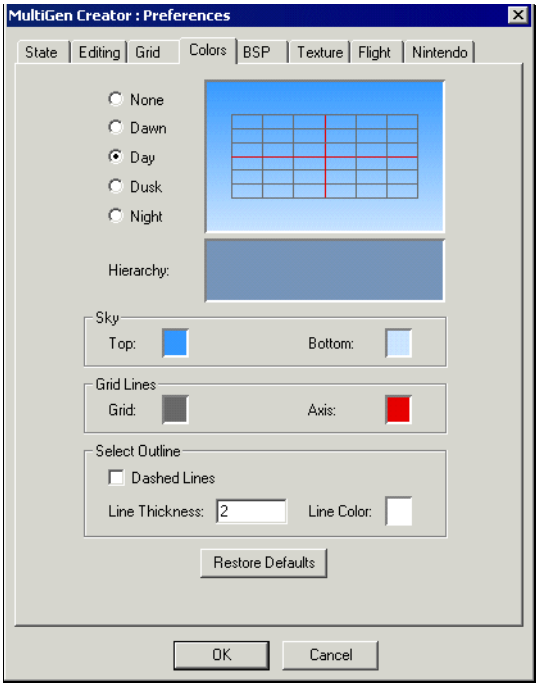
Control the appearance and size of the grid when you open a new file. To view Grid preferences, choose **Info/Preferences** (Shift+Ctrl+P) and click the **Grid** tab.



For a description of each preference in this panel, see “Grid Preferences” on page 56.

Color Preferences

Control the default colors for the sky at various times of day, the Hierarchy view background, grid, and selection lines, and selection line type.



Sky Color

Chooses the default sky color that each new file will display when created. See “Setting Sky Color” on page 269.

Hierarchy

Displays the Hierarchy view’s background color. See “Changing the Hierarchy View Color” on page 284.

Sky

Displays the top and bottom sky colors at the time of day indicated by the option buttons None, Dawn, Day, Dusk, Night, and Time of Day. Clicking the Top or Bottom color sample opens the Color palette so you can choose a new color. See “Changing Default Sky Colors” on page 269.

Grid Lines

Displays the current default grid and axis colors. See “Setting Default Grid Colors” on page 305.

Select Outline

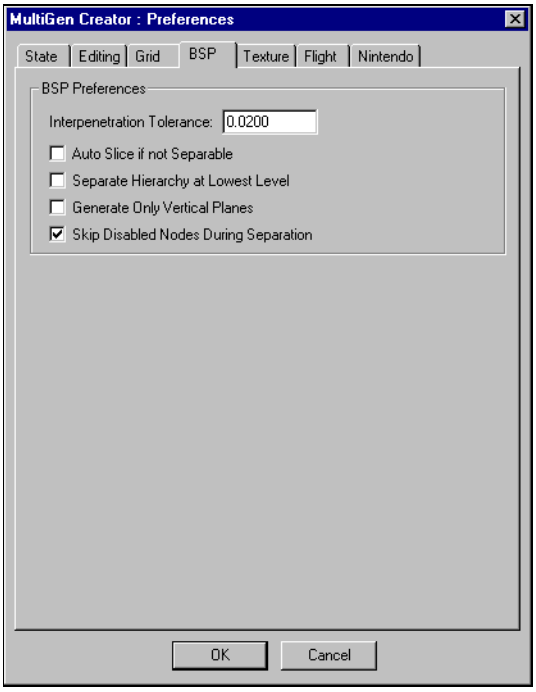
Displays the current line type, line thickness, and line color for highlighting a selection. See “Customizing Selection Highlights” on page 328.

Restore Defaults

Restores the settings in the Colors panel to their original default values.

BSP Preferences

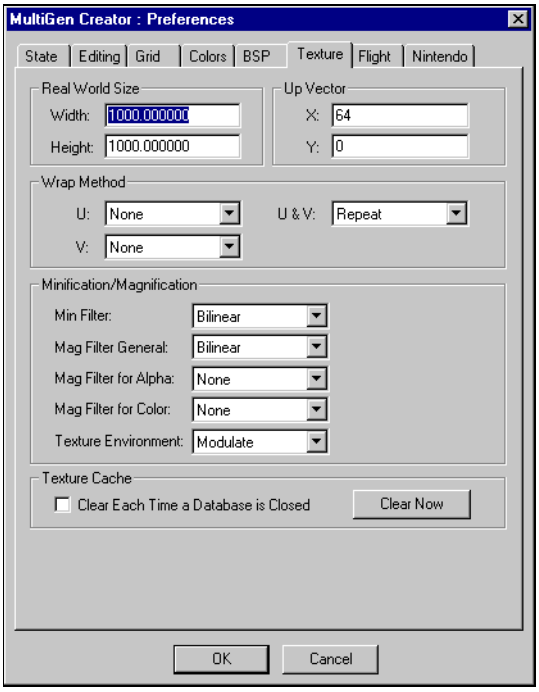
Control the way the database is separated when using the BSP option. To view BSP preferences (when this option is installed), choose **Info/Preferences** (Shift+Ctrl+P) and click the **BSP** tab.



For information, see “Setting BSP Preferences” on page 913.

Texture Preferences

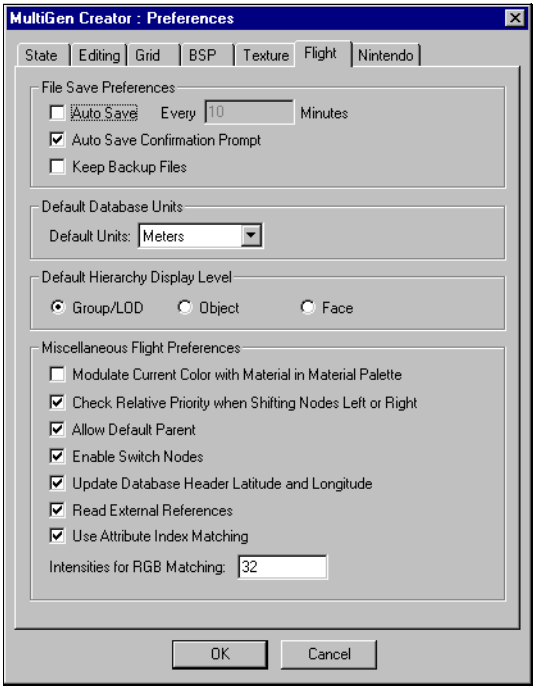
Sets defaults for how texture is sized and applied. To view Texture preferences, choose **Info/Preferences** (Shift+Ctrl+P) and click the **Texture** tab.



For information, see “Texture Variables and Preferences” on page 541.

Flight Preferences

The Flight Preferences Panel contains settings that control the behavior of modeling functions in Creator. Choose **Info/Preferences** (Ctrl+Shift+P) and click the **Flight** tab to view and edit these Flight preferences.



Auto Save Every k Minutes

Enables automatic database backup, and sets the number of minutes between saves. Creator checks the system clock *after each database edit* and saves each open file that has changed. Database files are not saved when the program is idle.

Auto Save Confirmation Prompt

Prints a message in the Status bar at the bottom of the Database window each time an automatic save is performed.

Keep Backup Files

Makes a backup copy (using the extension `.bak`) the *first time you save a file after you open it*. The purpose of this backup is to preserve a copy of your database at the beginning of each editing session. You must make a change to the database before Creator can save it; it is recommended that you make an insignificant edit and then save at the beginning of each session if you want to use this feature.

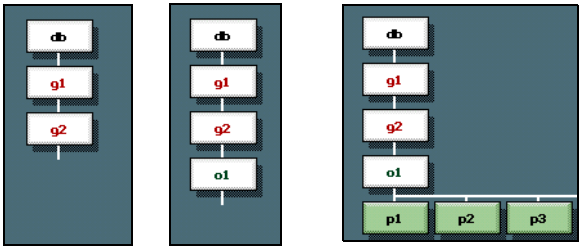
The backup copy will not be updated when you save new changes to the file during the current editing session. The next time the backup copy will be overwritten is after you close the file, reopen it, and then save it.

Default Database Units

Selects the default unit of measure for each new database. This is stored in the **Units** attribute of the database header node (see "Header Attributes" on page 831). When a database is created or opened, the unit of measure is displayed on the status bar, at the bottom right corner of the database window.

Hierarchy Display Level

Sets the default depth of the Hierarchy view to **Group/LOD**, **Object** or **Face**. For example, if **Object** is selected, Creator displays the hierarchy down to object nodes each time you open a new Hierarchy view.



Modulate Current Color with Material in Material Palette

Blends the Current material display (in the Material palette) with the Current color (see "Modifying Material Properties" on page 390).

Check Relative Priority When Shifting Nodes Left or Right

Restricts the effect of **Shift Right**, **Shift Far Right**, **Shift Left**, and **Shift Far Left** in the **Hierarchy** tool palette. When this checkbox is set, you cannot move a node to the left of a sibling with a lower **Relative Priority** attribute, or to the right of a sibling with a higher **Relative Priority** attribute. Database hierarchy is periodically sorted on the basis of relative priority (see "Relative Priority" on page 688).

Allow Default Parent

Enables automatic parenting in the database. Creator normally requires that a node be selected as the database parent (the point of attachment in the hierarchy) before performing an operation requiring a parent (see "Setting the Parent" on page 291). In automatic parenting mode, Creator uses a default group node as the database parent if a valid parent node is not selected.

The first time a default parent is required, Creator creates a group, usually named **Default**, and attaches it to the database header. This node continues to be the default parent, even if you move it to another point in the hierarchy. If you delete the default parent node, it is recreated the next time it is needed. If the ID **Default** is already in use when the default parent node is created, an arbitrary group **ID** is used for the default parent.

If **Allow Default Parent** is cleared, Creator does not perform operations requiring a parent unless a valid parent is specified.

Enable Switch Nodes

Turns on switch-node drawing when a database is first opened. If this preference is cleared, all the children of a switch node are displayed, regardless of the masks it contains, until the first time you edit the switch-node attributes (see "Switch Nodes and Switch Attributes" on page 861).

Update Database Header Latitude and Longitude

Includes the area of externally referenced files in the latitude/longitude extents of a database (stored in the **SW Corner** and **NE Corner** header attributes). If this checkbox is cleared, the values displayed in the *Header Attributes* window reflect the area of the database but not of any external files that are referenced (see "Header Attributes" on page 831).

Read External References

Enables the automatic loading and display of external references. If this checkbox is cleared, external references are loaded and displayed manually (see "External References" on page 370).

Use Attribute Index Matching

Enables color and material matching when elements are transferred from the Graphics buffer to a database during Copy/Paste, Populate, Scatter, and Library Substitution

operations. When **Use Attribute Index Matching** is set, Creator checks the Color palette for the closest possible RGBA match for each face and adjusts the palette indices for the pasted faces as needed. It also checks the Texture palette for identical file names. If a matching file name is found, the palette indices on the faces to which the texture is applied are changed accordingly; if a file name is not found, the palette indices of the affected faces are changed to -1 (no texture).

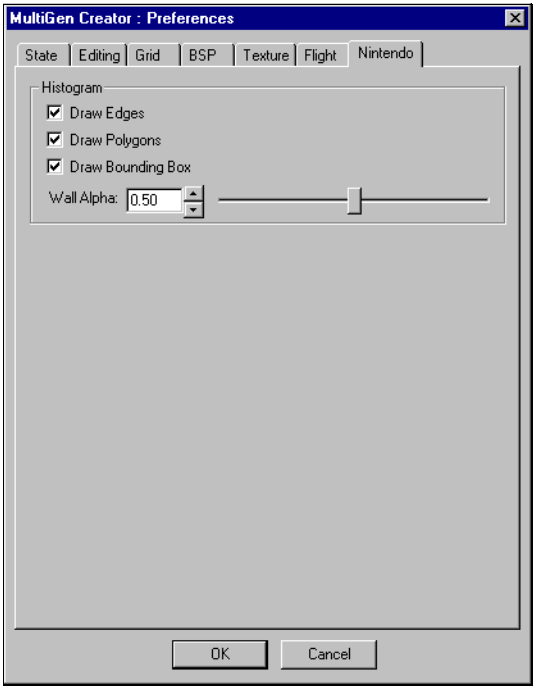
When this preference is cleared, Creator does not change the palette indices; pasted faces receive the color and texture that correspond to the original index numbers. This saves processing time if, for example, you are using a universal set of palettes, or if you intend to use the Attribute Resolver later in the project and do not care about interim matching (see "To Resolve Palette Inconsistencies" on page 880).

Intensities for RGB Matching

Controls the amount of searching done in the Color palette for color matching. Creator searches the Color palette for the best color match during functions such as Calculate Shading or when importing files from an older version of the OpenFlight format (see "Color Matching" on page 414).

Nintendo Preferences

Set default values for the Nintendo option. To view Nintendo preferences (when this option is installed), choose **Info/Preferences** (Shift+Ctrl+P) and click the **Nintendo** tab.



Draw Edges

Controls the display of wireframe edges around each volume calculated for the terrain histogram.

Draw Polygons

When set, causes the plateaus and sides of each volume calculated for the histogram to be displayed with solid polygons. The **Wall Alpha** parameter, can be used to apply transparency to the walls of the histogram.

Draw Bounding Box

Controls the display of a bounding box around the entire group. This box is for reference only and cannot be manually edited.

Wall Alpha

Controls the transparency of the sides of each volume block calculated for the bounding volume, if **Draw Polygons** is set. The transparency allows you to see the geometry underneath. If **Draw Polygons** is not set, volume walls are not displayed. **Wall Alpha** does not affect the appearance of terrain histogram plateaus.

Plug-ins

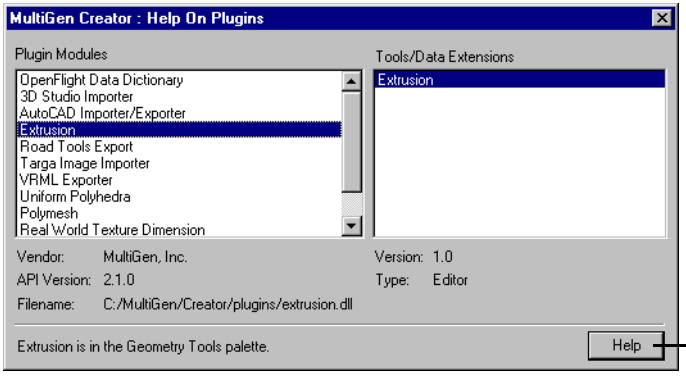
Creator allows you to extend its capabilities by installing plug-ins. Plug-ins can add tools to toolbars or add menu items to menus. Once they are installed, they are indistinguishable from the standard tools available in Creator. However, since some plug-ins are developed by third-party vendors, MultiGen cannot provide technical support for them.

This section shows you how to use **Help/On Plugins** and how to write your own plug-ins for Creator.

Using Help on Plugins

Plug-ins extend the capabilities of Creator. However, if a plug-in is provided by a third-party vendor, there may not be any information on its use in the Creator online help. You can still get some help on the plug-in by choosing **Help/On Plugins**.

This opens the *Help on Plugins* dialog box.



If a help file is available for the plug-in, the Help button will be enabled.

Click the name of the plug-in the **Plugin Modules** list. Information about the plug-in is shown in the lower part of the dialog box, including its file name and the vendor that created it.

In **Tools/Data Extensions**, click the name of the plug-in. Look in the Status bar of Creator for information about the location of the tool. For example, for the Extrusion tool, it says, "Extrusion is in the Geometry Tools palette."

If a Help topic *is* available for the plug-in, the **Help** button in the *Help on Plugins* dialog box will be enabled. Click this button to open the Help file.

Making Plug-ins

Programmers who are interested in writing their own plug-ins can do so using the Creator API. You can install the API using the Custom Install option. See the *MultiGen Creator Installation Guide and Release Notes* for instructions.

The Plug-In Wizard

A Plug-in Wizard is also provided with Creator (Windows only) as a plug-in. Its file name is `PluginWizard.dll`. To install this plug-in, drag it from the `\Free Stuff\Creator Plugins\PluginWizard` directory on the Creator CD to the `\plugins` directory within the directory where you installed Creator, and then restart Creator.

You can use the Plug-in Wizard to create a template for a Creator plug-in including all the source files needed for a plug-in module. Plug-ins created by the Plug-in Wizard can be built and loaded into Creator with no modifications. As with all files supplied in the `\Free Stuff` directory, this plug-in is supplied for your use with no support.

Creator Options and Product Levels

The Creator product line offers different options and levels that meet different scales of requirements to fulfill your particular simulation needs.

- The *Road Tools* option constructs synthetic roads that meet AASHTO standards, and which can be used for driving simulations, vehicle design, specialized driver

training, and accident re-enactment. See “Road Tools” on page 1231 for more information.

- The *CreatorPro* product level provides advanced terrain generation and adds vector editing, light point generation and instrumentation, among other features.
- The *Creator Terrain Bundle* product level provides large area terrain generation with integrated ground culture and global texture mapping.

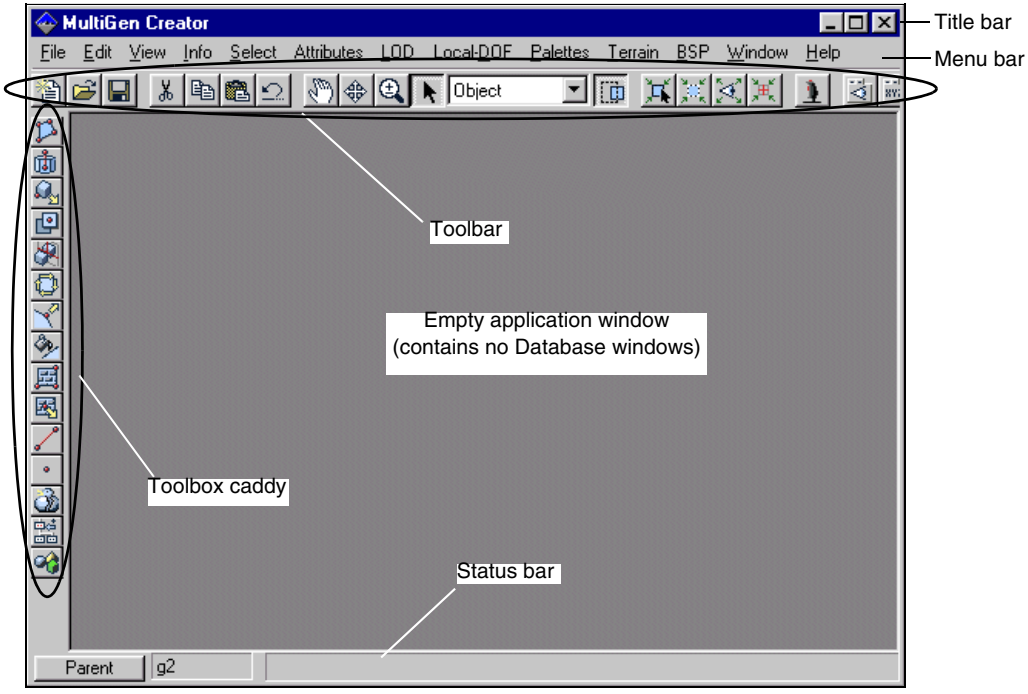
All of these product options and levels are also documented in a more general way in the books *The MultiGen Creator Modeler's Guide* and *Creating Terrain for Simulations*, which are included with Creator. *The Desktop Tutor*, which also is included with Creator, provides hands-on practice for most of the features of Creator, including its options. For complete information about the features of specific options and levels, contact your sales representative or send email to sales@multigen.com.

3 Windows and Views

The following topics describe Creator windows, the different views that are available within the database window, and how to switch between windows or to divide a database window into panes.

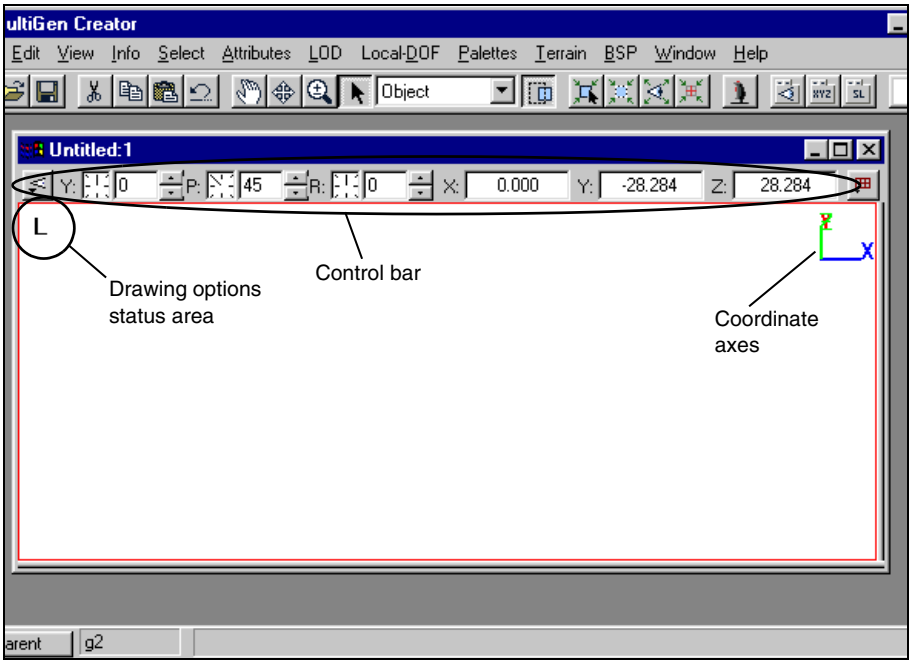
The Application Window

Creator's Application window has three rows of menus, buttons, and text-entry fields at the top: the *Title bar*, the *Menu bar*, and the *Toolbar*. A vertical set of toolboxes, the *Toolbox caddy*, appears on the left side of the window. A *Status bar* displays informational messages at the bottom of the Application window.



The Database Window

You use Creator's tools and controls to build, view, and edit 3D models in a *Database window*. When you first start Creator, the application window opens with a new Database window.



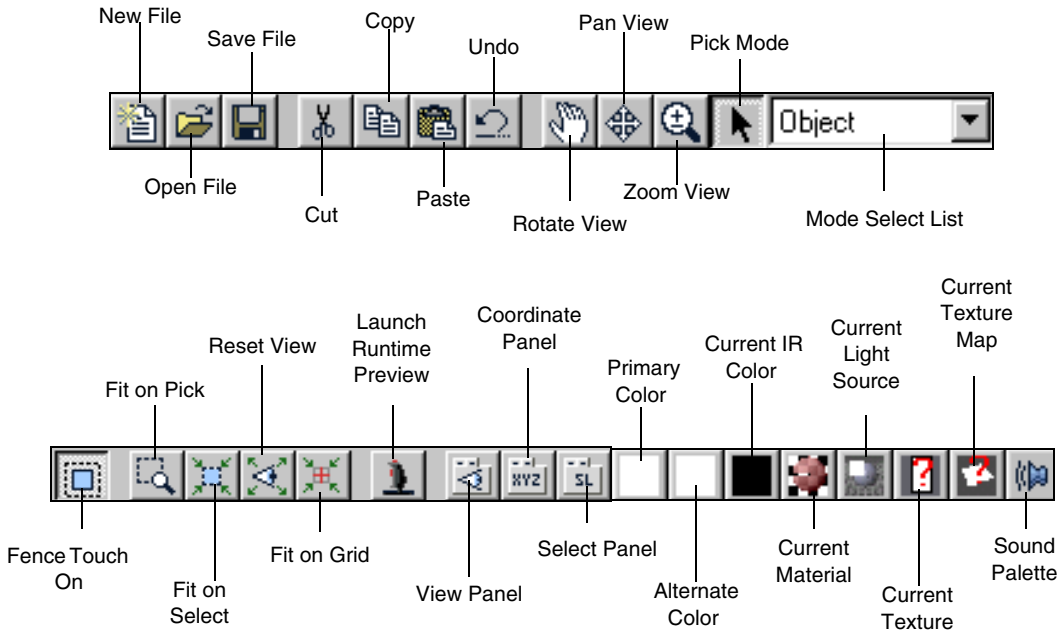
Each Database window has a *Control bar* with controls that affect only that Database window. In the upper-left corner of the Database window's display area is a status area that shows which drawing options are currently in effect. In the upper-right corner, a set of coordinate axes shows the viewer's orientation with respect to the database.

The Database window contains a representation of the contents of an *OpenFlight file*, sometimes called an *OpenFlight database*, or simply a *database file*. OpenFlight is the format used by Creator to save the models you create. A database file contains data describing a visual scene. This data consists of *geometry* (coordinate data), *hierarchy* (organizational data), and *attributes* (descriptive data).

You can open more than one database file in the Application window. Each file will appear in a separate Database window.

Toolbar Buttons

The Creator Toolbar, which is located just below the Menu bar, provides quick access to file operations, eyepoint controls, tool panels and palettes.



New File



Creates a new OpenFlight file. See “Creating New Database Files” on page 244.

Open File



Opens the *Open Database File* dialog box, which lists existing files. See “Opening Database Files” on page 242.

Save File



Saves the current database to disk. See “Saving Database Files” on page 244.

Cut



Removes the selected item and places it in the Text Buffer or the Graphics Buffer.

Copy



Copies selected items into the Text Buffer or the Graphics Buffer.

Paste



Inserts the contents of the Graphics Buffer under the current parent and starts **Translate** to position the new geometry, or inserts the contents of the Text Buffer at the cursor.

Undo



Reverses the last executed command. See “Undoing Work” on page 244.

Rotate View



Changes the eyepoint yaw, pitch and roll when you drag the mouse over the Graphics view. See “Adjusting The Eyepoint” on page 251.

Pan View



Moves the eyepoint up, down, left and right when you drag the mouse over the Graphics view. See “Panning in the Graphics View” on page 255.

Zoom View



Moves the eyepoint in and out when you drag the mouse over the Graphics view. See “Zooming in the Graphics View” on page 254.

Pick Mode



Allows you to select or fence items in the Graphics or Hierarchy views. See “Selecting” on page 327.

Mode Select List

Allows you to change Modeling modes without opening the mode toolbox. See “Selecting” on page 327.

Fence Touch On



When this control is on, anything touched by the selection fence becomes selected.

When this control is off, an item must be completely enclosed by a selection fence in order for it to become selected. See “Selecting” on page 327.

Fit on Pick (Ctrl+F)



Allows you to pick an item in the Graphics view, and then centers and scales the view on the item without selecting it.

See “View Options” on page 260 for more information.

Fit on Select



Centers the selected item and resizes the item to fill the Graphics view.

See “View Options” on page 260 for more information.

Reset View (U)



Restores the default viewing parameters and tracking plane set by the **Graphics View Defaults** State preferences. See “Understanding the Default Eyepoint Position” on page 262.

Fit on Grid (Ctrl+Shift+V)



Centers the eyepoint over the grid and moves the eyepoint in or out to make the grid fill the Graphics view. See “Changing the Center of Rotation” on page 257.

View Panel



Displays the View and Grid controls, which control the Graphics view display, including the grid. See “Grid and Trackplane Controls” on page 306 and “Drawing Controls” on page 271.

Coordinate Panel



Displays the Coordinate controls, which contains information about the coordinate points as they are selected or edited. See “The Coordinate Panel” on page 341.

Select Panel



Opens the Select List, which shows the ID of every item that is currently selected. See “Selecting” on page 327.

Primary Color



Displays the Current primary color and opens the Color palette. See “Primary Color and Alternate Color” on page 401.

Alternate Color



Displays the Current alternate color and opens the Color palette. See “Primary Color and Alternate Color” on page 401.

Current IR Color



Displays the Current infrared color and opens the IR palette. See “Infrared Color” on page 415.

Current Material



Displays the Current material and opens the Material palette. See “The Material Palette” on page 387.

Current Light Source



Displays the Current light source and opens the Light Source palette. See “Creating Light Sources” on page 602.

Current Texture



Displays the Current texture, if any, and opens the Texture palette. When there is no current texture, the icon contains a question mark. See “Texture Palettes” on page 422.

Current Texture Mapping



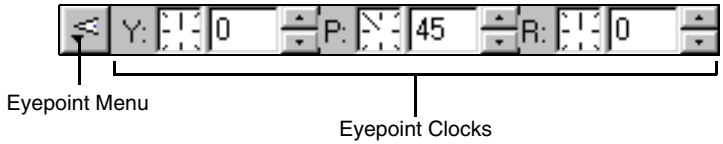
Opens the Texture Mapping palette. See “Texture Tools” on page 444.

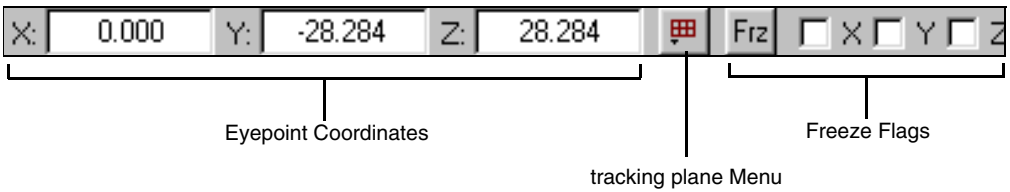
Sound Palette



Opens the Sound palette. See “The Sound Palette” on page 669.

Control Bar Features





The following sections describe the controls that appear just below the Toolbar in Creator.

Eyepoint Menu



Jumps to any of seven preset eyepoints. Stores up to nine additional custom eyepoints. See “Adjusting The Eyepoint” on page 251.

Eyepoint Clocks

Control the yaw, pitch and roll of the eyepoint around the eyepoint center of rotation. See “Clocks” on page 80.



Eyepoint Coordinates

Display the current eyepoint coordinates. See "Changing the Eyepoint Position" on page 251.



Tracking Plane Menu



Stores up to nine custom tracking planes for quick recall during modeling. See "To Store a Tracking Plane" on page 313.

Freeze Flags

Freeze the *x*, *y*, or *z* components of the current coordinate. See "To Store a Tracking Plane" on page 313.



Managing Database Windows

You can manage multiple Database windows with the standard commands found in the **Window** menu.

Window Menu

The **Window** menu contains the following commands:

Cascade (Shift+F5)

Resizes and staggers all maximized Database windows so that all of their title bars are visible.

Tile (Shift+F4)

Resizes and arranges all maximized Database windows so that all are visible.

Arrange Icons

Places the icons of all minimized Database windows next to each other.

Close All

Closes all open Database windows, and therefore, all open database files.

Switching Views

To switch between the Graphics and Hierarchy views, use one of the following methods:

- If the pane in which you want to change the view is not active, click in it. At times, you may want to make the view active without selecting anything in the database. To do so, right-click in the window.
- Choose **View/Graphics** (S) to change to the Graphics view, or **View/Hierarchy** (S) to change to the Hierarchy view.

Several other commands are available for switching the Graphics view to a more useful mode.

Zoom View

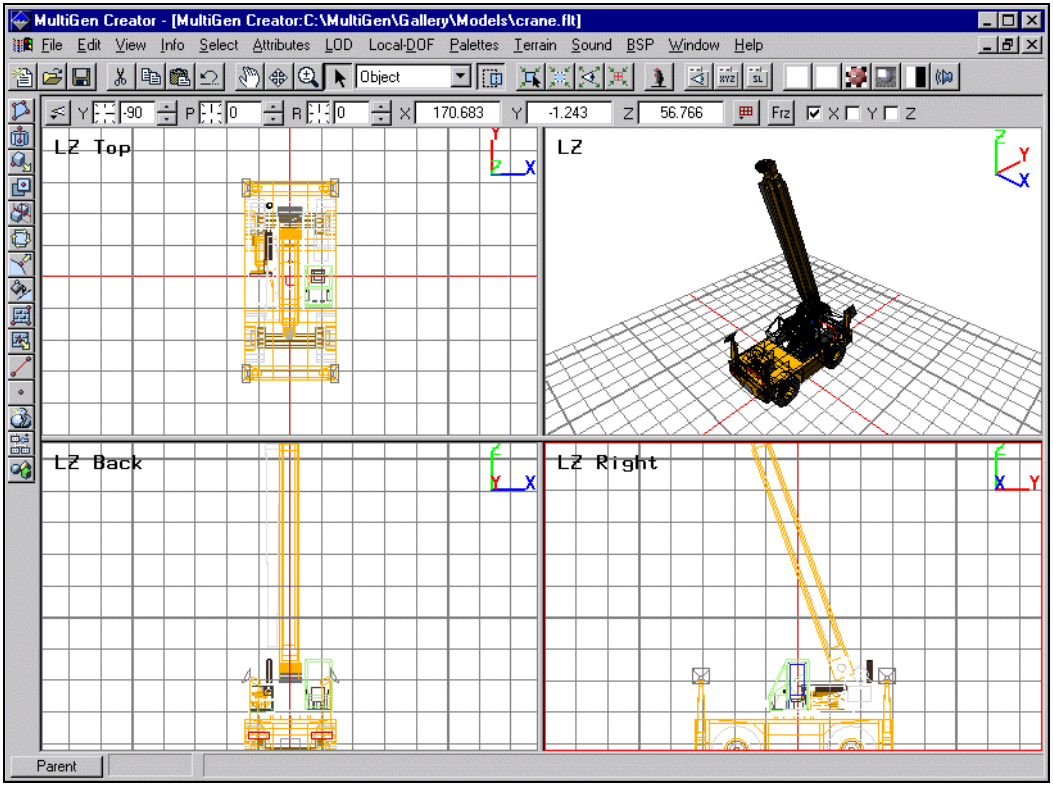
To switch from a multi-pane view to a magnified single-pane view:

Step 1. If the pane you want to magnify is not active, click in it.

Step 2. Choose **Window/Zoom View** (Shift+F6) to magnify the active pane and hide all other panes.

CAD View

To switch to a CAD-style view, choose **Window/CAD View** (Shift+F10). The CAD view consists of three panes with orthographic views and one pane with a default Graphics view.



Reset CAD View

To return to the default CAD view after changing the Graphics view, choose **Window/Reset CAD View**.

Standard View

To switch to a non-orthographic multi-pane view, choose **Windows/Standard View** (Shift+F9).

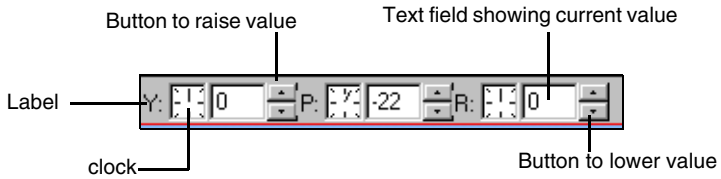
Special Controls

Most of the Creator controls, such as buttons, menus, and text fields, should be familiar to Windows users. This section explains controls that are unique to the Creator application.

Clocks

Certain Creator parameters have a circular range of values. These parameters can cycle through that range like a needle through the 360 degrees of a compass. Generally, these parameters are set using controls called *clocks*. Clock controls can have four components:

- A one-handed clock dial
- An editable text field showing the parameter's current value
- A button that raises the parameter's value
- A button that lowers the parameter's value



Clock Controls for Rotating The Eyepoint.

You can also use the clock controls to "spin" the model in the Graphics view. To do this:

Step 1. Choose **LOD/Auto LOD**.

Step 2. Hold down the Ctrl key and click one of the clocks (Y, P, or R) in the toolbar. The clock you click determines the type of spin.

Step 3. A *Spin Graphics View* dialog box appears when you click the clock. To stop the spinning, click **OK** in this box, or press Esc.

Toolbox Tear-off Windows

Toolboxes, which are opened by clicking buttons on the Toolbox caddy, are *tear-off* windows. When you click a button on the Toolbox caddy, a *docked* Toolbox appears. When it is docked to the Toolbox caddy, a Toolbox closes automatically with the next action you take. However, if you move (or *tear off*) the open Toolbox, it floats on top of the Database window and stays open until you close it. Tear-off—or toolbox—windows can be docked to other open Toolboxes by dragging them close enough to snap together.

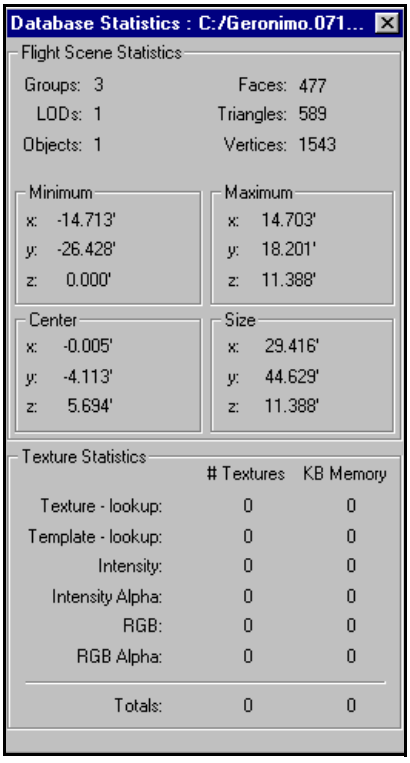
To hide all open Toolboxes, press F5. Press F5 again to show them.

Current Buttons

Each database window contains buttons on the control bar that display a thumbnail of the current color, alternate color, IR color, light source, material, texture, texture map, and sound for its database. Clicking once on a button opens its associated palette. Double-clicking a button lets you to edit the current selection (except current sound).

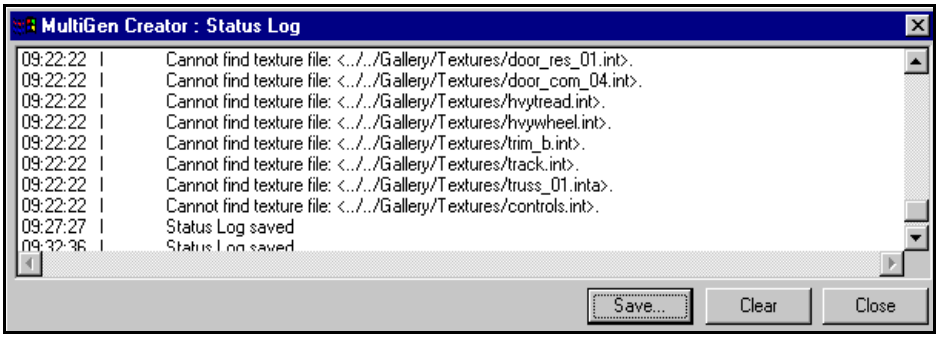
Statistics

Info/Statistics (Ctrl+?) displays summary information about the selected portion of the database. If nothing is selected, **Info/Statistics** displays summary information about the entire database. The information displayed can be useful for checking overhead in the database.



Error Messages

Creator beeps each time it issues a message. The message is displayed in the Status bar. To view a complete list of error messages, choose **Info/Status Log**.



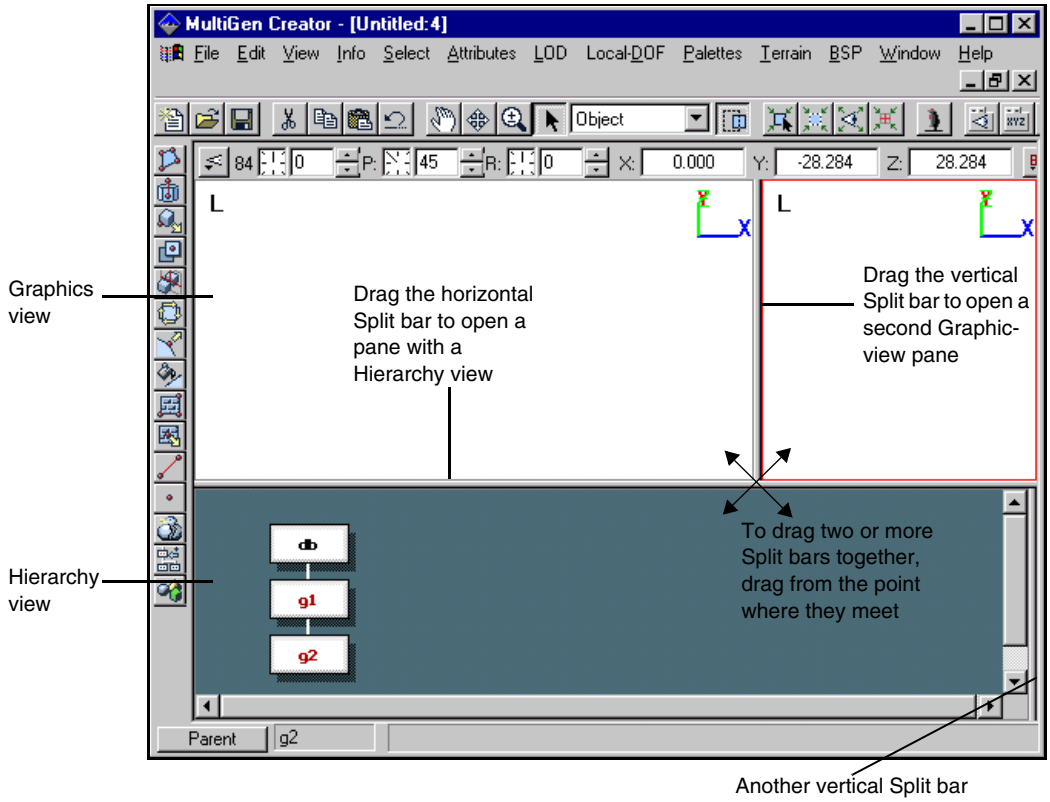
You can leave the *Status Log* window open at all times to see the error messages as they occur. You can also save the contents of the Status Log window to a text (*.txt) file. To save Status Log messages:

Step 1. Click **Save**. The *Save Status Log* dialog box opens.

Step 2. Choose a directory path, enter a file name in the **File name** field, and then click **OK**. The messages will be save to a file with the specified name and a `.txt` extension.

Window Panes

Using the Split bars, you can split each Database window into two, three, or four panes. A pane can display the geometrical data of a database file, which means that it shows the graphical representation of your models. This is known as the *Graphics view*. A pane can also display the structural data of a database file. This is known as the *Hierarchy view*.



You can work on the same model in different panes, but only one pane can be active at a time. The active pane is outlined in red. Actions that affect the geometry or hierarchy of models will appear in both the active and inactive panes. Other actions appear only in the active pane.

4 Toolboxes

The tools that create and manipulate items in the database are grouped into Toolboxes according to their functions. Toolboxes are located in the Toolbox Caddy on the left side of the Creator application window.

You open a toolbox by clicking its button in the Toolbox Caddy. When the Toolbox is docked, it closes automatically with the next action you take. However, if you move (or *tear off*) the open Toolbox, it floats on top of the Database window and stays open until you close it. Tear-off—or Toolbox—windows can be docked to other open Toolboxes by dragging them close enough to snap together.

You can create a custom Toolbox by selecting and dragging tools from their Toolboxes onto the Custom Toolbox palette. In this way, you can keep one Toolbox open with only the tools that you will use.

To hide all open Toolboxes, press F5. Press F5 again to show them.

Face Tools

Face tools create basic shapes that are the building blocks of a database.



Polygon Tool



The **Polygon** tool (Shift+I), located in the **Face** toolbox, builds arbitrarily-shaped polygons. The *Polygon* dialog box prompts you for each vertex of the polygon in succession.

To Create a Polygon

Step 1. Set a group or object node as the parent.

Step 2. Select **Face** modeling mode from the **Mode** list in the Toolbar.



Step 3. Open the **Face Tools** toolbox.

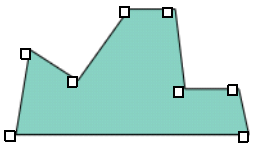


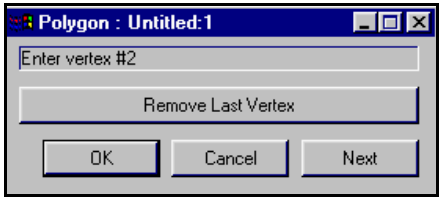
Step 4. Click **Polygon**.

The *Polygon* dialog box opens and prompts you to pick a location for the polygon's first vertex.

Step 5. In the Graphics view, click a location to place the first vertex. You can remove this vertex by clicking **Remove Last Vertex**.

Step 6. Repeat the previous step until you have placed all of the vertices for your polygon, and then click **OK** to create the polygon, **Next** to create another polygon, or **Cancel** to cancel the procedure.



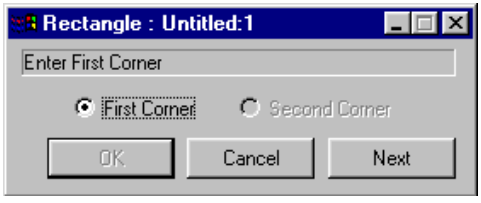


Rectangle Tool



The **Rectangle** tool, located in the **Face** toolbox, creates rectangular faces.





To Create a Rectangle

Step 1. Set a group or object node as the parent.



Step 2. Open the **Face Tools** toolbox.

Step 3. Click the **Rectangle** tool. The *Rectangle* dialog box opens.

Step 4. Click a position for the first corner. A point appears at the designated position and **Second Corner** becomes enabled.

You can change the position of the point by selecting **First Corner** and then selecting a new position in the Graphics view.

Step 5. Click a position for the second corner (diagonally opposite corner). To change the position, drag the point or select a different position.

Step 6. Click **OK** to create the rectangle and close the *Rectangle* dialog box, click **Next** to create the current rectangle and begin another rectangle, or click **Cancel** to cancel the operation.

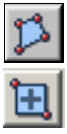
Rectangle Plus Tool



The **Rectangle Plus** tool, located in the **Face** toolbox, includes additional functionality for creating rectangular faces. You can create rectangular faces from two or three points that you specify on the tracking plane. You can dynamically change the dimensions by moving the points. The **Rectangle Plus** tool also allows you to create rectangular faces that are perpendicular to the tracking plane, and apply texture using different types of mappings.

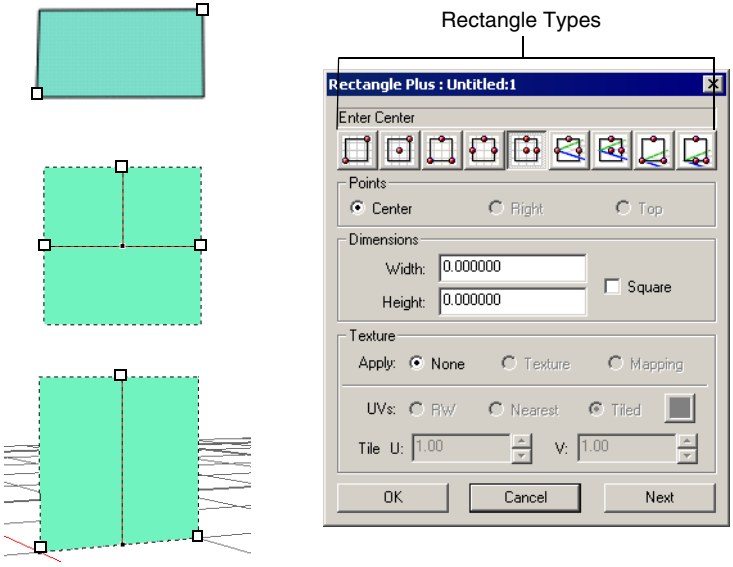
To Create a Rectangle

Step 1. Set a group or object node as the parent.



Step 2. Open the **Face Tools** toolbox.

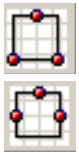
Step 3. Click the **Rectangle Plus** tool. The *Rectangle Plus* dialog box opens.



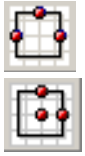
Step 4. Select one of the following rectangle types located at the top of the *Rectangle Plus* dialog box:



- **Rectangle from Two Corners** - Creates a rectangle from a lower left vertex and an upper right vertex.
- **Rectangle from Center and Corner** - Creates a rectangle from a center vertex and a corner vertex.



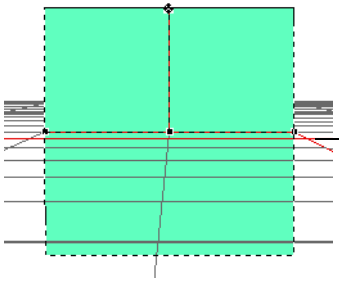
- **Rectangle from Bottom Left, Bottom Right, and Top** - Creates a rectangle from a bottom left vertex, a bottom right vertex, and a top vertex.



- **Rectangle from Center Line and Top** - Creates a rectangle from left and right vertices, and a top vertex.
- **Rectangle from Center, Right, and Top** - Creates a rectangle from a center vertex, a right vertex, and a top vertex.



- **Rectangle from Center Line and Top (Perpendicular to Trackplane)** - Creates a rectangle from left and right vertices, and a top vertex that is perpendicular to the tracking plane.



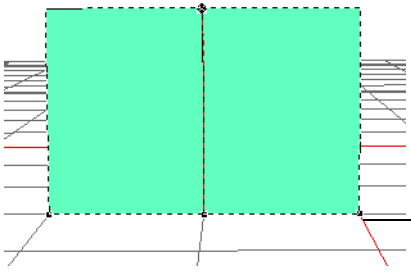
Center line (from left and right vertices) is on the tracking plane. Half of the rectangle is above the tracking plane, and the other half is below the tracking plane.



- **Rectangle from Center, Right, and Top (Perpendicular to Trackplane)** - Creates a rectangle from a center vertex, a right vertex, and a top vertex that is perpendicular to the tracking plane.



- **Rectangle from Bottom Left, Bottom Right, and Top (Perpendicular to Trackplane)** - Creates a rectangle from a bottom left vertex, a bottom right vertex, and a top vertex that is perpendicular to the tracking plane.



Bottom line (from bottom left and right vertices) is on the tracking plane. The entire rectangle is above the tracking plane.



- **Rectangle from Bottom Center, Right, and Top (Perpendicular to Trackplane)** - Creates a rectangle from a bottom center vertex, a bottom right vertex, and a top vertex that is perpendicular to the tracking plane.

You are prompted to pick the location of the rectangle's first vertex.

Step 5. In the Graphics view, click a position for the first vertex. A point appears at the designated position.

Step 6. Repeat the previous step until you have placed all of the vertices for your rectangle.

Note: You can change the position of any of the points of the rectangle by selecting the associated vertex option in the *Rectangle Plus* dialog box, and then selecting a new position in the Graphics view.

Step 7. You can change the **Width**, **Height**, or both dimensions of the rectangle. If you set **Square**, the rectangle that you define will have equal width and height dimensions. If you enter a value for either the width or height, the other dimension will be automatically updated to match the value you enter. Also, when you enter points of the rectangle using the mouse, the rectangle you define will remain square.

Step 8. Select the **Texture** or **Mapping** option to apply the current texture or texture mapping. If you select the **Texture** option, the texture coordinate u and v values that are assigned to the vertices of the new polygon will be set according to the orientation of the rectangle you define. If you select the **Mapping** option, the current texture mapping will determine the texture coordinate u and v values.

Note: If you specify the texture settings before you create the rectangle, the texture or texture mapping will automatically be applied to the completed rectangle.

Step 9. If you selected the **Texture** option, select one of the following options for applying the texture:

- **RW** - The real world values in the **Texture Attributes** are used to calculate the texture coordinate u and v values that are assigned to the vertices of the new polygon.
- **Nearest** - The real world values in the **Texture Attributes** are used to calculate the texture coordinate u and v values. The calculated u and v values are then rounded to the nearest whole number. This ensures that the texture repeats evenly across the polygon while still retaining the real world dimensions.
- **Tiled** - The texture is repeated according to the values that you specify for the **Tile U** and **V** dimensions.



Step 10. Click **Rotate UVs** to rotate the texture on the rectangle in 90-degree increments.

Step 11. Click **OK** to create the rectangle and close the *Rectangle Plus* dialog box, **Next** to create the current rectangle and begin another rectangle, or **Cancel** to cancel the operation.

Circle Tool



The **Circle** tool, located in the **Face** toolbox, constructs a polygon representing a circle, given the center of the circle and one point on its side.

To Create a Circle

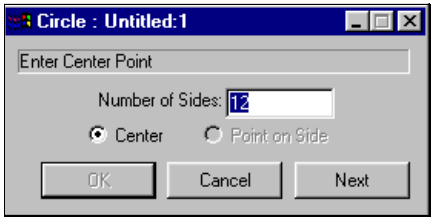
Step 1. Set a group or object node as Parent.



Step 2. Open the **Face Tools** toolbox.



Step 3. Click the **Circle** tool. The *Circle* dialog box opens.

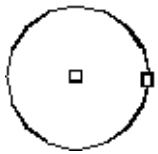


Step 4. Set the number of equal sides that the circular polygon will have. The greater the number of sides, the smoother the circle will appear.

Note: If you have hardware OpenGL support (such as a supported 3D graphics card), you can have up to 255 sides in a circle. If Creator detects only software OpenGL, then you will be limited to 127 sides in a circle.

Step 5. In the Graphics view, click a location for the circle's **Center**. The point appears in the Graphics view and the dialog box prompts you to select a **Point on Side**.

Step 6. In the Graphics view, click a location for the point on the side (perimeter) of the circle, and then click **OK** to create the circle, **Next** to create the current circle and begin another circle, or **Cancel** to cancel the procedure.



Circle Plus Tool



The **Circle Plus** tool, located in the **Face** toolbox, includes additional functionality for creating polygons that represent ellipses. You can create circular faces from up to three points that you specify on a circle, or you can create elliptical shapes from points that you specify on a rectangle. You can dynamically change the dimensions of the ellipse by moving the points. The **Circle Plus** tool also allows you to create ellipses that are perpendicular to the tracking plane, and apply texture using different types of mappings.

Note: Because circles are specialized forms of ellipses, the term *ellipse* is used to describe both a standard circle and an ellipse in this section.

To Create an Ellipse

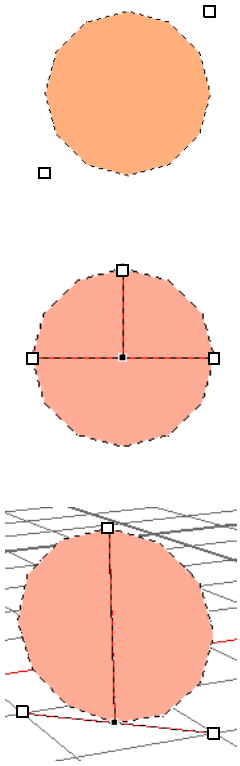
Step 1. Set a group or object node as the parent.

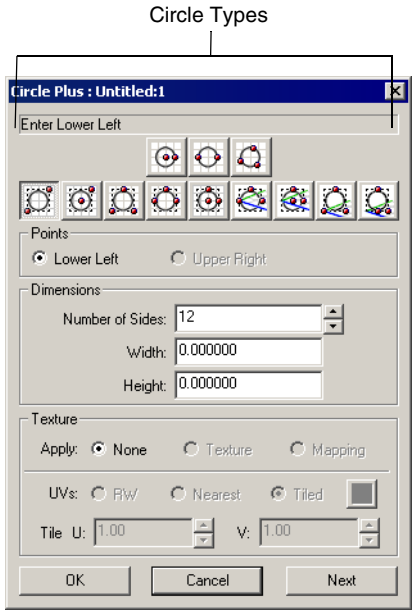


Step 2. Open the **Face Tools** toolbox.

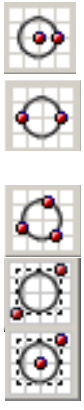


Step 3. Click the **Circle Plus** tool. The *Circle Plus* dialog box opens.





Step 4. Select one of the following circle types located at the top of the *Circle Plus* dialog box:



- **Circle from Center and Point on Edge** - Creates a circle from a center point and a point on the edge.
- **Circle from Point on Edge - Point on Opposite Edge** - Creates a circle from a point on the edge and a point on the opposite edge.
- **Circle from Three Points** - Creates a circle from three points on the edges.
- **Ellipse from Two Corners** - Creates an ellipse from two corner vertices on a rectangle.
- **Ellipse from Center and Corner** - Creates an ellipse from a center point, and a corner vertex on a rectangle.



- **Ellipse from Bottom Left, Bottom Right, and Top** - Creates an ellipse from bottom left, bottom right, and top vertices on a rectangle.



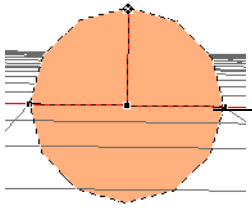
- **Ellipse from Center Line and Top** - Creates an ellipse from left and right vertices, and a top vertex on a rectangle.



- **Ellipse from Center, Right, and Top** - Creates an ellipse from a center point, a right vertex, and a top vertex on a rectangle.



- **Ellipse from Center Line and Top (Perpendicular to Trackplane)** - Creates an ellipse from left and right vertices, and a top vertex on a rectangle that is perpendicular to the tracking plane.



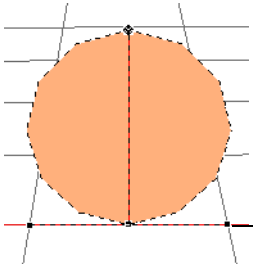
Center line (from left and right vertices) is on the tracking plane. Half of the ellipse is above the tracking plane, and the other half is below the tracking plane.



- **Ellipse from Center, Right, and Top (Perpendicular to Trackplane)** - Creates an ellipse from a center point, a right vertex, and a top vertex on a rectangle that is perpendicular to the tracking plane.



- **Ellipse from Bottom Left, Bottom Right, and Top (Perpendicular to Trackplane)** - Creates an ellipse from bottom left, bottom right, and top vertices on a rectangle that is perpendicular to the tracking plane.



Bottom line (from bottom left and right vertices) is on the tracking plane. The entire ellipse is above the tracking plane.



- **Ellipse from Bottom Center, Right, and Top (Perpendicular to Trackplane)** - Creates an ellipse from bottom center, right, and top vertices on a rectangle that is perpendicular to the tracking plane.

You are prompted to pick the location of the circle's first point.

Step 5. In the Graphics view, click a position for the first point. A point appears at the designated position.

Step 6. Repeat the previous step until you have placed all of the points for your ellipse.

Note: You can change the position of any of the points of the ellipse by selecting the associated vertex option in the *Circle Plus* dialog box, and then selecting a new position in the Graphics view.

Step 7. You can change the **Number of Sides** of the ellipse. You can also change the **Width**, **Height**, or both dimensions of the ellipse.

Step 8. Select the **Texture** or **Mapping** option to apply the current texture or texture mapping. If you select the **Texture** option, the texture coordinate *u* and *v* values that are assigned to the vertices of the new polygon will be set according to the orientation of the ellipse you define. If you select the **Mapping** option, the current texture mapping will determine the texture coordinate *u* and *v* values.

Note: If you specify the texture settings before you create the ellipse, the texture or texture mapping will automatically be applied to the completed ellipse.

Step 9. If you selected the **Texture** option, select one of the following options for applying the texture:

- **RW** - The real world values in the **Texture Attributes** are used to calculate the texture coordinate *u* and *v* values that are assigned to the vertices of the new polygon.
- **Nearest** - The real world values in the **Texture Attributes** are used to calculate the texture coordinate *u* and *v* values. The calculated *u* and *v* values are then rounded to the nearest whole number. This ensures that the texture repeats evenly across the polygon while still retaining the real world dimensions.
- **Tiled** - The texture is repeated according to the values that you specify for the **Tile U** and **V** dimensions.

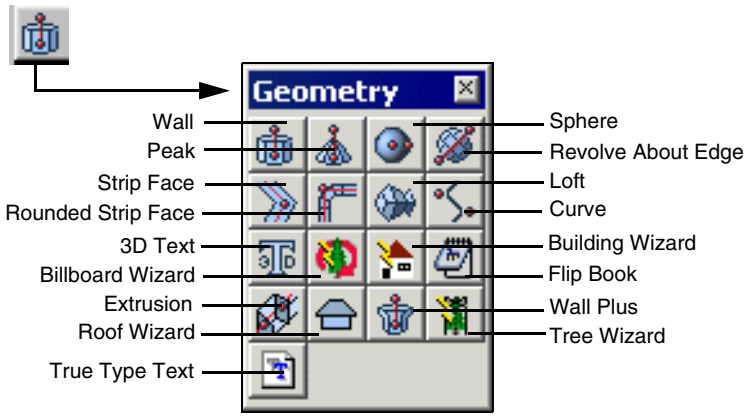
Step 10. Click **Rotate UVs** to rotate the texture on the ellipse in 90-degree increments.



Step 11. Click **OK** to create the ellipse and close the *Circle Plus* dialog box, **Next** to create the current ellipse and begin another ellipse, or **Cancel** to cancel the operation.

Geometry Tools

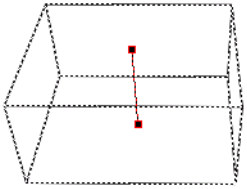
Geometry tools create complex polygons and 3D shapes from polygons.

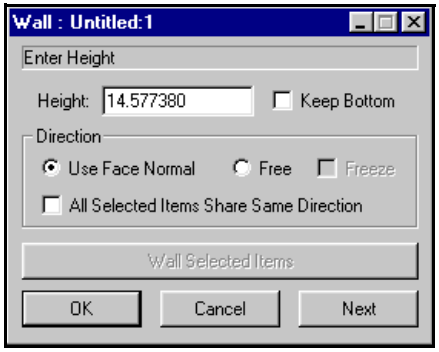


Wall Tool

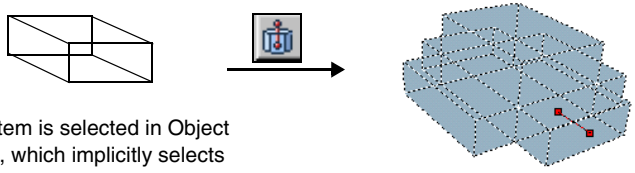


The **Wall** tool (Shift+W), located in the **Geometry** toolbox, creates a set of walls from each selected face. The *Wall* dialog box prompts for the **Height** of the new faces.





If noncoplanar faces (faces that do not lie in the same plane) are selected when you choose **Wall**, each face extrudes in the direction of its own normal (straight out), as shown below. In **Face** modeling mode, you can set **All Selected Items Share Same Direction** if you want the faces to extrude in the direction of the last selected face.



This item is selected in Object mode, which implicitly selects all its faces

To Create a Wall

Step 1. In Face modeling mode, select the face to be walled.



Step 2. Open the **Geometry Tools** toolbox.



Step 3. Click **Wall** (Shift+W).

Step 4. In the *Wall* dialog box enter a wall height in the **Height** field, or drag the height indicator up or down to achieve a satisfactory height (you do not need to point to the **Wall** construction edge). You can also set **Height** by

referencing an existing edge or vertex in the database with the middle mouse button.

Step 5. Set or clear **Keep Bottom**, as desired. **Keep Bottom** duplicates and reverses the original face to create a bottom for the extruded shape. If this checkbox is cleared, the new shape has a top and sides, but no bottom face.

Step 6. Set **Use Face Normal** if you want the face to extrude in the direction of its normal.

Step 7. Set **Free** if you want to control the direction in which to extrude the face. You can drag the height indicator in any direction. After you determine the direction, you can set the **Freeze** checkbox to lock the height indicator in place and adjust the height.

Step 8. Set **All Selected Items Share Same Direction** if you want all of the selected faces to extrude in the same direction. If you select more than one noncoplanar face and you set **Use Face Normal**, all faces extrude in the direction of the last face that you select.

Step 9. Click **OK** to quit the **Wall** tool, **Next** to create another wall, or **Cancel** to cancel the wall operation.

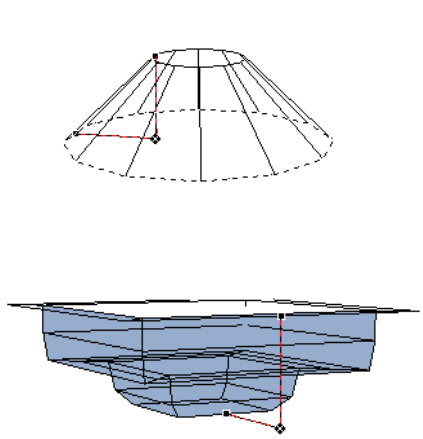
When you click **Next**, **Wall Selected Items** becomes active. Set the appropriate modeling mode, select the faces, and then click **Wall Selected Items** to wall the selected faces.

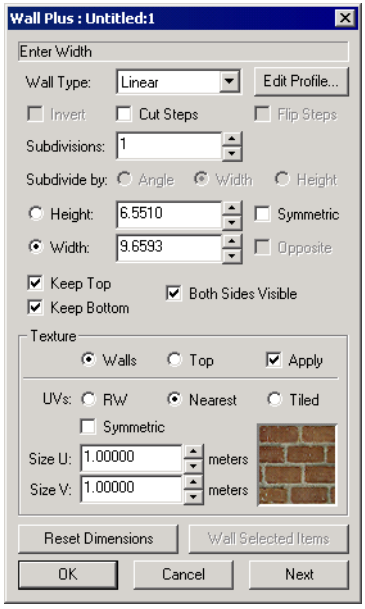
Wall Plus Tool



The **Wall Plus** tool, located in the **Geometry** toolbox, creates a set of walls from each selected face along a shape that you specify. Several different shapes of walls are available, including linear, parabolic, and elliptical. You can also specify a custom wall shape by entering and positioning individual points along the shape.

The wall shape is “swept” around the perimeter of the geometry when you extrude the walls to produce either a smooth or beveled effect.





When you start this tool, each selected polygon extrudes. Two construction edges are displayed in the Graphics view that represent the width and height offsets. You can control the width and height of the extrusion by dragging the end of the corresponding construction edge using the left mouse button, or entering the height and width values.

To Create a Wall

Step 1. In Face modeling mode, select the faces to be extruded.



Step 2. Open the **Geometry Tools** toolbox.



Step 3. Click **Wall Plus**.

Step 4. In the *Wall Plus* dialog box, choose one of the following wall shapes:

- **Linear** - Extrudes polygons linearly along the width and height dimensions.

- **Parabolic** - Extrudes polygons along one of two parabolas defined by the width and height dimensions. Depending on the dimensions you enter, one type of parabola appears curved in and the other type appears curved out.
- **Elliptical** - Extrudes polygons along an ellipse defined by the width and height dimensions. Similar to the parabolic wall type, one type of ellipse appears curved in and the other appears curved out.
- **Custom** - Extrudes polygons along a wall shape that you modify when you click **Edit Profile**. See "To Edit a Wall Profile" on page 102 for the procedure.

Step 5. If you chose the **Parabolic** or **Elliptical** wall type, set **Invert** to switch to the other type of parabola or ellipse.

Two types of either parabolas or ellipses are defined by the width and height settings. One type appears curved in and the other type appears curved out.

Step 6. Set **Cut Steps** to cut a step into each subdivision of the wall.

Step 7. If you set **Cut Steps,** you can set **Flip Steps** to reverse the orientation of the steps that are cut into each subdivision of the wall. In this way, the steps can be cut in or cut out.

Step 8. Enter a value for **Subdivisions** to specify the number of times to subdivide the walls along the given shape.

Step 9. Select one of the following dimensions along which to subdivide the extruded polygons:

- **Angle** - Subdivides the wall polygons evenly along the interior angle of the ellipse. The **Elliptical** wall type is non-linear (linear subdivisions, by definition, are evenly spaced in only width and height dimensions).
- **Width** - Subdivides the wall polygons evenly along the width dimension.
- **Height** - Subdivides the wall polygons evenly along the height dimension.

Step 10. Enter a height for the wall in the **Height** field, or drag the control point up or down to achieve a satisfactory height.

The height of the wall is measured perpendicular from the plane containing the original polygon. A positive height causes the polygons to be extruded above the originals, and a negative height causes the polygons to be extruded below the originals.

Step 11. Enter a width for the wall in the **Width** field, or drag the control point backward or forward to achieve a satisfactory width.

The width of the wall is measured perpendicular from the original edges. A positive width causes the polygons to be extruded inside the originals, and a negative width causes the polygons to be extruded outside.

Step 12. Set **Symmetric** to keep the width and height values equal. When you change one value, the other value also changes.

Step 13. If you set **Symmetric**, you can set **Opposite** to keep the width and height values equal in magnitude but opposite in direction.

Step 14. Set **Keep Top** to create a polygon that closes the walls.

Step 15. Set **Keep Bottom** to keep the original polygon that is being walled.

Step 16. Set **Both Sides Visible** to render both sides of the polygons. This might be useful when **Keep Top** is not set, and you are making an open solid and want to see both sides of your new geometry.

Step 17. Select **Walls** or **Top** to define how you want to texture the polygons that make up the corresponding parts of the new geometry. You can apply texture to the walls, the top, or both.

Step 18. Set **Apply** to automatically apply texture to the currently specified parts (walls or top).

Step 19. Select one of the following options for applying a texture:

- **RW** - The real world values in the **Texture Attributes** are used to calculate the texture coordinate *u* and *v* values that are assigned to the vertices of the new polygons.
- **Nearest** - The real world values in the **Texture Attributes** are used to calculate the texture coordinate *u* and *v* values. The calculated *u* and *v* values are then rounded to the nearest whole number. This ensures

that the texture repeats evenly across the polygon while still retaining the real world dimensions.

- **Tiled** - The texture is repeated according to the values that you specify for the **Size U** and **Size V** dimensions.

Step 20. Set **Symmetric** to keep the **Size U** and **Size V** values equal. When you change one value, the other value also changes.

Step 21. Enter the texture's real world *u*, *v* dimensions in **Size U** and **Size V**. (These values are always measured in meters, regardless of the database units employed for your database.)

Step 22. Click **Reset Dimensions** to reset both the width and height dimensions to zero.

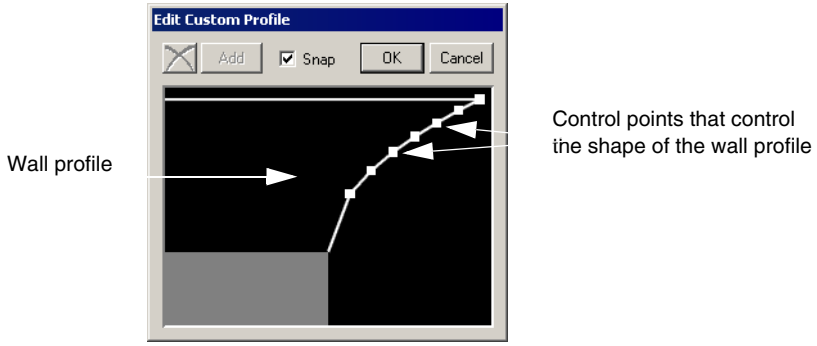
Step 23. Click **OK** to quit the **Wall Plus** tool, **Next** to create another wall, or **Cancel** to cancel the wall operation.

When you click **Next**, **Wall Selected Items** becomes active. Set the appropriate modeling mode, select the faces, and then click **Wall Selected Items** to wall the selected faces.

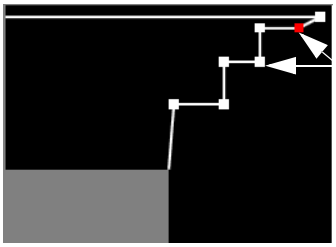
To Edit a Wall Profile

Step 1. In the *Wall Plus* dialog box, click **Edit Profile** to design the profile of your custom wall type.

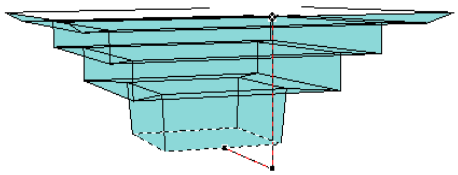
When the *Edit Custom Profile* window displays, a wall shape appears with control points (the wall shape is the last shape that was saved).



Step 2. Select a control point, and drag it with the left mouse to modify the wall shape in the *Edit Custom Profile* window. The currently selected control point appears red. For example, you can drag control points to create steps for a beveled effect in your wall.



Drag the control points to create steps for a beveled effect on the walled geometry.



Step 3. Click **Add** to add a new control point after the selected control point.

Note: You can also add a point at any position on an existing edge by clicking the left mouse button on the edge.



Step 4. Click **Delete** to delete the selected control point.

Step 5. Set **Snap** to align control points with existing points when they are dragged. Horizontal lines, vertical lines, or both appear in the display to show when a point is being aligned with another point.

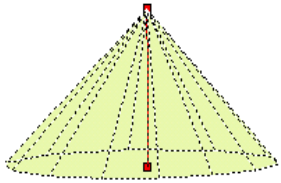
Note: You can also align a point with other points if you hold down the Ctrl key and drag it.

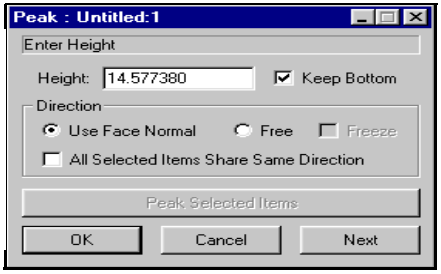
Step 6. Click **OK** to save the profile, or **Cancel** to cancel the operation and close the window.

Peak Tool



The **Peak** tool (Shift+P), located in the **Geometry** toolbox, creates a set of peaked faces from each selected face. The *Peak* dialog box prompts for the **Height** of the new faces. As with **Wall**, **Peak** applies independently to each selected face. In **Face** modeling mode, you can set **All Selected Items Share Same Direction** if you want the faces to extrude in the direction of the last selected face.





To Create a Peak



Step 1. In Face mode select the face to be peaked.

Step 2. Open the **Geometry Tools** toolbox.



Step 3. Choose **Peak** (Shift+P).

Step 4. In the *Peak* dialog box enter a peak height in the **Height** field, or drag the height indicator up or down to achieve a satisfactory height. You do not need to point to the **Peak** construction edge.

You can also set the height by referencing an existing edge or vertex in the database with the middle mouse button.

Step 5. Set or clear **Keep Bottom**, as desired. **Keep Bottom** duplicates and reverses the original face to create a bottom for the extruded shape. If this checkbox is cleared, the new shape has a top and sides, but no bottom face.

Step 6. Set **Use Face Normal** if you want the face to peak in the direction of its normal.

Step 7. Set **Free** if you want to control the direction in which to peak the face. You can drag the height indicator in any direction. After you determine the direction, you can set the **Freeze** checkbox to lock the height indicator in place and adjust the height.

Step 8. Set **All Selected Items Share Same Direction** if you want all of the selected faces to peak in the same direction. If you select more than one noncoplanar

face and you set **Use Face Normal**, all faces peak in the direction of the last face that you select.

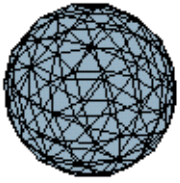
Step 9. Click **OK** to create the peak and close the dialog box, **Next** to create the current peak and begin another peak, or **Cancel** to cancel the Peak operation.

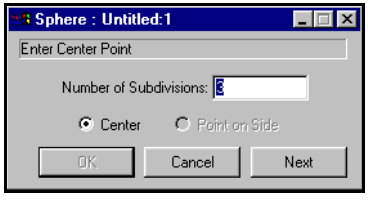
When you click **Next**, **Peak Selected Items** becomes active. Set the appropriate modeling mode, select the faces, and then click **Peak Selected Items** to peak the selected faces.

Sphere Tool



The **Sphere** tool, located in the **Geometry** toolbox, constructs a geodesic sphere based on an icosahedron (a 20-sided polygon).





To Create a Sphere

Step 1. Select **Object** from the **Mode** list in the Toolbar.

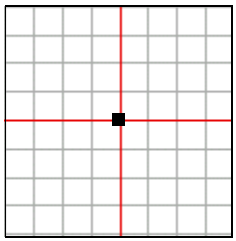


Step 2. Open the **Geometry Tools** toolbox.

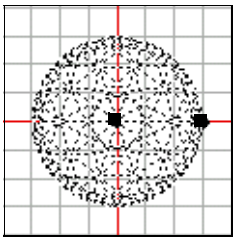


Step 3. Click the **Sphere** tool. The *Sphere* dialog box opens.

Step 4. In the Graphics view, select a location for the sphere's **Center** point.



Step 5. Click a location for **Point on Side**; a point on the outer edge of the sphere.



Step 6. Set the **Number of Subdivisions** the sphere will have. This step is optional. You can accept the default setting of three subdivisions and skip to the next step.

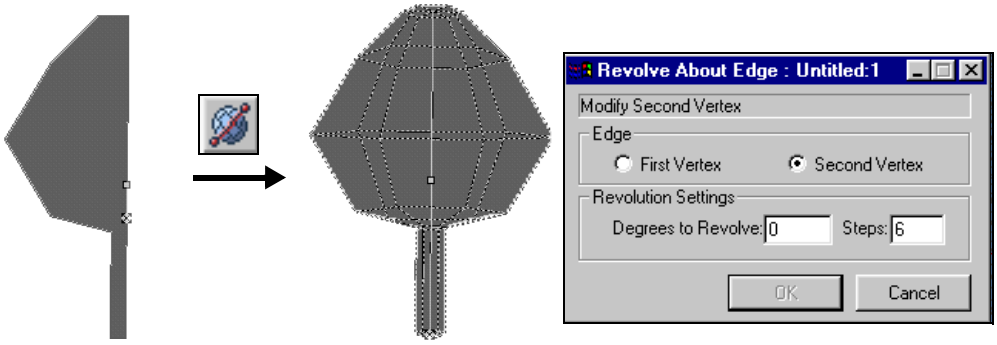
Number of Subdivisions controls the number of faces in the sphere. The final number of faces is n-squared * 20, where n = **Number of Subdivisions**. Thus, if n = 1, the sphere has 20 sides, if n = 2 the sphere has 80 sides, and so on.

Step 7. Click **OK** to create the sphere and close the *Sphere* dialog box, **Next** to create another sphere, or **Cancel** to quit the procedure.

Revolve About Edge Tool



The **Revolve About Edge** tool, located in the **Geometry** toolbox, creates one or more convex objects by revolving the selected face about a defined edge. The edge serves as the axis of rotation.



To Revolve a Face About an Edge



Step 1. Open the **Face Tools** toolbox.

Step 2. Create a face using the **Polygon** tool.

Step 3. Select the face.



Step 4. Open the **Geometry Tools** toolbox.

Step 5. Click the **Revolve About Edge** tool.

Note: Make sure **Degrees to Revolve** is set to 0 (zero). This makes it easier to set the first and second degree of rotation.

Step 6. The first vertex is automatically selected. If you want to choose a different vertex, click **First Vertex** in the *Revolve About Edge* dialog box and then, in the Graphics view, click to set the first vertex.

Step 7. In the Graphics view, click to set the second vertex.

Step 8. Set the **Degrees to Revolve** to 360.

Step 9. Set **Steps** to 6.

Step 10. Click **OK**.

Revolve About Edge Parameters

These controls let you set the parameters for the **Revolve About Edge** tool.

Degrees to Revolve

Controls the number of degrees to use. The default is 0. If you select 360 degrees, it wraps the face into a solid geometry, which looks similar from all angles. For example, a circular face becomes a sphere. If you select 180 degrees, the geometry has one flat side. For example, a circular face becomes a hemisphere.

Steps

Controls the number of sides for the geometry. More steps makes the complex geometry faces appear smoother and circular faces more circular.

Strip Face Tool



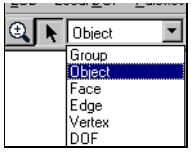
The **Strip Face** tool, located in the **Geometry** toolbox, builds a series of fixed-width polygons along a path you lay out with the mouse.

To Build a Strip Face

Step 1. Set a parent for the new strip face.



Step 2. Select either **Object** or **Face** modeling mode. In **Object** modeling mode, **Strip Face** inserts an object node above the new faces.





Step 3. Open the **Geometry** toolbox.



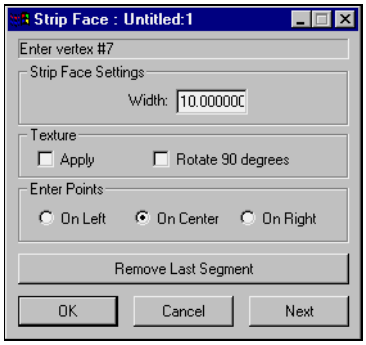
Step 4. Click **Strip Face**.

Step 5. In the *Strip Face* dialog box:

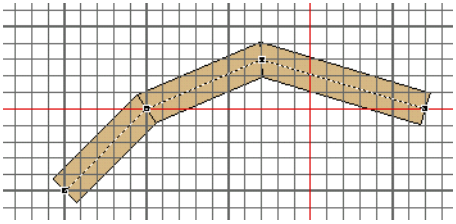
Set **Width** to specify the width of the next path segment you create.

To automatically apply the Current Texture to all path segments, set **Apply**. (You can open the Texture palette and change the Current Texture at any time - all the faces created together get the same texture pattern.)

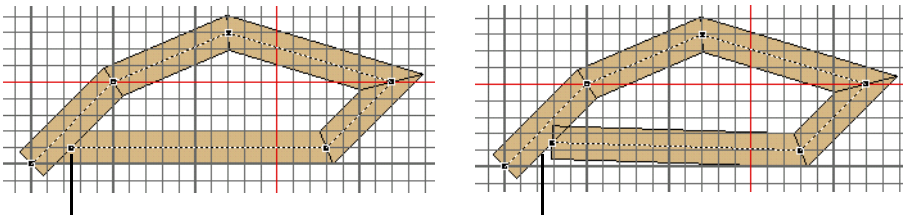
Click **Enter Points: On Left**, **On Center**, or **On Right** to specify whether the path you enter determines the left edge, center, or right edge of the next segment.



Step 6. In the Graphics view, enter points to specify each segment of the path.



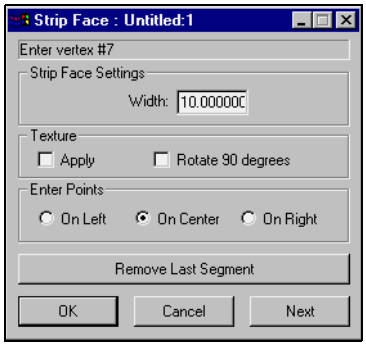
If you enter a point with an edge reference (by middle-clicking an existing edge), the new segment aligns to the referenced edge. To prevent automatic alignment, press Alt while middle-clicking the edge.



Step 7. Click **OK** to accept the new faces, **Next** to accept the faces and build another strip face, or **Cancel** to discard the change.

Strip Face Parameters

Set the Strip Face parameters to create path segments. The *Strip Face* dialog box contains the following parameters:



Width

Defines the width of the next path segment. Changing this value does not affect previous segments.

Apply

Applies the Current Texture to all segments in the strip face. You can change the Current Texture at any time while using this tool.

Rotate 90 Degree

Rotates the Current Texture 90 degrees before applying it to the strip face.

Enter Points

Determines whether the next point you enter defines the left edge, center line, or right edge of the next path segment.

Remove Last Segment

Removes the last segment of the strip face.

Rounded Strip Face Tool



Rounded Strip Face, in the **Geometry** toolbox, builds a connected series of fixed-width faces that are joined together with arcs. This tool is suitable for creating items like roads and railroad tracks.

Each segment of a rounded strip face is projected down towards the *x-y* plane onto all *visible* underlying polygons, without using the tracking plane. This has the following implications:

- You do not have to set the database parent to use **Rounded Strip Face** unless you set **Create as Object**.
- You cannot project a rounded strip face onto vertical (or nearly vertical) faces.
- If you do not want to project the strip face onto certain polygons, turn off their display before you use **Rounded Strip Face**.

To Build a Rounded Strip Face

Step 1. Examine the area of the database that you will be working in, and turn off the display of any polygons onto which you do not want to project the rounded strip face.



Step 2. Open the **Geometry** toolbox.



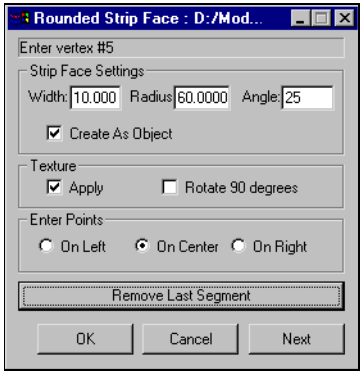
Step 3. Click **Rounded Strip Face**.

Step 4. In the *Rounded Strip Face* dialog box:

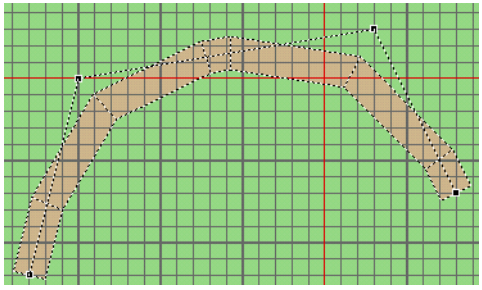
Set **Width** to specify the width of the next path segment you create.

To automatically apply the Current Texture to all path segments, set **Apply**.

Click **Enter Points: On Left**, **On Center**, or **On Right** to specify whether the path you enter determines the left edge, center, or right edge of the next segment.



Step 5. In the Graphics view, enter points to specify each segment of the path.



Step 6. Click **OK** to accept the rounded strip faces, **Next** to accept the faces and build another rounded strip face, or **Cancel** to discard the change.

Rounded Strip Face Parameters

Set the **Rounded Strip Face** parameters to create rounded strip faces. The *Rounded Strip Face* dialog box contains the following parameters:



Width

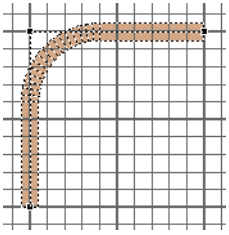
Controls the width of each segment of the rounded strip face.

Radius

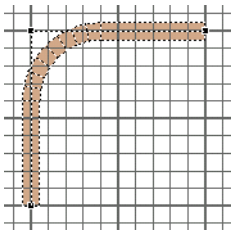
Determines the maximum amount of bend between each segment in the path. By decreasing this value, you can bend the path more sharply. **Radius** must be at least twice the value of **Width**.

Angle

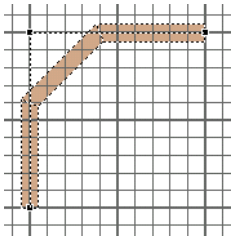
Controls the number of faces in the arc between segments of the strip face. Decreasing **Angle** increases the number of faces in the arc.



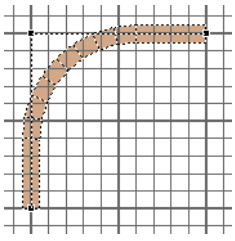
Width = 5
Radius = 20
Angle = 5



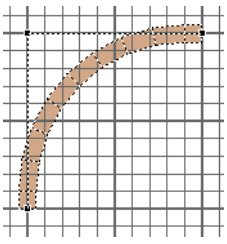
Width = 5
Radius = 20
Angle = 10



Width = 5
Radius = 20
Angle = 45



Width = 5
Radius = 30
Angle = 10



Width = 5
Radius = 45
Angle = 10

Create as Object

Collects the new strip faces into an object node and attaches the object to the current parent. Make sure to set the parent before setting this checkbox. If **Create as Object** is not set, rounded strip faces become subfaces of the polygons they are projected onto.

Apply

Applies the Current Texture to the entire path. You can change the Current Texture at any time while using this tool.

Rotate 90 Degrees

Rotates the Current Texture 90 degrees before applying it to the rounded strip face.

Enter Points: On Center, On Left, On Right

Controls whether the next point you enter defines the centerline, left edge, or right edge of the next path segment.

Remove Last Segment

Removes the last segment of the rounded strip face.

Loft Tool



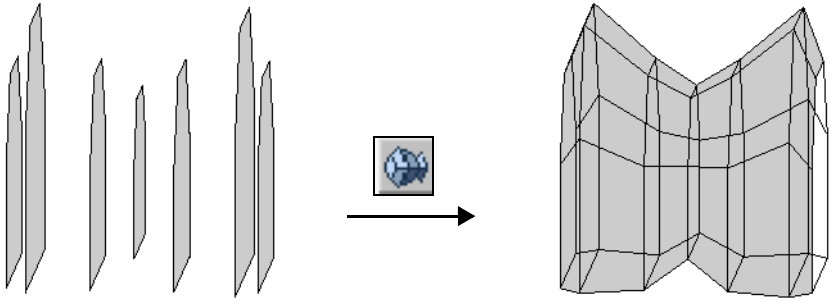
Loft creates a three-dimensional object from a set of cross-sectional faces. If you want to control the order in which the faces are lofted together, choose **Info/Preferences/ Editing** and select **Loft Faces in Order Selected**. Then select the faces, one at a time, in the order you want to loft them. This preference is usually preferable if the faces are not parallel.

To Loft Faces



Step 1. Open the **Geometry Tools** toolbox.

Step 2. Click the **Loft** tool.



Step 3. Select the faces to loft. The faces created by the loft are placed in the hierarchy as children of the last object node.

Note: After you perform a loft, the original cross sections remain selected. It is usually a good idea to delete these cross sections, since it prevents them from showing through the surface and also reduces the number of polygons in your object. While they are still selected, press the Delete key to remove them.

Curve Tool



Geometry curves are created with the **Curve** tool. These geometries can be copied, pasted and modified using the same tools used to modify other geometries. These curves are stored with the database and can be used in the realtime system as paths for eyepoints or objects. They are colored green in the Graphics view with construction points that you can select and modify.

To Create a Curve

Step 1. Click the node in the Hierarchy view that you want to use as the parent node.

Step 2. Click the **Parent** button.



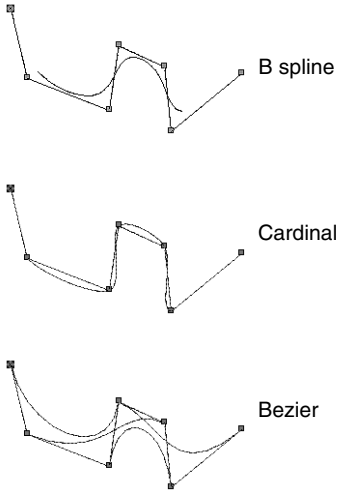
Step 3. Open the **Geometry Tools** toolbox.



Step 4. Click the **Curve** tool to display the *Create Curve* dialog box.



Step 5. Select the type of curve: **B spline**, **Cardinal**, or **Bezier**.



Note: If you want to change the type of curve later (after the curve is done), use the *Curve Attributes* window.

Step 6. In the Graphics view, click at least four points to place the control points for the curve. These control points determine the size and shape of the curve. Use **Remove Last Vertex** to erase the last control point you set.



Note: If the curve is not visible, click **Draw Control Points** in the View panel.

Step 7. To create another curve, click **Next**.

Step 8. Click **OK** to finish creating curves, or click **Cancel** to abort the operation.

Editing a Curve

You can edit an existing curve either by moving its control points, or by changing the curve type in the *Curve Attributes* window.

Control points allow you to determine the shape of the curve. To hide or show the control points for a curve in the Graphics view, click **Draw Control Points** in the View panel.





To select a curve for modifying you must be in **Curve** modeling mode. You can select it from the **Mode Select** menu in the Toolbar, or from the **Mode** toolbox.

To Edit the Curve's Control Points

To change the shape of a curve by changing the control points:



Step 1. Open the **Mode** toolbox.

Step 2. Click the **Vertex Mode** tool to change to Vertex mode (Shift+V).

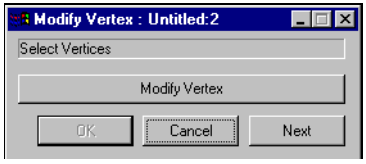
Step 3. Hold down the Shift key and select control points with the left mouse button or drag a selection box around the control points.

Note: If the control points are not visible, click **Draw Control Points** in the View panel.



Step 4. Open the **Modify Vertex** toolbox.

Step 5. Click **Modify Vertex**.



Step 6. Click in the Graphics view to reset the location of the control point.

Step 7. To modify another vertex, click **Next,** then repeat the steps above.

Step 8. Click **OK** to finish modifying the curve, or click **Cancel** to abort the operation.

To Change the Curve Type

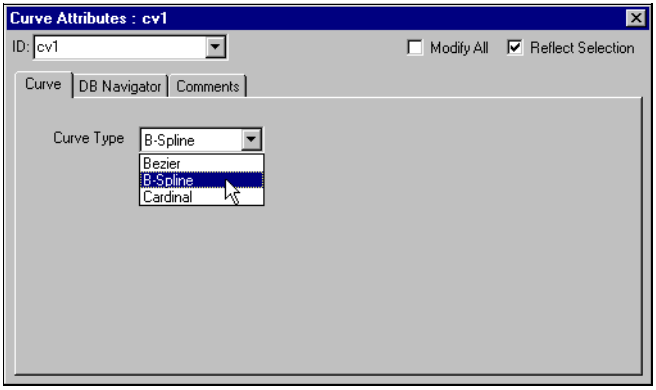
To change the shape of the curve by changing the *type* of curve (for example, from a Bezier to a B-Spline curve type):

Step 1. In the Hierarchy view, select the curve node.

Step 2. Double-click the node to display the *Curve Attributes* window.

Step 3. Click the **Curve** tab.

Step 4. Select a new curve type from the **Curve Type** list.



To Move a Curve

You can move curves using the tools in the **Maneuver** toolbox. For example, to translate a curve, use Shift+T.

To Copy or Paste Curves

Copy and paste an existing curve using **Edit/Copy** (Ctrl+C) and **Edit/Paste** (Ctrl+V).

To Delete a Curve

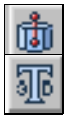
Delete a curve using **Edit/Delete** or use the Delete or Backspace key.

3D Text Tool



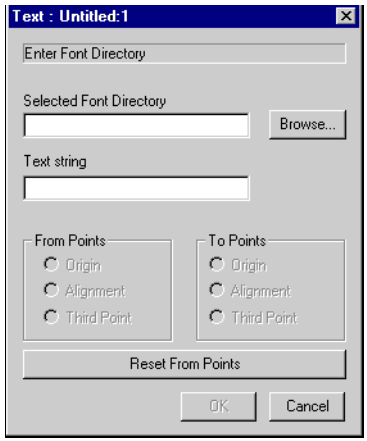
The **3D Text** tool creates 3D text in your database. The 3D text is similar to other geometries, so you can modify it with any geometry tools. To create 3D text geometry, you must select a font to use, enter the text, then choose its location in the database.

To Create a 3D Text Object



Step 1. Open the **Geometry Tools** toolbox.

Step 2. Click the **3D Text** tool. The *Text* dialog box opens.



Step 3. Type the path to a 3D font directory in the **Selected Font Directory** field, or use the **Browse** button to select a directory.

Step 4. Select the control file in the directory. The control file maps each keyboard character to its corresponding graphic character file (there is one control file per font).

Step 5. Type the text into the **Text String** field. These must be printable ASCII characters; spaces are allowed.

Step 6. In the Graphics view, click the location to place the text. To modify the position of the text, use the **From Points** and **To Points** in the *Text* dialog box.

Text Tool Parameters

Text tool parameters define and change the 3D text in the database.

Selected Font Directory

Type in the font directory path or select the **Browse** button to open a file selection box and select a directory.

Text String

Enter the text string to place in the database. These can only be printable ASCII characters and spaces.

From Points

Specify three points on the text string for the **From Points**, then set the **To Points**. The **From Points** are used to position and align the 3D text in the database.

To Points

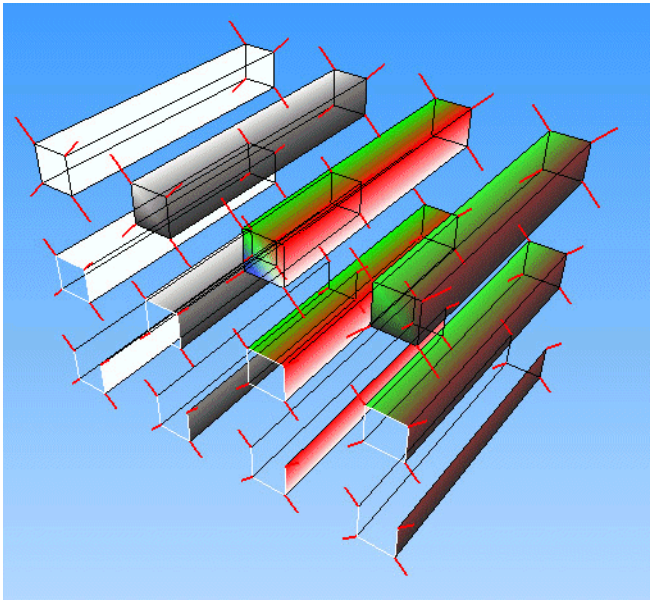
To modify the position of the text in the database, select an **Origin**, **Alignment**, and **Third Point** for the text string in the Graphics view. Each **From Point** maps to the corresponding **To Point**.

Extrusion Tool



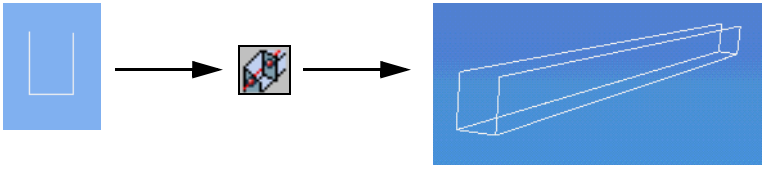
The **Extrusion** tool allows you to extrude a face or faces in the Graphics view. You can control the shape of the extruded face by either using your mouse to enter control points or by extruding along a predefined curve. You can also save the extrusion path for use in the realtime system or drive the path in the Graphics view (if you have the Road Tools option).

The **Extrusion** tool also allows you to control how textures are applied to each face. Texture mapping is fully controllable, including the way the texture is applied to each face and the orientation of the texture. You can map different textures to each face of an object and each face can have a different texture mapping method. For example, a cube can have different textures on each face (but only one texture per face).



The attributes of a face affect its extrusion. For example, if you set the face attribute to **Wireframe** in the *Face Attributes* window, you can create an "open extrusion." That is, one side of the resulting polygon will be open.

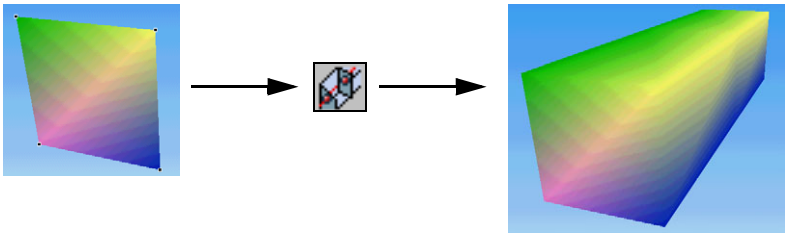
Open Extrusion



The square polygon has its Face attribute set to wireframe. When extruded, it results in an open topped polygon similar to a rain gutter.

You can also set the vertex attributes of a face before extruding it. In the figure below, each vertex is set to a different color and its drawing attribute is set to **Gouraud** shading (in the *Face Attributes* window). When extruded, the colors are blended across the faces.

Colored Vertex Extrusion



The square polygon has colored vertices. When extruded, it creates a polygon with colors blended across its faces.

Two extrusion methods are shown here: one using control points to define the extrusion, the other one using a curve to define the extrusion. Note that you can combine these two techniques by defining a few control points, and then selecting the curve and extruding.

To Extrude a Face

The Extrusion tool works on faces only. You must be in Face mode and have the faces selected before starting the tool. If you are applying textures to the new polygons, open the Texture palette first and select the textures.

Step 1. Select the face or geometry you want to extrude with the **Extrusion** tool.

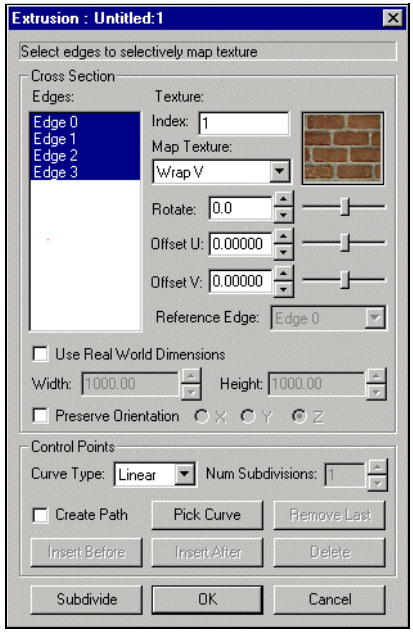


Step 2. Open the **Geometry Tools** toolbox.



Step 3. Click the **Extrusion** tool.

The selected edges are shown in the **Edges** section of the *Extrusion* dialog box.



Step 4. Set the tracking plane to the plane along which you want to extrude the face. For example, if you created the face in the YZ plane, set it to XY or XZ plane.

Step 5. In the Graphics view, click a set of points to define the extrusion. If you want to delete points, use **Remove Last**.

Note: You may want to turn off **Grid Snap** in the View panel so it is easier to precisely set the control points.

Step 6. If you are applying textures to the new polygons, type the texture index number in the **Index** field (-1 in this field indicates that no texture will be applied).

Step 7. Select the texture preferences. The textures are displayed on the extruded polygons and all changes to the texture preferences are shown immediately in the Graphics view.

Note: If the textures are not visible in the Graphics view, press T to turn on the texture display.

Step 8. Click **OK** to finish the Extrusion.

To Extrude a Face Using a Curve

The Extrusion tool can use an existing curve as a shape along which to extrude the faces. Note that the curve is made with the **Curve** tool, not the **Construction Curve**.



Step 1. Open the **Geometry Tools** toolbox.

Step 2. Click the **Curve** tool and create a curve (see "To Create a Curve" on page 116).



Step 3. Select the face or geometry you want to extrude.



Step 4. Click the **Extrusion** tool.

Step 5. Click **Pick Curve**, then click the curve in the Graphics view.

The face(s) will extrude in the same shape as the curve.

Note: You can save this curve as a path by setting **Create Path**.

Step 6. Set texture preferences, if desired.

Step 7. Click **OK** to finish the Extrusion.

To Save a Path With the Extrusion Tool

Paths can be saved for use in the realtime system or driven in the Graphics view (if you have the Road Tools option). Paths are based on the extrusion method. If you create an extrusion using your mouse, the path is similar in shape to the extruded face. If you create an extrusion using a curve, the path is similar in shape to the curve.

Step 1. Start the extrusion using either method (by selecting the control points or by using a curve).

Step 2. Set **Create Path** in the *Extrusion* dialog box.

Step 3. Click **OK** to finish the extrusion.

Step 4. To drive the path, select it in the Hierarchy view, and choose **Road/Drive Road**.

Note: The **Road** menu is only available if you have the Road Tools option.

Extrusion Preferences

The *Extrusion* dialog box contains the following controls:

Edge

Shows all the edges on the faces that were selected when the **Extrusion** tool was started. Edges can be selected singly or collectively in the *Extrusion* dialog box, so you can apply textures to them. Click the edge to which you want to apply a texture, then Shift+click (or click and drag) to select consecutive edges. Use Ctrl+click to select non-consecutive edges.

Index

Indicates the texture index number associated with the selected edges. Select an edge and then type in the Index number of the texture. A index number of -1 indicates that no texture in associated with the edge.

Map Texture

This list allows you to select one of four mapping types:

- **Per Polygon** restarts the texture on each polygon and repeats the texture within the polygon.
- **From Above** places the texture on the polygon similar to the Put texture tool. This is the only mapping type that enables **Reference Edge**.
- **Wrap UV** wraps the texture around and across the polygon while maintaining the aspect ratio of the texture.
- **Wrap V** wraps in the *v* direction (along the direction of the curve) while maintaining the aspect ratio of the texture.

Rotate

Allows you to specify the degree to rotate the texture as it is applied to the polygons. The range is from -180 to 180.

Offset U

Allows you to offset the texture in the ***u*** direction. The range is from -1 to 1.

Offset V

Allows you to offset the texture in the ***v*** direction. The range is from -1 to 1.

Reference Edge

Allows you to choose the edge with which to set the texture mapping. This feature only works with the **From Above** mapping type.

Use Real World Dimensions

Uses the textures actual size and repeats as many times as size allows.

Width

Used with **Use Real World Dimensions**, allows you to set the width of the texture.

Height

Used with **Use Real World Dimensions**, allows you to set the height of the texture.

Curve Type

Allows you to select one of four curve types: **Linear**, **Bezier**, **B-spline**, or **Cardinal**. You can change curve types anytime during the extrusion.

Num Subdivisions

Allows you to change the number of subdivisions (control points) for the curve. Only works with the **Bezier**, **B-spline**, or **Cardinal** curves, not **Linear** curves.

Create Path

Saves a path node with the database. The shape of the path is based on the extrusion method used. The path is shaped liked the extruded polygon or the curve that was used to create the extruded polygon. This path can be used by the realtime system or can be driven in the Graphics view (with the Road Tools option).

Pick Curve

Allows you to use an existing curve as a template to shape the extrusion.

Remove Last

Removes the last control point that you entered.

Insert Before

Inserts a control point in front of the currently selected control point.

Insert After

Inserts a control point after the currently selected control point.

Delete

Deletes the currently selected control point.

Subdivide

Subdivides the set of control points into the number of divisions specified in **Num Subdivisions**.

Flip Book Tool



The **Flip Book** tool, located in the **Geometry** toolbox, creates (or modifies) flip book style animations quickly. In OpenFlight, flip book animations can be represented by animated group nodes with each child of the group representing a different frame of the animations; these are useful for creating animated sequences. For the flip book animations created by this tool, each frame is represented by a single textured polygon attached to the animated group node. Each polygon (frame) can have a different texture pattern. Using the Flip book tool, you choose and order the texture patterns applied to each frame. You can also experiment with different settings and preview the animation before actually creating (or modifying) the geometry in the database.

To Create an Animation

Step 1. To create a new animation, first create a new group node, and attach a polygon. The group found above the polygon becomes the animated group to which the frames belong.

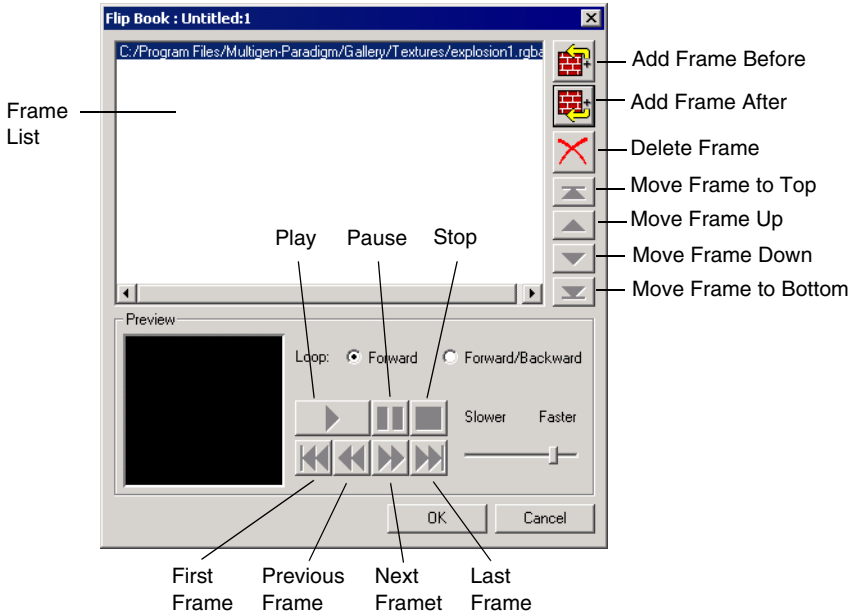
When you create a new animation, you should consider how the creation process will affect your texture coordinates. If the original polygon does

not have a texture applied, the tool will apply texture coordinates using a default 3-Point Put, oriented around the bounding box of the polygon. If the original polygon has a texture already applied, the vertices of each new polygon will inherit the texture coordinates of the original polygon. Therefore, you may find it preferable to apply a texture to the original polygon, before invoking the tool. If you do this, all new polygons created for other frames will have coordinates like those of the original polygon.

Step 2. With your polygon selected, choose the Flipbook Tool from the Geometry toolbox.



The Flipbook dialog appears, providing various options for your animation. You are now ready to add frames to your animation.

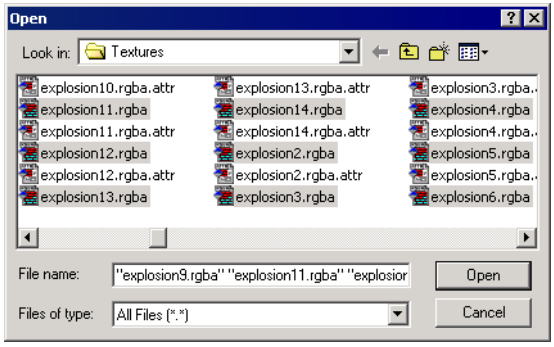


Step 3. Choose the **Add Frame Before** button or **Add Frame After** button to begin adding frames to your animation. The buttons activate a file chooser panel.

Note: If your original polygon does not yet have texture applied to it, you can choose *either* button to begin adding frames to your animation. If your original polygon already has a texture applied to it, the filename of the

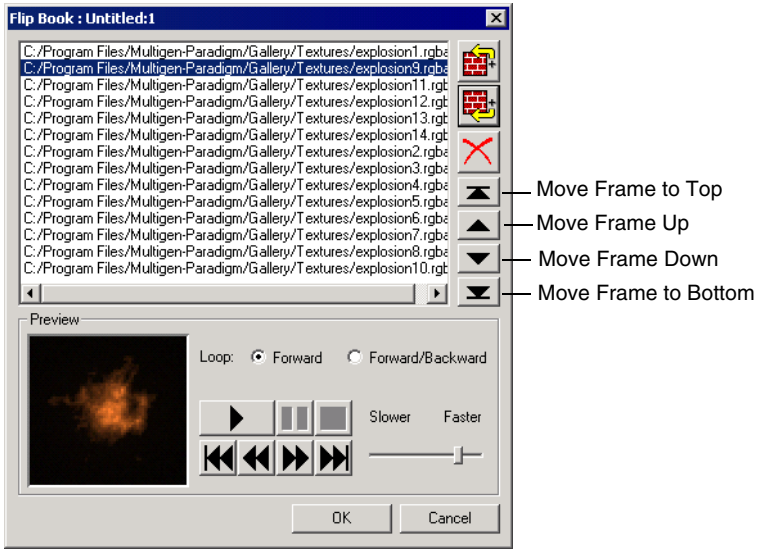
texture appears in the frame list; add frames before or after this original, using the corresponding button.

Step 4. Choose the texture (or textures) you want from the file chooser, and click **Open**. If you are using a series of textures, it is generally easier to choose an entire group now, and reorder them within the Flipbook dialog, than to select individual textures repeatedly.



(If you cannot find the expected texture files within your directory, choose **All Files** from the **Files of type** menu, to make sure all files are visible)

Step 5. Select individual textures in the frame list and use the arrow buttons to the right of the frame list to rearrange the textures for your animation.



If you opened an extended series of textures in the previous step, they may not appear in the proper sequence. The arrow buttons are for reordering the textures in your animation; the red "X" will delete unwanted textures.

Step 6. Choose the type of animation you want:

Forward – The first child of the animated group node is played first, followed by the second, third, etc. After the last child is played, the animation repeats again, beginning with the first child.

Forward/Backward – Like forward-looping, the animation begins by playing the first child, followed by the second, third, etc. After the last child, however, the animation continues by playing the animated group in reverse order.

Step 7. Adjust the speed of your animation, using the slider bar. The adjustments you make can be seen by using the preview controls to the left of the slider.

Step 8. Preview your animation, using the VCR-style control buttons to the left of the slider bar. The preview option is especially handy, because it allows you the opportunity to experiment and fine-tune your animation, before actually creating the geometry in the database.

Step 9. Choose **OK** to add the geometry to your database. The images corresponding to the frame list are added to the texture palette of you current database, and all the geometry is created.

After you have created an animation with the **Flip Book** tool, you can preview the animation by selecting the group node and choosing **Attributes/Animate**.

To display (or edit) the attributes of your animation group, double-click the group node in the Hierarchy view, or select the node and choose **Attributes/Modify Attributes**. The *Group Attributes* window appears.

You can also return to the **Flip Book** tool for modifications of an existing animation. Select the animated group node and choose the **Flip Book** tool. When you do this, the tool detects all the "frames" (textured polygons) below the animated group node, and lets you modify them with the same controls used for creating animations.

Billboard Wizard



The Billboard Wizard creates billboard models easily. Billboards are useful in a scene because they reduce the number of polygons needed to represent complex objects, such as trees. For more information, see the “Billboard Wizard Tutorial” on page 1293 and “Billboards” on page 380.

Tree Wizard



The Tree Wizard creates tree models from cross-hatched polygons. For more information, see the “Tree Wizard Tutorial” on page 1304.

Building Wizard

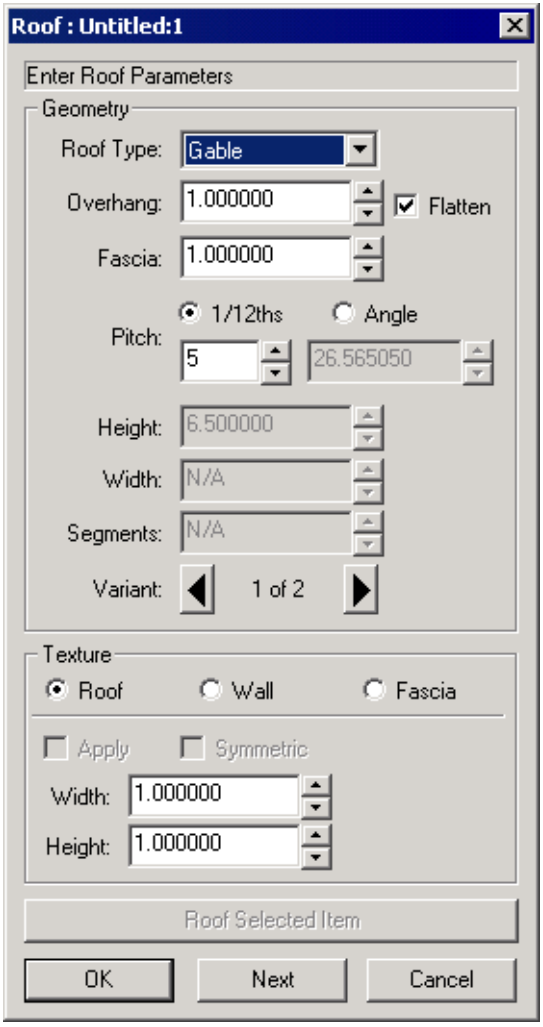


The Building Wizard creates models of houses (or other buildings) with varying types of roofs. For more information, see the “Building Wizard Tutorial” on page 1324.

Roof Tool



The **Roof** tool, located in the **Geometry** toolbox, creates a roof from a selected polygon. In the *Roof* dialog box, you choose from a number of architecturally-derived roof types, and enter parameters related to the specific type you have chosen.



The Roof tool is generally used to add a roof quickly to a building model already under construction. To use the Roof tool, you must first select a face on which to place the roof. If you do not already have a building under construction, and want to produce an

entire building quickly (including a roof), you can use the Building Wizard, which is also located in the *Geometry* toolbox. For information on the Building Wizard, see the "Building Wizard Tutorial" on page 1324.

To Create a Roof

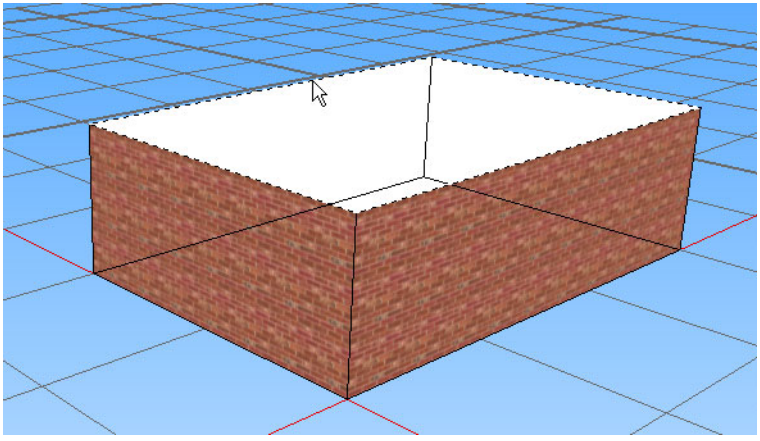
There are two tasks to perform with the Roof tool:

- Enter the Parameters for Roof Geometry.
- Apply Textures to the Roof.

Enter the Parameters for Roof Geometry

The top section of the Roof tool covers the parameters for roof geometry. Keep in mind that the values entered for the Roof tool are based upon the **Default Units** set in the **Info/ Preferences/Flight** panel. (The default setting is *meters*, unless you have changed your preferences.)

Step 1. In Face modeling mode, select a face to be roofed (the top of the building to be roofed).



In order to use the Roof tool, first choose a face be roofed, such as this polygon on the top of a building.



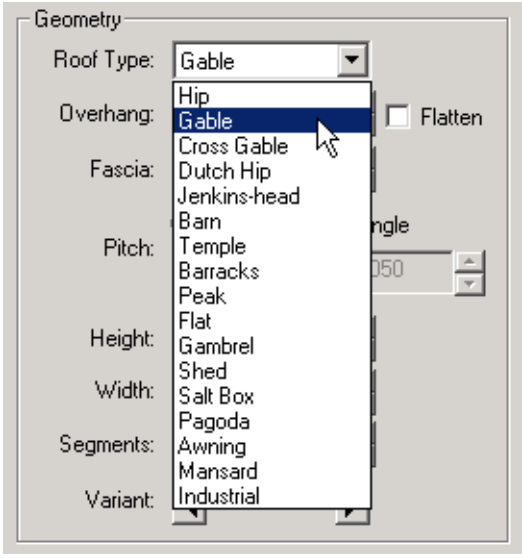
Step 2. Open the **Geometry Tools** toolbox.



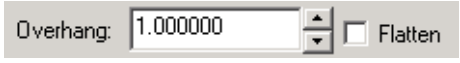
Step 3. Click **Roof** to open the *Roof* dialog box.

Step 4. Choose a **Roof Type**, such as **Gable**, from the *Roof* dialog box. (The roof types available to you are based upon the shape of the face you are using; not all roof types are available for every face.)

Depending on the type of roof you choose, various parameters become active. (Not all parameters apply to every type of roof.) As you enter parameters, the roof reflects your choices interactively.

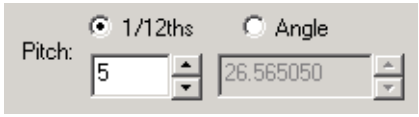


Step 5. Enter a value for **Overhang**. If you set the option for **Flatten**, the roof is raised so that the bottom of the roof is even with the top of the walls.

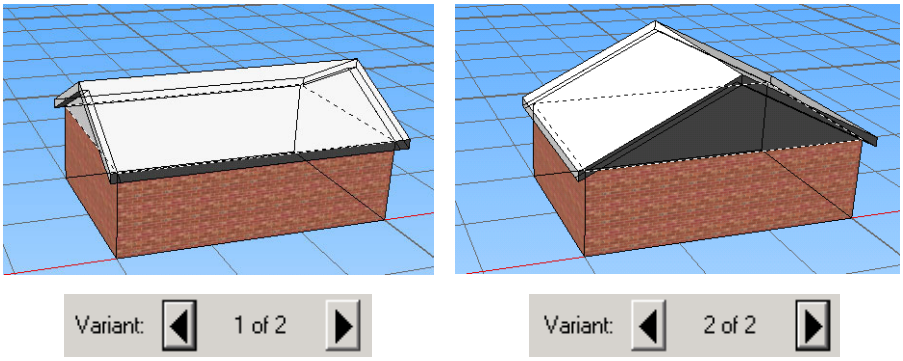


Step 6. Enter a value for **Fascia**.

Step 7. Enter a value for **Pitch**. You can use **1/12s** if you prefer that method; or you can choose the **Angle** option, and enter the value numerically.



Step 8. Enter any additional parameters which are required for the **Roof Type** you have chosen. For example, with a **Gable** roof, **Width** and **Height** are not active; yet these parameters are required if you choose a **Gambrel** roof. Additionally, if you wish to explore variations of a particular roof type, use the **Variant** arrows in the dialog box.

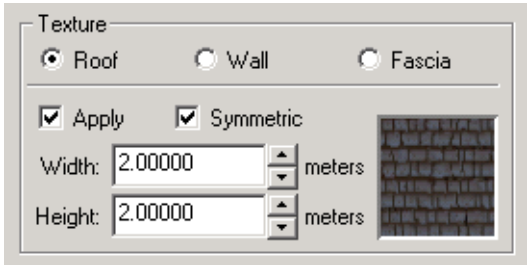


Variants of the Gable Roof Type

Apply Textures to the Roof

The lower section of the dialog box contains options for applying textures to different parts of your roof. The **Wall** and **Fascia** options will not always be available, depending on the **Roof Type** selected. Before proceeding with this part of the dialog box, open the *Texture Palette* window (if it is not already open), and load the textures you want to use

for your roof. (For details regarding the Texture Palette window, see “Texture Palettes” on page 422.)



Step 1. Choose **Roof** to add texture to the top of your roof.

Step 2. In the *Texture Palette*, choose the texture you wish to apply to the roof.

Step 3. Choose **Apply** in the Roof tool dialog box (if it is not already checked). Examine the roof after the texture is applied, to see if you wish to adjust how the texture relates to real world dimensions. If you wish to modify how the texture relates to real world dimensions, enter new values for **Width** and **Height**. When the **Symmetric** option is checked, you may enter a value for either **Width** or **Height**, and the other value will be changed symmetrically. (Remember, The real world dimensions for texture **Width** and **Height** are always measured in meters.)

Step 4. Choose **Wall** to add texture to the wall portion of your roof.

Step 5. In the *Texture Palette*, choose the texture you wish to apply to the roof.

Step 6. Choose **Apply** in the Roof tool dialog box (if it is not already checked). Examine the roof after the texture is applied, to see if you wish to adjust how the wall texture relates to real world dimensions. If you wish to modify how the texture relates to real world dimensions, enter new values for **Width** and **Height**.

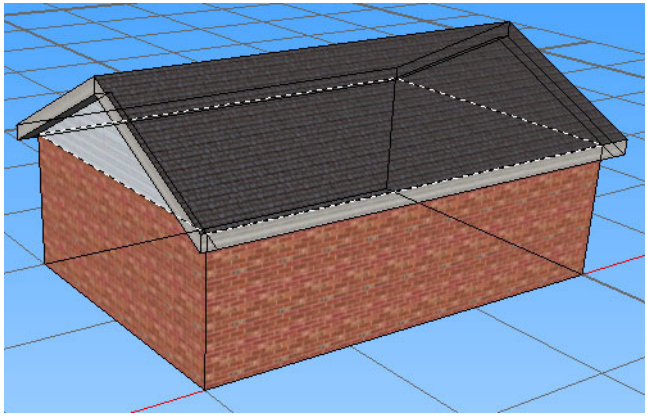
Step 7. Choose **Fascia** to add texture to the fascia of your roof, and follow the same procedure which you used for applying texture to other parts of the roof.

Step 8. In the *Texture Palette*, choose the texture you wish to apply to the roof.

Step 9. Choose **Apply** in the Roof tool dialog box (if it is not already checked). Examine the roof after the texture is applied, to see if you wish to adjust

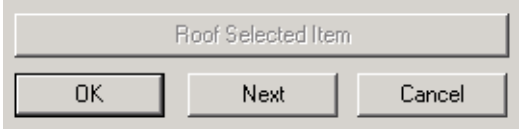
how the fascia texture relates to real world dimensions. If you wish to modify how the texture relates to real world dimensions, enter new values for **Width** and **Height**.

When you have completed these steps, your roof will be complete.



Concluding Actions

After your roof is complete, you can close the Roof tool, continue roofing other faces in your model, or cancel the operation entirely.



- Choose **OK** to quit the Roof tool.
- Choose **Next** to create another roof. When you choose **Next**, **Roof Selected Item** becomes active. Select the next face to be roofed, click **Roof Selected Item**, and begin entering the parameters for the new roof.
- Choose **Cancel** to cancel the roof operation.

True Type Text Tool



The **True Type Text** tool for Creator allows you to create 2D or 3D text in your database based on any True Type font loaded on your system. The text geometry created is similar to other geometry, so you can modify it with other geometry tools. To create geometry based on a True Type font, you select the font to use, enter the text, then specify its location in the database.

Unlike the 3D Text tool, which requires existing pre-built font geometry, this tool builds geometry on-the-fly, based on the font definitions contained in *any* True Type font on your computer. It reads the outlines defined in the True Type font and tessellates those outlines to create 2D geometry. Optionally, these outlines can be raised (extruded) to create 3D geometry.

To Create Geometry from True Type Text

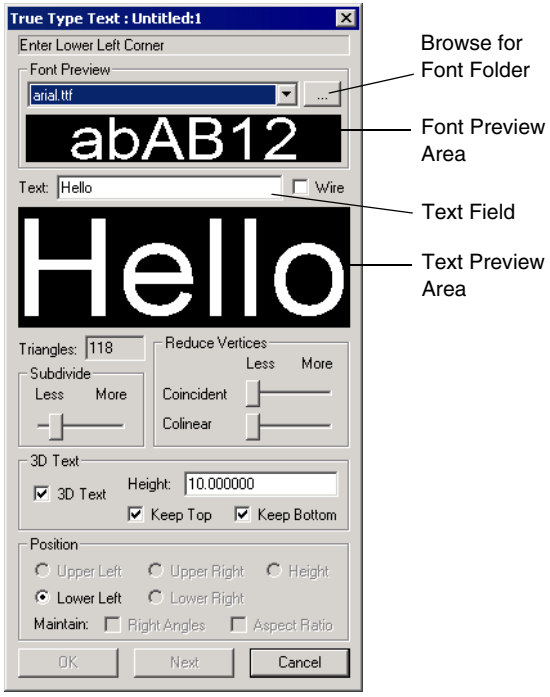
Step 1. Before starting the **True Type Text** tool, set the parent node to a group, object or polygon node. Depending on which kind of parent you select, the new geometry you create is attached in different ways.

If the parent you select is a group node, an object node will be created and attached to the parent group for each letter in the text string you specify. Each of these objects, in turn, will contain the polygons of that letter of the string. If you choose an object or polygon node as the parent node, all polygons of all the letters of the string will be created and attached directly to the parent object or polygon. Choosing a group or object node as parent is suitable for creating either 2D or 3D text, while a polygon parent is suitable for creating 2D text in the form of subfaces.

Step 2. With your polygon selected, choose the **True Type Text** tool from the Geometry toolbox.



A dialog is displayed in which you specify how you want the geometry for your text to be created.



Step 3. Select a True Type font from the pull-down menu in the **Font Preview** box.

The first time you start this tool after installing it, it tries to locate your system font directory. If successful, all True Type fonts located in this directory will be displayed in the menu in the **Font Preview** box. When you select one of the fonts listed, a preview of the font will be displayed in the **Font Preview Area** and any geometry already created will be re-created using the new font selected.

If the tool cannot locate the system font directory, or you want to load True Type fonts from a different directory, press the **Browse for Font Folder** button. After you select a folder from the chooser that is displayed, all the True Type fonts found in that directory are loaded in the Font Preview box.

Note: You can identify True Type font files by the suffix [.ttf].

Step 4. Enter a text string in the **Text** field.

The **Text Preview Area** below the **Text** field displays how the outlines of the letters will tessellate, given the current tessellation settings. You can preview the tessellated outlines in solid or wireframe. Select **Wire** for wireframe, or leave the box unchecked for solid. The **Triangles** field displays the number of polygons (triangles) that will be created for the current text string using the current tessellation settings.

Step 5. Specify tesselation settings.

For each letter in your string, the outline(s) for that character are read from the True Type font and tessellated to create triangles. When you create 2D text, these triangles correspond one-to-one to the polygons of the geometry created. When you create 3D text, these triangles form the tops and bottoms of your letters. The walls of the letters are formed by extruding the edges of the outlines between the tops and bottoms. In this way, interior polygons are automatically excluded from the 3D text geometry created.

By changing the tessellation settings, you can create more or fewer triangles. To create geometry at the desired resolution, it is important to understand how the outlines representing the letters in the True Type font are derived and then tessellated. For each letter:

- First, the outline(s) of the letter(s) are subdivided
- Next, coincident points on the resulting outline(s) are removed
- Next, collinear points on the resulting outline(s) are removed
- Finally, the resulting outline(s) is (are) tessellated into triangles

The tessellation settings you can control include how many times the original outlines are subdivided as well as the tolerance for calculating coincident and collinear vertices. You control these settings using the corresponding slider controls in the dialog. As you change these tessellation settings, the **Text Preview Area**, the **Triangles** field, and the geometry you are creating are all updated automatically.

Step 6. Apply color to the text you are creating.

The color of the text is determined by the color presently selected in the Color Palette. If you wish to change the color, simply open the Color Palette, and select a new color.

Step 7. Specify 2D or 3D text.

If you want to create 3D text, choose **3D Text**. When creating 3D text, you specify whether or not you want to keep the tops and/or bottoms of the 3D text by using the corresponding checkboxes. You can also specify the height of the extruded walls by entering an explicit value into the **Height** field or by dragging the height construction edge.

Note: If you specify a positive height value, the polygons of the 3D text created will face "outward". If you specify a negative height value, the polygons will face "inward". In this way (if you also choose to not keep tops) you can create text that appears "etched in". Of course, if this is not desirable, you can use the Reverse Face tool to reverse the orientation of the geometry.

Step 8. Position the text geometry.

To position the geometry, you specify the bounding area (2D text), or volume (3D text), inside which the geometry will be built. For 2D text, this area is a parallelogram for which you can specify each corner. For 3D text, you also specify a parallelogram which bounds the footprint of the 3D text created. In addition, you specify a height value which completes the definition of the volume inside which the geometry will fit.

For both 2D and 3D text, the parallelogram is specified by four points, the **Lower Left**, **Lower Right**, **Upper Right** and **Upper Left** corners. A construction vertex is displayed at each corner of the parallelogram, and a construction polygon (wireframe) is displayed around the parallelogram itself. For **3D Text**, an additional construction edge is displayed representing the **Height** of the extruded geometry, as well as several additional construction polygons (wireframe) representing the sides of the volume. These construction polygons come in handy for edge referencing while positioning the geometry. The construction elements are displayed using the following colors:

Construction Element	Type	Color
Lower Left Corner	Vertex	Red
Lower Right Corner	Vertex	Green
Upper Right Corner	Vertex	Blue
Upper Left Corner	Vertex	Yellow

Construction Element	Type	Color
Height	Edge	Red
Sides of Parallelo-gram Area or Volume	Polygons	Red

When you are ready to position the geometry, the first point you enter in the graphic view will be the lower left corner. After you specify this point, the tool creates the other points automatically for you and displays the construction geometry. At this point you can specify any of the other points. The state of the **Position** radio buttons at the bottom of the dialog reflects which point the tool is expecting you to enter. If you want to enter a different point, simply select the radio button corresponding to the point you want to enter. As an alternative, hold down the CTRL key while clicking the left mouse button on the construction vertex you want to move. This will select that point to move.

Note: When you are creating 3D text, you can CLTL+click on either end of the height construction edge to select it.

You can control how the parallelogram is formed using the **Maintain Right Angles** and **Maintain Aspect Ratio** check boxes. If **Maintain Right Angles** is selected, the corners of the parallelogram are adjusted to form a rectangle when moving the lower right, upper right or upper left corners. If **Maintain Aspect Ratio** is selected, the dimensions of the parallelogram are adjusted such that the aspect ratio of each letter defined in the True Type font definition is preserved. When **Maintain Aspect Ratio** is selected and you change either the width or the height of the geometry by moving the lower right or upper left corner, respectively, the "other" dimension is automatically adjusted to preserve the aspect ratio of the text.

Step 9. Adjust individual letters.

Normally, when you change the tessellation settings or choose a new color from the color palette, the changes are applied to the geometry of all the letters of your text string simultaneously. In this way, your letters are all created at the same resolution using the same color. There may be times, however, when you want to create individual letters at different resolutions or using different colors. To allow this, you can "freeze" individual letters of your string. Frozen letters are not re-tessellated or re-

colored when you change the tessellation settings or choose a new color in the palette. By freezing a letter, you can "lock" the tessellation and color of the corresponding geometry.

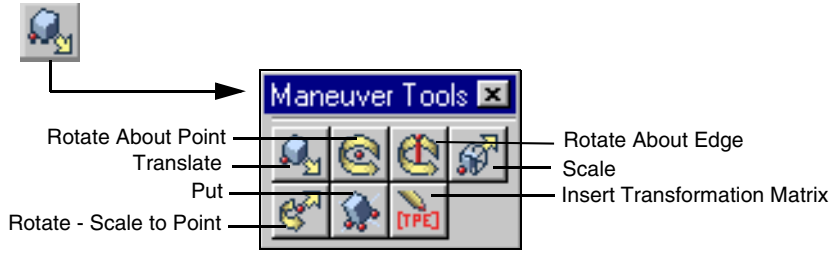
To freeze a letter of your string, click on it with the left mouse button in the **Text Preview Area**. Letters that are frozen appear with a gray background. To unfreeze a letter, click on it again.

Note: When you unfreeze a letter, it is immediately re-tessellated and re-colored to reflect the current settings. Also, all letters are repositioned and/or resized when you move any of the corner points or enter a new height - even those that are frozen.

Step 10. Choose **OK** to add the geometry to your database, **Next** to create a new text string, or **Cancel** to quit the tool without adding the geometry to your database.

Maneuver Tools

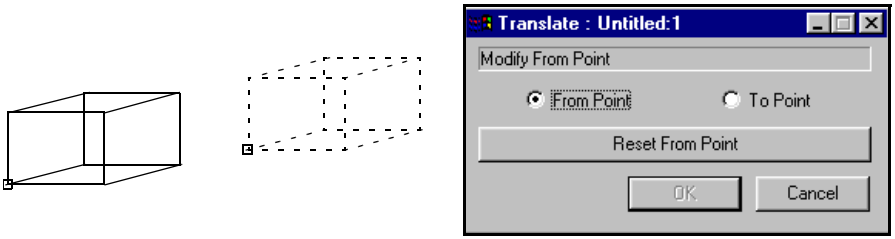
Maneuver tools move, scale, and rotate geometry in the database.



Translate Tool



Translate (Shift+T) moves selected items in the database. It is often useful to turn on one or more **Freeze Flags** (see “Freeze Flags” on page 77) to constrain the direction of translation.



Translate moves each selected item by changing the coordinate assigned to each vertex, which means that it changes the geometry of the item. To reposition an item without changing its geometry, place a transformation matrix in the database tree. See "Insert Transformation Matrix Tool" on page 157 for more information about placing a transformation matrix in a database.

To Translate an Item

Step 1. Set the appropriate modeling mode and select the item or group of items to be translated.

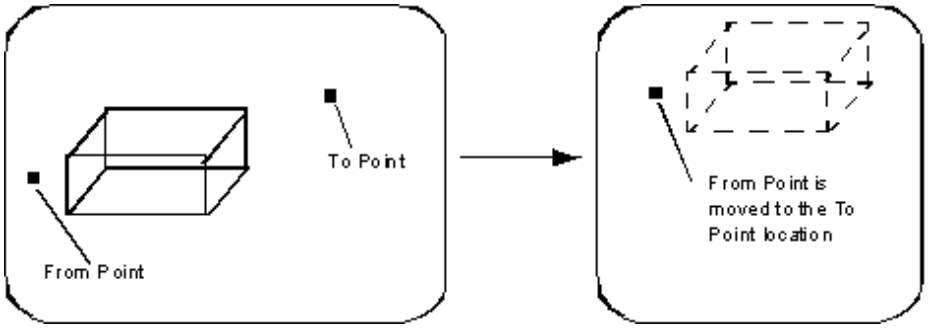
Step 2. Open the **Maneuver** toolbox and click **Translate**. The *Translate* dialog box opens and a **From Point** reference point appears on the selected item. The **From Point** defines the origin of the translation. You can change it by selecting a different position anywhere in the Graphics view.



Note: The **From Point** does not have to be located on the selected item. The vector between **From Point** and **To Point** determines the distance and direction of the translation.

If you want to change the location of the **From Point**, click **Reset From Point** to change it back to its default position at the center of the selected item or group of items.

Step 3. Select **To Point**, if it is not already selected, and then click the new position. The item moves so that the reference point selected in the previous step is placed at the new position. You can change the location by selecting a different position.

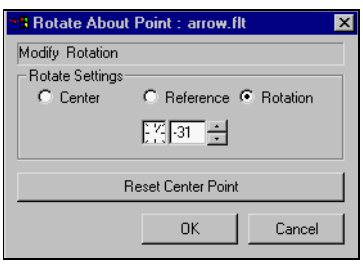


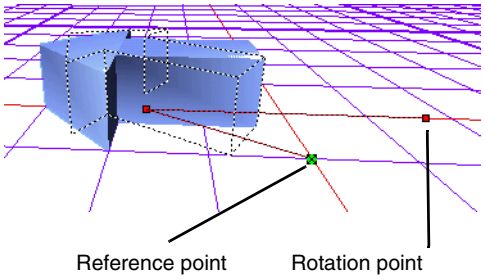
Step 4. Click **OK**.

Rotate About Point Tool



Rotate About Point (Shift+R), in the **Maneuver** toolbox, rotates the selected item around a point using an axis perpendicular to the tracking plane.





To Rotate Geometry About a Point

Step 1. Select the geometry to be rotated.

Step 2. Set the tracking plane to define the axis of rotation. The axis is perpendicular to the tracking plane.



Step 3. Open the **Maneuver** toolbox.

Step 4. (Click **Rotate About Point** Shift+R).

Step 5. To change the center of rotation, click **Center** in the *Rotate About Point* dialog box and select a new point in the Graphics view.

Step 6. Click **Rotation** to select the **Rotation** point in the Graphics view.

Step 7. Drag the pointer in the Graphics view or use the clock to rotate the geometry.

Step 8. When you are satisfied with the rotation, click **OK**.

Rotate About Point Parameters

Set the Rotate About Point parameters to rotate an item around a point. The *Rotate About Point* dialog box contains the following parameters:

Center Point

Selects the center of rotation for editing.

Rotation

Selects the **Rotation** point for editing. Rotate the item by dragging the pointer in the Graphics view or editing the degrees of rotation in the clock.

Reference

Selects the **Reference** point for editing. The angle of rotation is defined by the angle between the **Reference** point, **Center** point, and **Rotation** point.

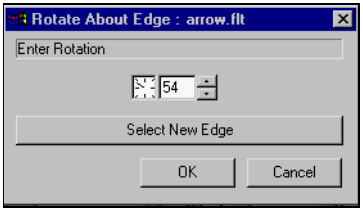
Reset Center Point

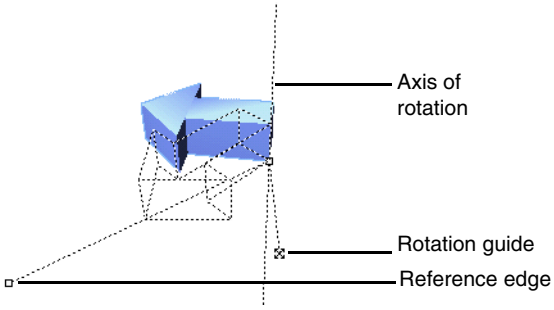
Resets the **Center** of rotation to its default position.

Rotate About Edge Tool



Rotate About Edge, in the **Maneuver** toolbox, rotates the selected item around an edge you select in the database. The *Rotate About Edge* dialog box prompts for the edge, then displays the axis and a graphical rotation guide.





To Rotate Geometry About an Edge

Step 1. Select the geometry to be rotated.



Step 2. Open the **Maneuver** toolbox.

Step 3. Click **Rotate About Edge**.

Step 4. Select the edge around which you want to rotate the geometry.

Step 5. Drag the guide to rotate the geometry, or use the clock to specify an angle of rotation. Positive numbers rotate the geometry clockwise; negative numbers rotate the geometry counterclockwise.

Step 6. When you are satisfied with the rotation, click **OK**.

Rotate About Edge Parameters

Set the Rotate About Edge parameters to rotate an item around an edge. The *Rotate About Edge* dialog box contains the following parameters:

Rotation Clock

Controls the number of degrees of rotation. You can also rotate the item by dragging the Rotation Guide in the Graphics view.

Select New Edge

Discards the current rotation and starts over.

Scale Tool



Scale (Shift+S), in the **Maneuver** toolbox, changes the size of selected items. You can scale the item as a whole, or scale it independently along the *x*, *y*, or *z* axis.

You can specify an amount to scale, or specify a size to scale to. The amount to scale is multiplied with the original size to determine the new size.

To Scale an Item

Step 1. Select an item in the database.



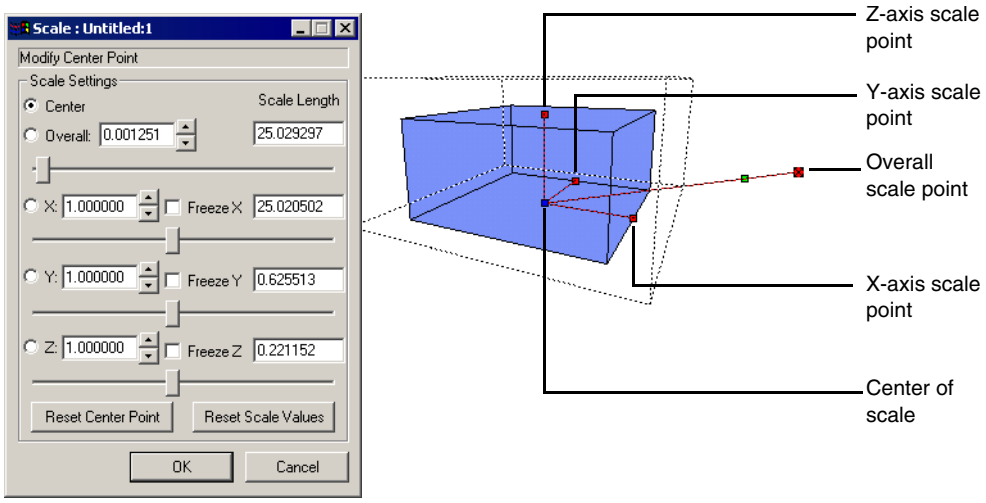
Step 2. Open the **Maneuver** toolbox.



Step 3. Click **Scale**.

Step 4. If you want to change the default center of scale, click **Center** and enter or select a new coordinate in the Graphics view.

The vertices of the selected item move away from the center if the scale is greater than one and move towards the center if the scale is less than one.



Step 5. To scale the item in all three dimensions, edit the **Overall Scale** value, enter a desired size in the **Scale Length** field, or click **Overall Scale** and resize the item by dragging the pointer in the Graphics view.

Step 6. To scale the item relative to one axis at a time, click **X**, **Y**, or **Z** to select the corresponding **Scale** point. Then, resize the item by either dragging the pointer in the Graphics view, entering a scale value, or by entering a desired size in the **Scale Length** field.

Step 7. When you are satisfied with the scale, click **OK**.

Note: If you entered values, you may need to click the Tab key for the change to occur.

Scale Parameters

Set Scale parameters to scale an item. The *Scale* dialog box contains the following parameters:

Center Point

Selects the center of the scale for editing in the Graphics view. The vertices of the selected item move away from the center when the scale is greater than one and move towards the center when the scale is less than one.

Overall Scale

Resizes the item along the *x*, *y* and *z* axes.

You can also scale the item in all dimensions by selecting **Overall Scale** and dragging the mouse in the Graphics view.

Scale Length

A size that you can specify to scale to. This value is the length from the **Center** point to a scale point.

X, Y, and Z

Scale the item in one dimension at a time.

You can also scale the item in one dimension by clicking **X**, **Y**, or **Z** (to select the corresponding **Scale** point) and dragging the pointer in the Graphics view.

Freeze X, Y, and Z

Freezes the item in the corresponding dimension.

Reset Center Point

Resets the center of scale to its default position.

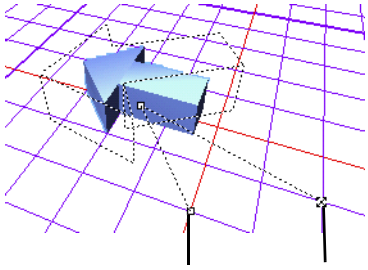
Reset Scale Values

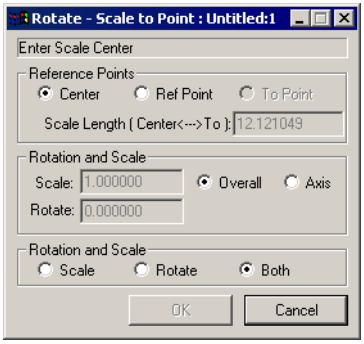
Resets each Scale point to its original (unscaled) position.

Rotate - Scale to Point Tool



Rotate-Scale to Point, in the **Maneuver** toolbox, scales and rotates selected geometry. The axis of rotation is perpendicular to the tracking plane.





To Rotate - Scale to Point

Step 1. Select the geometry to be rotated and scaled.

Step 2. Set the tracking plane to define the axis of rotation. The axis is perpendicular to the tracking plane.



Step 3. Open the **Maneuver** toolbox.

Step 4. Click **Rotate-Scale to Point**.

Step 5. In the *Rotate-Scale to Point* dialog box, select **Scale**, **Rotate**, or **Both** to control whether the geometry is scaled, rotated, or both scaled and rotated.

Step 6. To change the default center of rotation, click **Center** and enter a new point in the Graphics view.

Step 7. Click **Ref Point** and enter a **Reference** point in the Graphics view. The **Reference** point will be scaled and rotated to match the **To** point you enter in the next step.

Step 8. Click **To Point** and enter a **To** point in the Graphics view, or specify a desired size to scale to in **Scale Length**.

Step 9. You can change the amount of rotation and scale by dragging the pointer in the Graphics view, or by editing the **Scale** and **Rotate** values in the *Rotate-Scale to Point* dialog box.

Step 10. When you are finished, click **OK**.

Rotate-Scale to Point Parameters

Set the Rotate-Scale to Point parameters to scale and rotate an item around a point. The *Rotate-Scale to Point* dialog box contains the following parameters:

Center

Selects the **Center** point of scale and rotation for editing.

Ref Point

Selects the Reference point for editing. The angle of rotation is defined by the angle between the **Reference** point, **Center** point, and **To** point. The distance between the **Center** point and the **Reference** point defines the unit scale.

To Point

Selects the **To** point for editing. This point defines the amount of rotation and the amount of scale applied to the original item.

Scale Length

A size that you can specify to scale to. This value is the length from the **Center** point to the **To** point.

Scale

Displays the amount of scale applied to the selected item. To change the scale, edit this text or select the **To** point and drag the pointer in the Graphics view. The **Overall/Axis** buttons control whether **Scale** is applied in all three dimensions or only along the line from the **Center** point to the **Reference** point.

Scale: Overall / Axis

Control the effect of **Scale**. **Overall** scales the selected item in all three dimensions. **Axis** scales the geometry only along the line from the **Center** point to the **Reference** point.

Rotate

Displays the rotation applied to the selected item. To change the rotation, edit this text or select the **To** point and drag the cursor into the Graphics view.

Scale / Rotate / Both

Selects whether the geometry is scaled, rotated, or scaled *and* rotated with this tool.

Put Tool



Put, in the **Maneuver** toolbox, simultaneously translates, rotates, and scales a selected item by mapping three **From** points on the item to three **To** points in the database.

To Put an Item

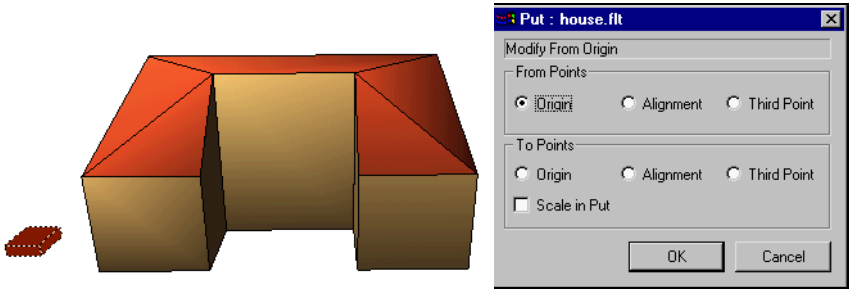
Step 1. Select the item to be repositioned.



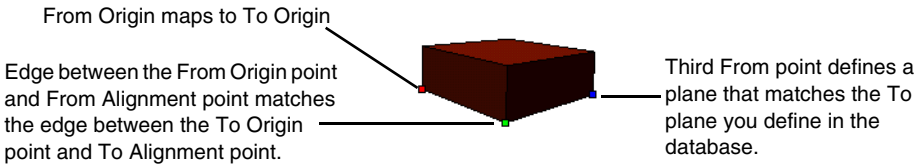
Step 2. Open the **Maneuver** toolbox.

Step 3. Click **Put**. The *Put* dialog box appears to prompt you for three **From** points on the selected item, and three **To** points in the database.

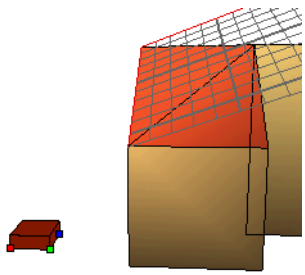




Step 4. Pick the **From** points: **Origin**, **Alignment**, and **Third point** on the selected item. These points line up with **To** points you enter in the next step.

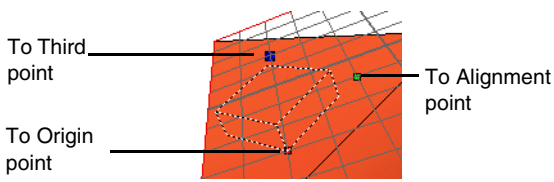


Step 5. Set the tracking plane for entering **To** points. (You can also enter **To** points by middle-clicking existing vertices.)



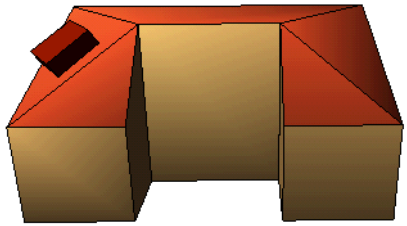
Step 6. Set **Scale in Put** if you want to use the **Alignment** point to rescale the selected item.

Step 7. Enter three **To** points.



Scale in Put checkbox cleared

Step 8. When you are satisfied with the new position of the item, click **OK**.



Put Parameters

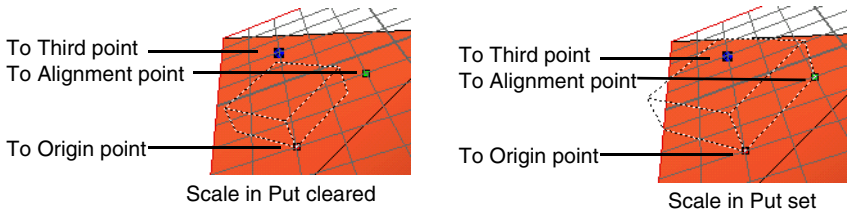
Set Put parameters to reposition an item. The *Put* dialog box contains the following parameters:

From and To Points

- **Origin** is a reference point. The **From Origin** point on the selected item matches the **To Origin** point you enter in the database.
- **Alignment**, together with the **Origin** point, defines an alignment edge. The **From** edge on the selected item lines up with the **To** edge you enter in the database. If **Scale in Put** is set, the selected item is rescaled so the **Alignment** points exactly match.
- **Third Point** defines an alignment plane. The **From** plane on the selected item matches the **To** plane you specify in the database.

Scale in Put

Rescales the selected item by exactly matching the **From Alignment** point on the selected item with the **To Alignment** point you enter in the database. If **Scale in Put** is clear, the selected item is not rescaled.



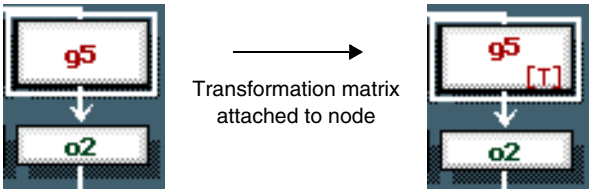
Insert Transformation Matrix Tool



The **Insert Transformation Matrix** tool in the **Maneuver** toolbox places a transformation matrix on nodes selected at or above object level in the database hierarchy.

A *transformation* is a method for changing the position or orientation of an object by attaching a *transformation matrix* to the node. This transformation, for example a translation or scaling, is applied to the node and all of its descendants in the hierarchy. This is a convenient way to modify the position of several objects at once without individually modifying each of their coordinates.

If a node has a transformation matrix attached to it, the node is shown with a letter that indicates the type of transformation. For example, a **T** for **Translate**.



Other transformations are also shown with letters, as follows:

- **S** for **Scale**
- **E** for **Rotate Edge**
- **R** for **Rotate Point**
- **P** for **Put**

- **p** for **To Point**
- **M** for **General Matrix**

More than one transformation is possible for each node; a letter is added to the node for each transformation.

To Apply a Transformation Edit

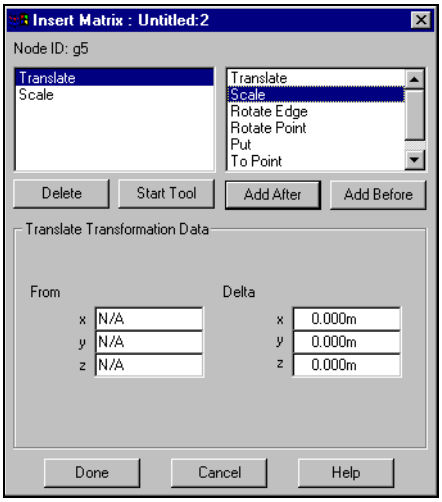
Step 1. Select the node to which you will apply the transformation edit.



Step 2. Open the **Maneuver Tools** toolbox.



Step 3. Select **Insert Transformation Matrix**.



Step 4. Select a transformation type from the list in the upper-right corner of the *Insert Matrix* dialog box.

Step 5. Click **Add After** or **Add Before** to move the transformation to the edit list on the left. If you are including more than one transformation, use **Add After** or **Add Before** to add each transformation to the edit list in the correct order. Each transformation that you add affects the final transformation. For example, **Rotate** followed by **Translate** is not the same as **Translate** followed by **Rotate**.

Note: To change the order in the edit list, use the **Delete** button, and then add the transformation again using **Add After** or **Add Before** to achieve the correct order. You can reuse each transformation as many times as necessary.

Step 6. Select a transformation in the edit list and click **Start Tool**. The appropriate maneuver tool is displayed.

If you prefer to enter the transformation numerically, edit the values shown in the lower part of the *Insert Matrix* dialog box instead of clicking **Start Tool**.

Step 7. Choose the appropriate settings in the tool dialog box.

Step 8. Repeat the previous step for each transformation in the edit list.

To Edit an Existing Transformation

Step 1. Select the node with the transformation.



Step 2. Open the **Maneuver Tools** toolbox.



Step 3. Click **Insert Transformation Matrix**.

Step 4. Edit the transformation as described in "To Apply a Transformation Edit" on page 158.

To Convert a Transformed Node to a Regular Node

Select the node and choose **Local-DOF/Make Geometry**. The node and its children in the hierarchy will be converted (all transformations are permanently applied). There is no **Undo** for this command (see "To Remove Geometry from a Local Coordinate System" on page 367).

Duplicate Tools

Tools in the **Duplicate** toolbox place copies of selected geometry throughout the database. The copies can be placed singly, along a line, scattered, or in a random pattern.



Duplicate Tool



Duplicate (Shift+D) copies a selected item or group of items and translates the copy from the **From Point** to a specified **To Point**. Nodes for the copied geometry are created under the same parent as the original geometry.

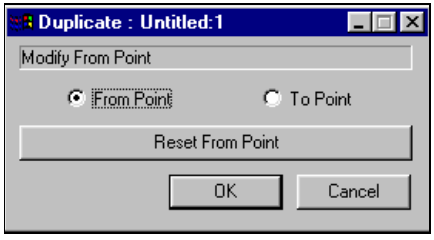
To Duplicate an Item

Step 1. Set the modeling mode for the geometry to be duplicated.

Step 2. Select the item to be duplicated.



Step 3. Open the **Duplicate** toolbox and click **Duplicate**. The *Duplicate* dialog box opens with the default **From Point** displayed.



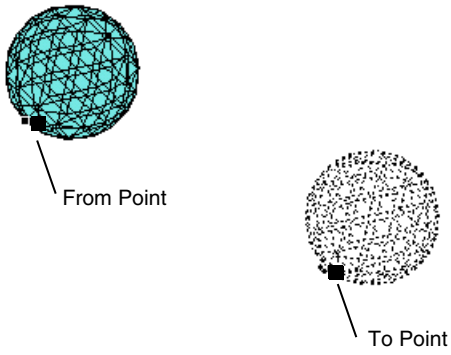
The **From Point** defines the origin of translation for the duplicate geometry.

Step 4. Select a new **From Point** in the Graphics view,

To reset **From Point** to its default, click **Reset From Point**.

Step 5. Select **To Point** in the *Duplicate* dialog box, if it is not already selected, and then in the Graphics view click the location where a duplicate geometry is to be placed. The vector between the **From Point** and the **To Point** determines the distance and direction of translation.

Step 6. Click **OK**. A copy of the selected item is placed so that the **From Point** coincides with the **To Point**.

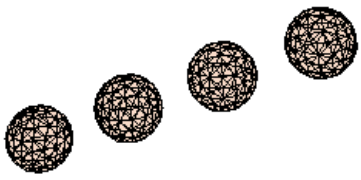


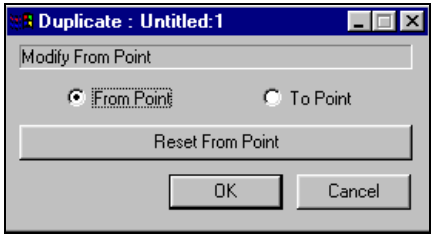
To Create a Sequence of Evenly-Spaced Duplicates

Step 1. Position the first copy, and keep the *Duplicate* dialog box open.

Step 2. Click the **Duplicate** tool again. Each time you click **Duplicate**, another copy is inserted in the same direction, with the same offset, as the first copy.

Step 3. Click **OK**.





Replicate Tool



The **Replicate** tool creates a string of copies of a selected item. If the selected item is a group, the copies are created by placing a transformation on the group node and storing the number of replicates in its **Replication** attribute. Otherwise, replicates are created by inserting new copies of the selected nodes under the same parent as the original geometry.

Replicates appear only once in the database hierarchy, and cannot be transformed individually. When you select a replicate, it is displayed with a red outline in the Graphics view.

To Create Replications

There are two ways to replicate a node. The first method uses the *Replicate* dialog box to control the number of replications.

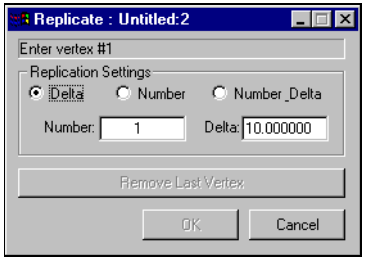
Step 1. In the Hierarchy view, select the node that you want to replicate.



Step 2. Open the **Duplicate** toolbox.



Step 3. Click **Replicate**.

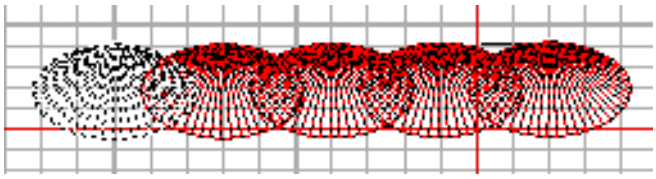


Step 4. Set the **Replication Settings** to **Delta**, **Number** or **Number_Delta**.

Delta controls the spacing between replicates using the distance supplied by **Delta**. Creator inserts as many copies of the original as fits between each pair of vertices.

Number controls the number of replicates. The specified number of replicates is placed evenly between the vertices.

Number_Delta controls the number of replicates, and the spacing between them. When **Number_Delta** is selected, **Number** determines the number of replications and **Delta** determines the distance between replicates. The two control points define a line along which replicates are inserted.



Step 5. If **Number** or **Number_Delta** was set, enter the number of replicates to be created in the **Number** field.

If **Delta** or **Number_Delta** was set, enter the distance between replicates in the **Delta** field.

Step 6. In the Graphics view, select the first vertex for placing the replicates.

Note: To reposition a vertex, click **Remove Last Vertex** and click a new location.

Step 7. Select a second vertex. You must select at least two vertices, but can select more.

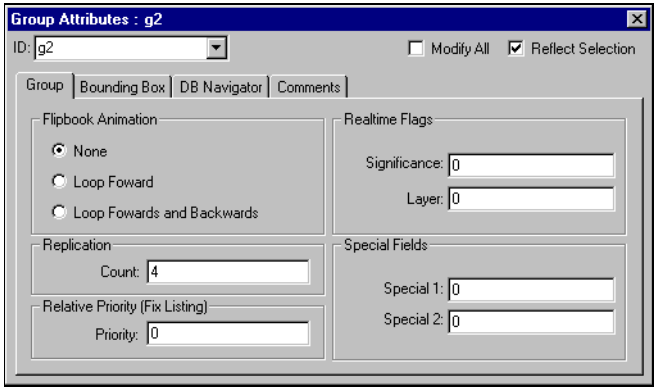
Step 8. Click **OK** to replicate the item or **Cancel** to quit the Replicate operation.

You can also replicate a node using the *Group Attributes* window to control the number of replications.

Step 1. Click the group node that you want to replicate.

Step 2. Put a transformation matrix on the node, using the procedure described in "To Apply a Transformation Edit" on page 158.

Step 3. Double-click the group node to open the *Group Attributes* window.



Step 4. Click the **Group** tab and enter the number of replications in the **Replication** field.

This attribute is the number of copies displayed in addition to the original group. Creator applies the transformation matrix to the original group to produce the first copy, to the first copy to produce the second copy, and so on.

To Convert a Replicated Group to a Set of Objects

Select the group and choose **Local-DOF/Make Geometry** (see "To Remove Geometry from a Local Coordinate System" on page 367).

Populate Tool



The **Populate** tool takes the model you previously copied to the Graphics Buffer and randomly distributes as many copies as you specify over selected terrain polygons.

To Use the Populate Tool

Step 1. Open the database of the model you want to place on the terrain.

Step 2. Make sure the center of the model is at the origin (0,0,0). A model whose center is not at the origin will be offset, and may not populate the selected

polygons. (See “Centering a Model at the Origin” on page 352 if you need to correct the feature’s position.)

Step 3. Select the feature and choose **Edit/Copy (Ctrl+C)**.

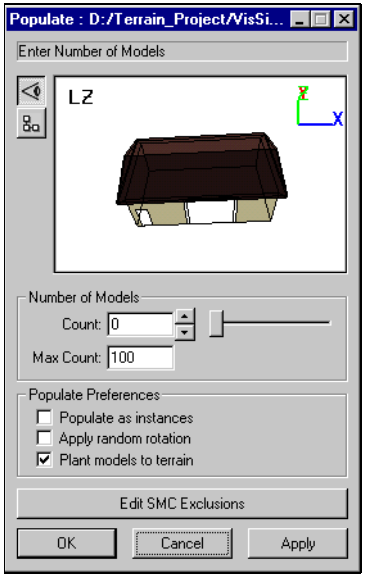
Step 4. Select **Face** mode in the Toolbar, and then select the terrain polygons that you want to populate. Only polygons that have the Terrain attribute set can be populated. See “Face Attributes” on page 840.



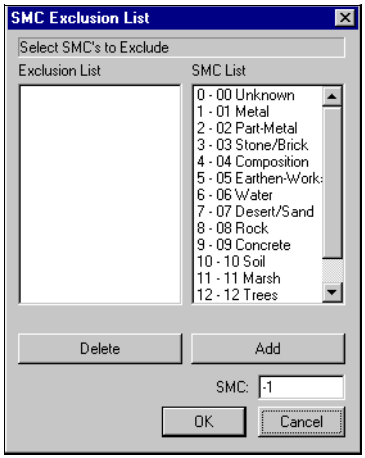
Step 5. Open the **Duplicate** toolbox.



Step 6. Click the **Populate** tool. The *Populate* dialog box appears.



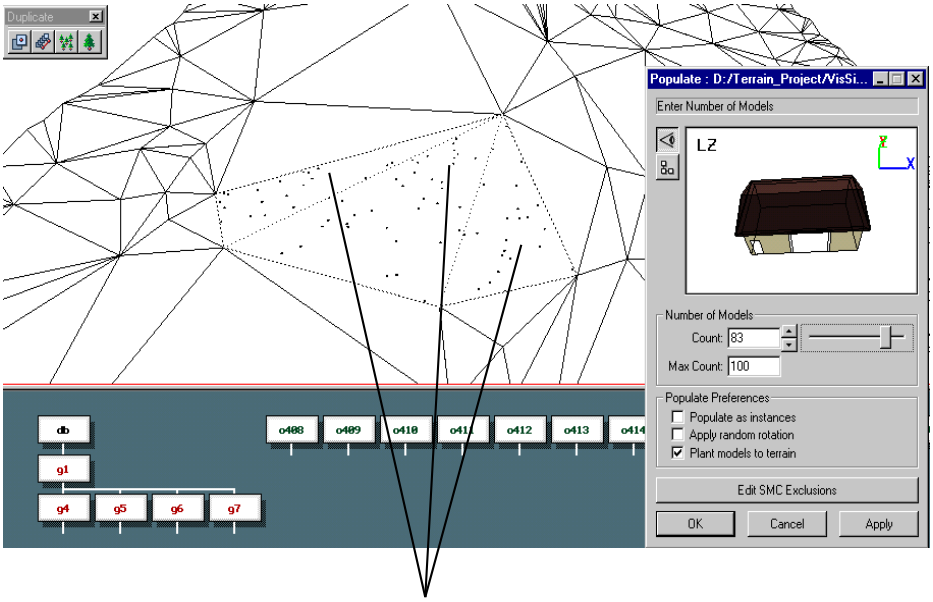
Step 7. If you selected a large terrain area, you can exclude terrain polygons that have specific Surface Material Codes. (For example, you can prevent trees from being placed on polygons that have asphalt or concrete SMCs.) To do this, click **Edit SMC Exclusions**. The *SMC Exclusion List* dialog box appears.



Select items from the list and click **Add**. When you have selected all the SMCs you want to exclude, click **OK** to return to the *Populate* dialog box.

Step 8. Use the slider to specify the *total* **Number of Models** you want to place on the selected terrain polygons, and then click **Apply**.

For example, in the following figure, 82 models are randomly distributed across the three selected terrain polygons. The number of models in the Graphics view changes dynamically as you move the slider, so you can see the effect of changing the **Number of Models** value. To limit the range of the slider, enter a value in the **Maximum Number of Models** field.



The 82 models specified in the dialog box are randomly distributed across the selected polygons

Step 9. To include a single copy of the model in the database and then reference each occurrence as an instance, set **Populate as instances**. When this feature is cleared, the geometry of the model is inserted as many times as you specified.

Populate as instances saves polygons, but it does not allow you to conform each model to the terrain, and it requires extra calculations by the realtime system. If you are doing an air simulation, this is a good choice; if you are doing a ground simulation, the effects will be noticeable if you use instances, because you cannot plant the models on the terrain.

Step 10. To rotate each model to a different, random orientation, set **Apply random rotation**.

Step 11. To force the bottom of each model to conform to the terrain on which it rests, set **Plant models on terrain**. (This option is not available if you choose **Populate as instances** because the geometry of a model cannot be adjusted for each instance.)

Step 12. Click **OK**.

Scatter Tool



The **Scatter** tool takes the model you previously copied to the Graphics Buffer and places a copy of that model wherever you click on the terrain.

To Use the Scatter Tool

Step 1. Make sure the center of the feature is at the origin (0,0,0). A feature whose center is not at the origin will be offset, and may not populate the selected polygons. (See "Centering a Model at the Origin" on page 352 if you need to correct the feature's position.)

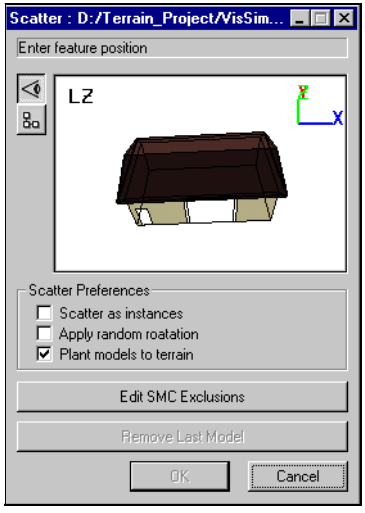
Step 2. Select the feature and choose **Edit/Copy (Ctrl+C)**.

Step 3. Make sure you are in Face mode, and then select the terrain polygons on which you want to scatter models. Only polygons that have the Terrain flag set in their attributes can be populated using **Scatter**.

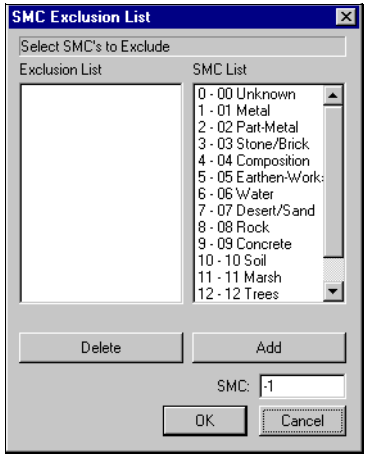
Step 4. Select **Top** from the **Eyepoint** menu in the Toolbar, to place the eyepoint directly above the terrain. This allows you to place points with accuracy.



Step 5. Click the **Scatter** tool. The *Scatter* dialog box appears.



Step 6. If you selected a large terrain area, you can exclude terrain polygons that have specific Surface Material Codes. For example, you can prevent trees from being placed on polygons that have asphalt or concrete SMCs. To do this, click **Edit SMC Exclusions**. The *SMC Exclusion List* dialog box appears.



Select items from the list and click **Add**. When you have selected all the SMCs you want to exclude, click **OK** to return to the *Scatter* dialog box.

Step 7. To include a single copy of the model in the database and then reference each occurrence as an instance, set **Scatter as instances**. If this feature is cleared, the geometry of the model is inserted into the database each time you click the mouse.

Scatter as instances saves polygons, but it does not allow you to conform each model to the terrain, and it requires extra calculations by the realtime system. If you are doing an air simulation, this is a good choice; if you are doing a ground simulation, the effects will be noticeable if you use instances, because you cannot plant the models on the terrain.

Step 8. To rotate each model to a different, random orientation, set **Apply random rotation**.

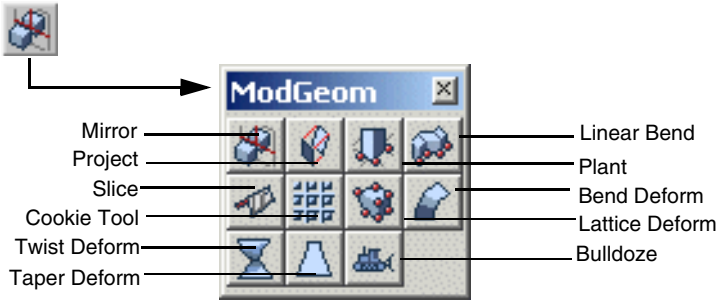
Step 9. To force the bottom of each model to conform to the terrain on which it rests, set **Plant models on terrain**. (This option is not available if you choose

Scatter as instances because the geometry of a model cannot be adjusted for each instance.)

Step 10. Click points on the selected terrain polygons to place copies of the model. When you are finished, click **OK** in the *Scatter* dialog box.

Modify Geometry Tools

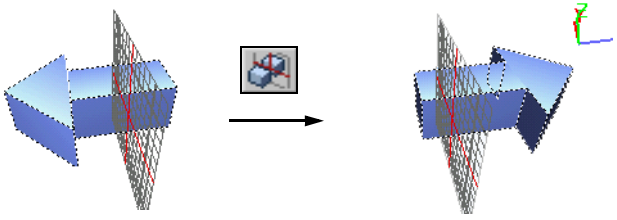
Tools in the **Modify Geometry** toolbox alter the shape of the geometry along an axis.



Mirror Tool

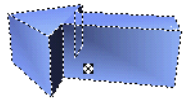


The **Mirror** tool, located in the **Modify Geometry** toolbox, flips the selected item across the tracking plane.

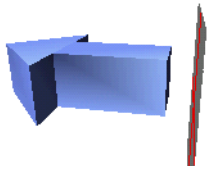


To Mirror a Copy

Step 1. Select one or more items.



Step 2. Set the tracking plane across which the item will be mirrored.



Step 3. Set a parent for the new copy of the item.



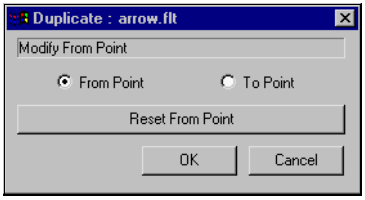


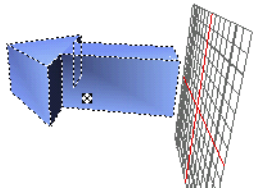
Step 4. Open the **Duplicate** toolbox in the Toolbox caddy.



Step 5. Click **Duplicate**.

Step 6. In the *Duplicate* dialog box, click **OK** without changing the **From** point or **To** point.



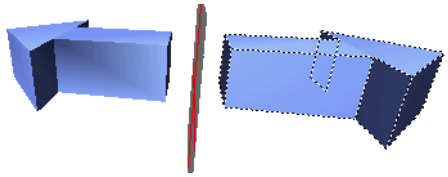




Step 7. Open the **Modify Geometry** toolbox.



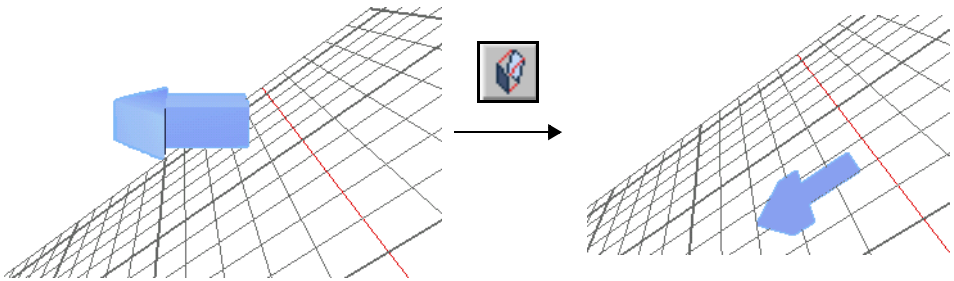
Step 8. Click **Mirror**.



Project Tool



The **Project** tool, located in the **Modify Geometry** toolbox, flattens the selected item onto the tracking plane.



To Use the Project Tool

Step 1. Set the tracking plane to the position where you want the geometry to be flattened.

Step 2. Select the geometry that you want to be flattened to the tracking plane. The selected geometry can be vertices, faces, objects, edges, or groups.



Step 3. Open the **Modify Geometry** toolbox.



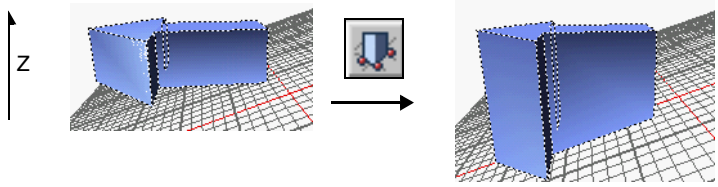
Step 4. Click **Project** to force all selected vertices onto the tracking plane.

Note: **Project** is a very useful tool for moving vertices when all the vertices of a polygon do not lie on the same plane. See “Correcting Non-coplanar Vertices” on page 357

Plant Tool



The **Plant** tool, located in the **Modify Geometry** toolbox, drops the *z* component of the lowest vertices in the selected item onto the tracking plane. For example, you can use **Plant** to make the bottom of a chimney conform to a slanted roof.



To Use the Plant Tool

Step 1. Set the tracking plane to the position where you want the planted vertices to rest.

Step 2. Select the geometry that you want to plant on the tracking plane. The selected geometry can be vertices, faces, objects, edges, or groups.



Step 3. Open the **Modify Geometry** toolbox.



Step 4. Click **Plant** to force the lowest vertices onto the tracking plane.

Linear Bend Tool



The **Linear Bend** tool, located in the **Modify Geometry** toolbox, builds models from extruded shapes like those created with **Wall**. Each time an item is bent, it is sliced and replaced by two items at the same level in the database hierarchy.

To Bend an Item

Step 1. Set the appropriate modeling mode.

Step 2. Select the geometry you want to bend.



Step 3. Open the **Modify Geometry** toolbox.

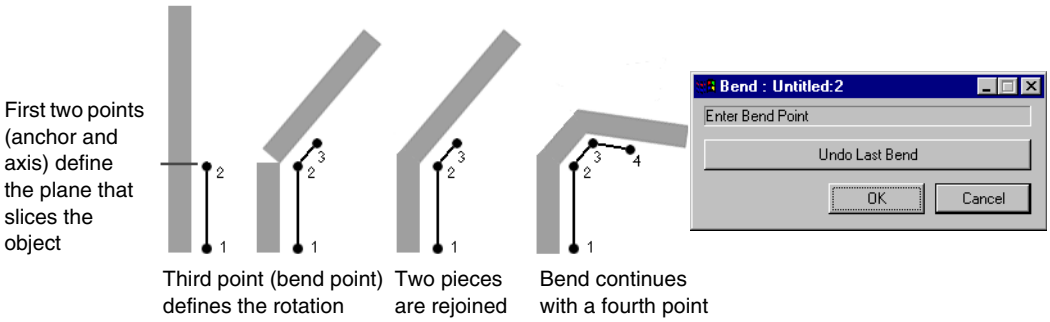


Step 4. Click **Linear Bend**.

Step 5. Enter an **Anchor point**. The **Anchor point** and the next point you will enter (the **Axis point**) define a reference edge that should be parallel to the long axis of the geometry. The geometry will be sliced perpendicular to this edge at the **Axis point**.

Step 6. Enter the **Axis point**.

Step 7. Enter the **Bend point**. This defines the angle of the bend.



The geometry will now bend at this point. To change the amount of bend, click **Undo Last Bend**. You can make multiple bends by clicking again. Each time you add a new bend point, the previous **Bend point** and **Axis points** become the new **Axis** and **Anchor points**, respectively.

Note: You may get spikes if you bend the model at a sharp angle. Also, if the original model is made up of lines that are not parallel, the joints may not be smooth.

Step 8. Click **OK** to finish the bend or click **Cancel** to quit.

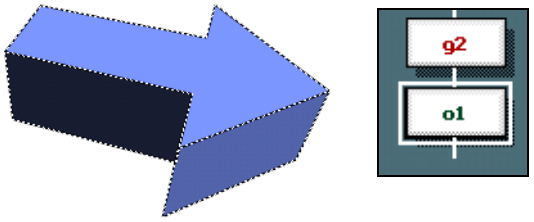
Slice Tool



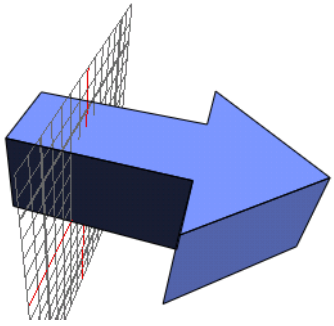
The **Slice** tool, located in the **Modify Geometry** toolbox, cuts an item along the tracking plane, deletes the original, and attaches the new items to the original parent.

To Slice an Item

Step 1. Select an item in any modeling mode. If the **Slice Displayed Nodes Only** Editing preference is set, **Slice** will only process nodes that are displayed in the Graphics view (see "Editing Preferences" on page 53).



Step 2. Set the tracking plane where you wish to cut the item.

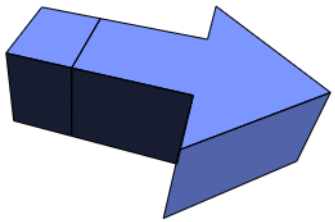


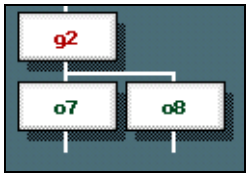


Step 3. Open the **Modify Geometry** toolbox.



Step 4. Click **Slice**. The two new items are attached to the original parent.





Cookie Cutter Tool



The **Cookie Cutter** tool, located in the **Modify Geometry** toolbox, slices all selected items along the lines of the grid, similar to the **Slice** tool (see "Slice Tool" on page 174). The individual faces are placed under the current parent node in the hierarchy. You can ignore any nodes that are not displayed in the Graphics view by choosing **Info/ Preferences/Editing** and setting the **Slice Displayed Nodes Only** checkbox.

To Use the Cookie Cutter Tool

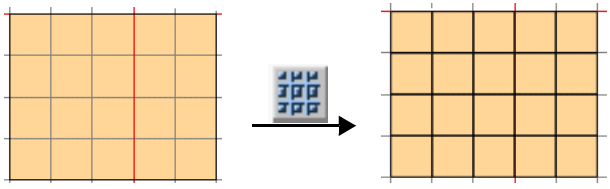
Step 1. Select the face or geometry you want to slice with the **Cookie Cutter** tool.



Step 2. Open the **Modify Geometry** toolbox.



Step 3. Click the **Cookie Cutter** tool.



Step 4. The face is automatically sliced along the grid lines by the **Cookie Cutter** tool.

Lattice Deform Tool



The **Lattice Deform** tool, located in the **Modify Geometry** toolbox, encloses the selected geometry with a grid of 3D lattice points. The geometry is embedded in this lattice space. To deform the geometry, select the grid points and then drag the points in the Graphics view.

To Deform Geometry in a Lattice

Step 1. Set the appropriate modeling mode.

Step 2. Select a geometry in the Graphics view or Hierarchy view.

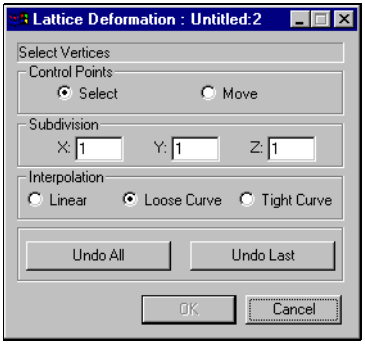


Step 3. Open the **Modify Geometry** toolbox.



Step 4. Click **Lattice Deform**.

Step 5. Click **Select** in the *Lattice Deformation* dialog box.



Step 6. Click the first point, and then hold down Shift and click to select each additional point.

Note: You can also drag a selection box around a portion of the geometry instead of individually clicking each point.

Step 7. Click **Move**.

Step 8. Drag the points in the Graphics view until you achieve the desired results.

Step 9. Click **OK** to finish or **Cancel** to quit.

Lattice Deform Parameters

These parameters let you control the way the **Lattice Deform** tool changes the geometry.

Subdivision

Sets the number of grid points you want in the *x*, *y*, and *z* planes.

Interpolation

Allows you to choose a **Linear**, **Loose Curve**, or **Tight Curve** path along which to deform the control points.

Other Deform Parameters

Undo Last removes the last deformation; **Undo All** returns the geometry to its original shape.

Bend Deform Tool



The **Bend Deform** tool, located in the **Modify Geometry** toolbox, bends a geometry along an axis. You can also select the degrees to bend the geometry using the mouse or by selecting the number of degrees in the *Bend* dialog box.

To Bend Geometry Along an Axis

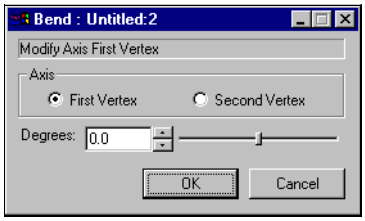
Step 1. Select a geometry in the Graphics view or Hierarchy view.



Step 2. Open the **Modify Geometry** toolbox.



Step 3. Click the **Bend Deform** tool to open the *Bend* dialog box.



Step 4. Click the first point of the axis about which the geometry will bend.

Step 5. Click another point to set the direction of the axis.

Step 6. Drag in the Graphics view, or set a different tracking plane and then drag until you achieve the desired results. The third point specifies the plane in which the geometry is being bent and the amount of the bend.

Note: You can also use the **Degrees** controls in the *Bend* dialog box to set the number of degrees more precisely. Click and drag the slider, use the up or down arrow keys, or type in the number of degrees.

Step 7. Click **OK** when you achieve the desired results, or click **Cancel** to return the geometry to its original shape.

Bend Deform Parameters

Axis

First Vertex and **Second Vertex** set the axis points that define the orientation of the bend deformation.

Degrees

Sets the amount of scaling to apply to the bend deformation. The range in values is from -360 to 360. Depending on the shape of the geometry and the axis you define, different **Degrees** values may result in identical shapes.

Twist Deform Tool



The **Twist Deform** tool, located in the **Modify Geometry** toolbox, twists the geometry along any axis you choose.

To Twist Geometry Along an Axis

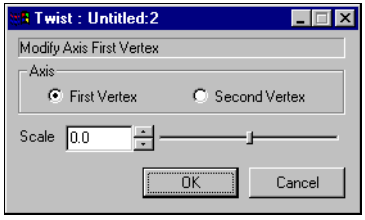
Step 1. Select the geometry you want to deform.



Step 2. Open the **Modify Geometry** toolbox.



Step 3. Click **Twist Deform**.



Step 4. Click the first point of the axis. Use the tracking plane to help place the point in the correct axis.

Step 5. Click the second point of the axis. The first and second points define a line, which is where the twist will occur.

Step 6. Define the amount of twist to apply, using either the **Scale** slider or the **Scale** arrows. You can also type in the **Scale** values or use the mouse to scale the geometry in the Graphics view. The range of **Scale** values is from -360 to 360.

Step 7. Click **OK** when you are satisfied with the result, or click **Cancel** to quit.

Twist Deform Parameters:

Axis

First Vertex and **Second Vertex** set the axis points that define the orientation of the twist deformation.

Scale

Sets the amount of scaling to apply to the twist deformation. The range in values is from -360 to 360. Depending on the shape of the geometry and the axis you define, different **Scale** values may result in identical shapes.

Taper Deform Tool



The **Taper Deform** tool, located in the **Modify Geometry** toolbox, scales the geometry along any axis you choose. The geometry is scaled perpendicular to the axis based on a specified scale value. The maximum amount of scaling occurs at the second vertex of the axis and decreases (or increases) down the axis toward the first vertex, where no scaling occurs.

To Taper Geometry Along an Axis

Step 1. Set the appropriate modeling mode.

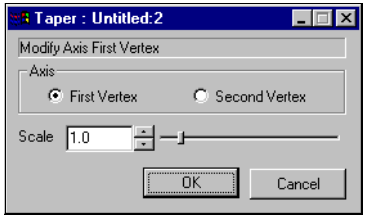
Step 2. Select the geometry.



Step 3. Open the **Modify Geometry** toolbox.



Step 4. Click the **Taper Deform** tool to open the *Taper* dialog box.



Step 5. Click the first point on the geometry. Use the tracking plane to help place the point in the correct axis.

Step 6. Click the second point on the geometry. The first and second points define a line, which is where the taper will occur.

Step 7. Define the amount of taper to apply, using either the **Scale** slider or the **Scale** arrows. You can also type in the **Scale** values, or use the mouse to set the scale in the Graphics view.

Step 8. Click **OK** when you are satisfied with the result, or click **Cancel** to quit.

Taper Deform Parameters

Axis

First Vertex and **Second Vertex** set the axis points that define the orientation of the taper deformation.

Scale

Sets the maximum amount of scaling to apply to the geometry. The range in values is from 0 (zero) to 10. A scale value of 1 indicates no scaling. Depending on the shape of the geometry and the axis you define, different **Scale** values may result in identical shapes.

Bulldoze Tool



The **Bulldoze** tool, located in the **Modify Geometry** toolbox, flattens selected faces or areas of terrain. For example, **Bulldoze** can level terrain polygons to create a "footprint" area for the addition of airports, buildings, roads, and so forth.

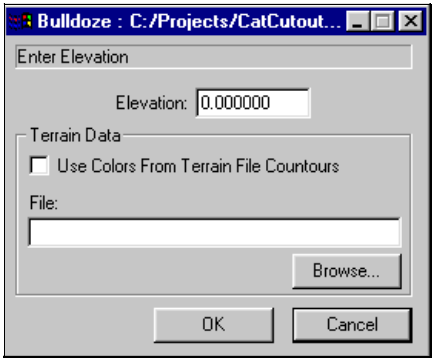
To Bulldoze Faces

Step 1. Turn on **Face** modeling mode.

Step 2. In the Graphics view, select the faces to be flattened.

Step 3. Open the **Modify Geometry** toolbox and select **Bulldoze**. The vertices in the selected area are highlighted and the *Bulldoze* dialog box opens.



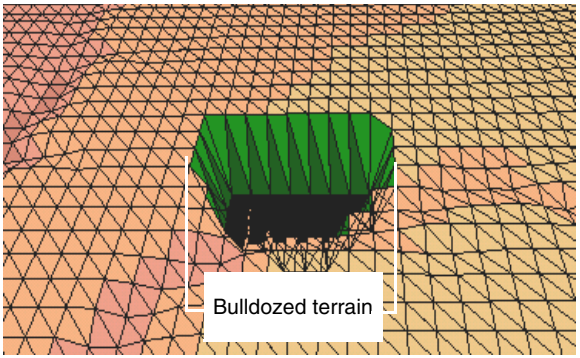


Step 4. Enter the elevation that you want to assign to the terrain faces when they are flattened.

Step 5. If you want to copy the elevation contour color from an existing terrain file and apply it to the flattened area, set the **Use Colors From Terrain File Contours** checkbox, and enter the path and file name of the DED file to be used.

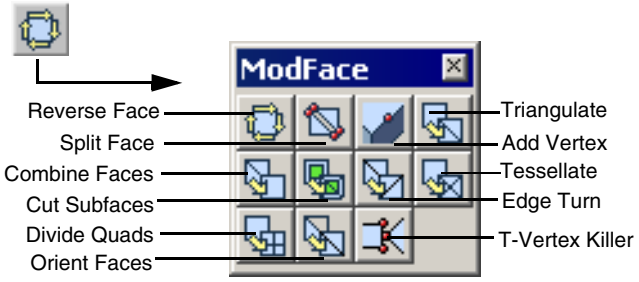
Note: Although you select a DED file, the **Bulldoze** tool looks for the .prefs file associated with the DED file. This file is created when you generate terrain from the DED file. (For example, if you generate terrain from Hawaii.ded, Creator stores the terrain window settings, including contour colors, in the file Hawaii.prefs.) If you move or delete the .prefs file, or if you did not generate terrain with the DED file, you will not be able to get the elevation contour color from the file.

Step 6. Click **OK**. The selected vertices are moved and 'flattened' to the new elevation.



Modify Face Tools

Tools in the **Modify Face** toolbox transform selected polygons but do not change the shape of the overall geometry.



Reverse Face Tool



The **Reverse Face** tool reverses the order of the vertices belonging to the selected face, making it visible from the opposite side.

To Reverse a Face

Step 1. Select the face you want to reverse.



Step 2. Open the **Modify Face** toolbox.



Step 3. Click the **Reverse Face** tool. The face is automatically reversed.

Split Face Tool



The **Split Face** tool divides one or more faces into smaller faces by adding an edge between each pair of vertices you select.

Polygons can also be divided using **Edit/Triangulate** or **Slice** in the **Geometry** toolbox.

To Split a Face

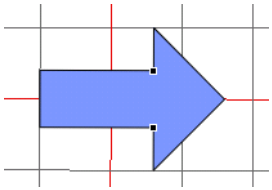


Step 1. Open the **Modify Face** toolbox.

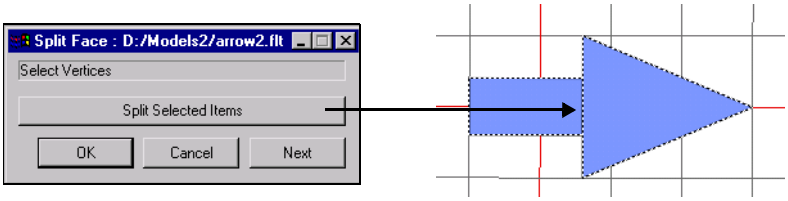
Step 2. Click **Split Face**.

Step 3. Select two or more vertices in the same face that are not already connected by an edge. (Press Shift while selecting to add vertices to the Select List.)

If you select more than two vertices, edges are added in the order that the vertices were selected.



Step 4. Click **Split Face** in the *Split Face* dialog box.



Step 5. Click **OK** to accept the new faces, **Next** to accept the faces and split another face, or **Cancel** to discard the change.

The two new faces are attached to the original parent.

Split Selected Item

The *Split Face* dialog box contains a single parameter, **Split Selected Item**, which adds an edge between each pair of selected vertices. Edges are added in the order the vertices were selected.

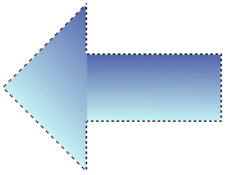
Add Vertex Tool



The **Add Vertex** tool adds vertices to each selected edge.

To Add Vertices to an Edge

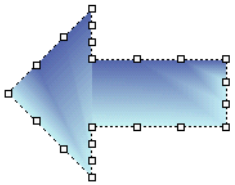
Step 1. Select one or more edges at any level above vertex. In this example, the geometry is selected in Face mode.

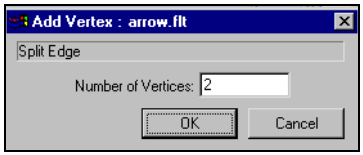




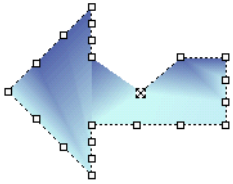
Step 2. Open the **Modify Face** toolbox.

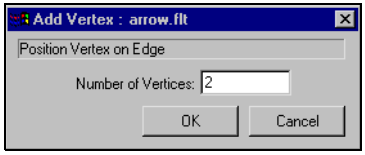
Step 3. Click **Add Vertex** and specify the number of vertices you wish to add to each edge in the *Add Vertex* dialog box.





Step 4. While the *Add Vertex* dialog box is open, you can reposition any new vertex by Ctrl-clicking it in the Graphics view and dragging it to a new position.





Step 5. When you are satisfied with the new vertices, click **OK**.

Add Vertex Parameter

The *Add Vertex* dialog box contains the following parameter:

Number of Vertices

The number of vertices to be added, at equal intervals, along each selected edge.

While the *Add Vertex* dialog box is open, you can reposition any new vertex by Ctrl-clicking it in the Graphics view and dragging it to a new position.

Triangulate Tool



The **Triangulate** tool divides each selected face into triangles and replaces it in the hierarchy with the new triangular faces. Triangulate is especially useful for correcting concave faces.

To Divide Faces Using Triangulate

Step 1. Select the faces you want to triangulate.

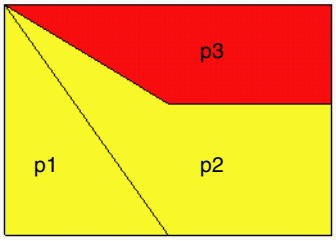


Step 2. Open the **Modify Face** toolbox.

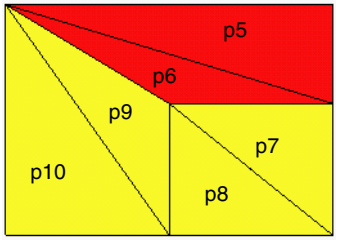


Step 3. Click **Triangulate**. Alternatively you can choose **Edit/Triangulate**.

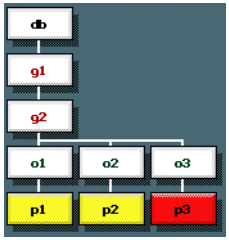
The following figure shows how three faces are divided using **Triangulate**.



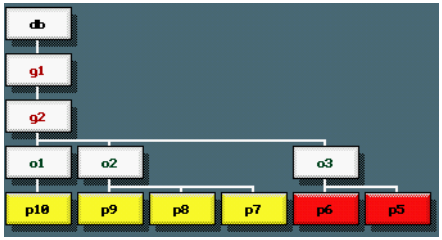
Three faces before triangulation, including the concave face p2



After triangulation, there are six faces



Hierarchy before triangulation



Hierarchy after triangulation, showing that the parent/child relationship of Objects and Faces is preserved

Combine Faces Tool



The **Combine Faces** tool merges selected faces that share edges into the smallest possible number of faces. The selected faces must be coplanar, and have identical color, texture and material attributes. Each new face is attached only once in the hierarchy, regardless of the number of original parents.

To Combine Faces

Step 1. Select the faces you want to combine.

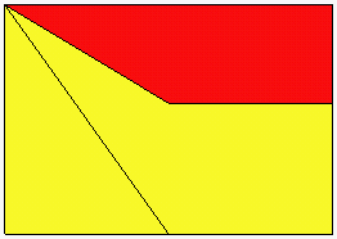


Step 2. Open the **Modify Face** toolbox.

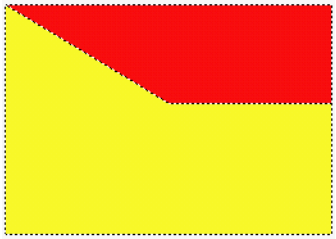


Step 3. Click **Combine Faces**. Alternatively you can choose **Edit/Combine Faces**.

The following figure shows the effects of **Combine Faces** in the Graphics and Hierarchy views.

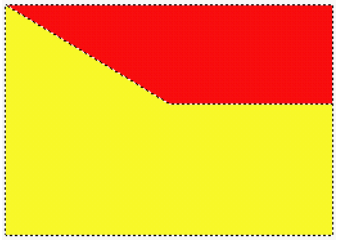


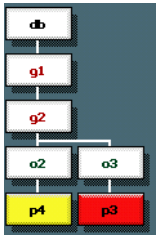
A rectangular area with three faces



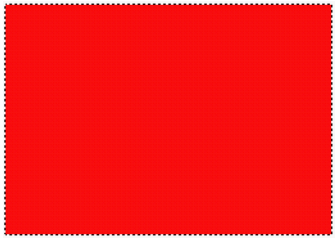
After Combine Faces, the two yellow polygons become a single face

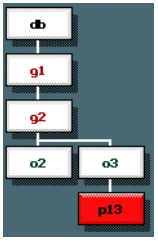
The Editing preference **Combine Similar Faces Only** is enabled by default, ensuring that only faces with the same color and texture are combined. When this preference is disabled, faces are combined without regard to color and texture.





After Combine Faces, with Combine Similar Faces Only enabled





After Combine Faces, with Combine Similar Faces Only disabled

Cut Subfaces Tool



The **Cut Subfaces** tool merges subfaces into each selected parent face by removing the area under the subface and triangulating the remaining area of the face. Z-buffered realtime systems require extra drawing overhead to display subfaces, therefore many systems severely limit the number of subfaces you can include in a database. The **Cut Subfaces** tool eliminates subfaces, improving performance.

To Cut Subfaces

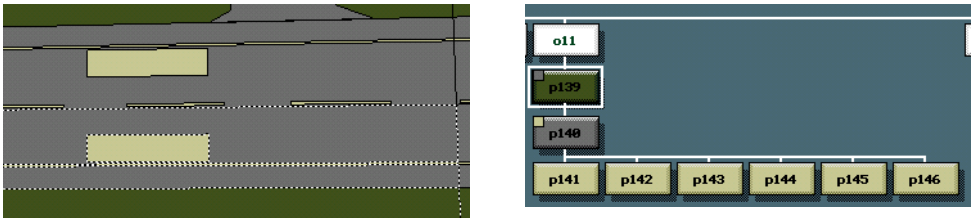
Step 1. Select the parent face of the subface you want to cut.



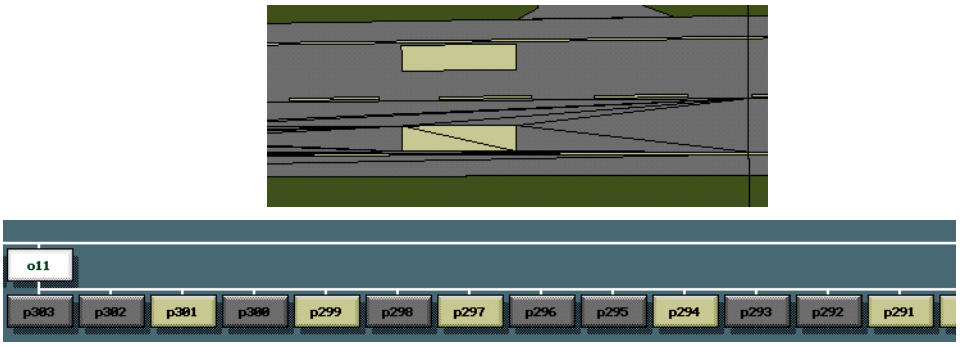
Step 2. Open the **Modify Face** toolbox.



Step 3. Click **Cut Subfaces** (or choose **Edit/Cut Subfaces**).



The selected section of the runway is a subface of the grassy area, and the markings are subfaces of the runway

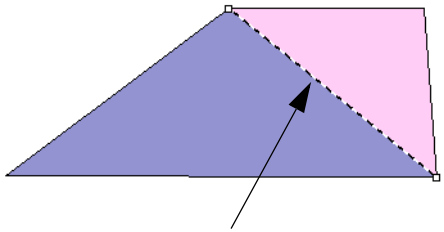


After Cut Subfaces, the runway and markings are cut into the grassy area as a series of triangular faces

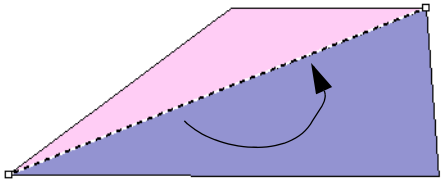


Edge Turn Tool

The **Edge Turn** tool flips the triangulation of two triangular polygons that share a common edge.The common edge is turned counter-clockwise. This allows you to quickly maximize triangle strips for rendering purposes. You can also change the basic shape of geometry.



Two triangular polygons that share a common edge



When the **Edge Turn** tool is used, the triangulation changes when the edge is turned counter-clockwise, and the vertices remain in the same positions.

The attributes for the resulting triangles and vertices are defined by the following set of rules:

- The attributes of the original triangular polygons do not change; only the vertices are repositioned.
- When one vertex becomes two, the two new vertices retain all the attributes of the original vertex, including position, color, material, and texture *u* and *v* coordinates.
- Where two vertices become one, the position of the new vertex is the same as that of the original vertices. The color of the new vertex is assigned the average of the two colors of the original vertices; the normal of the new vertex is assigned the average of the two normals of the original vertices; and the texture *u* and *v*

coordinates as well as all the other vertex attributes of the new vertex are arbitrarily chosen from one of the original two vertices.

To Turn Common Edges



Step 1. Open the **Modify Face** toolbox.



Step 2. Click the **Edge Turn** tool. The *Edge Turn* dialog box displays and prompts you to select two closed triangles with a shared edge in the Graphics view. When you select the triangles, the common edge automatically turns counter-clockwise and changes the triangulation pattern.



Step 3. Click **OK** to save all of the edge turns and close the dialog box; **Undo All** to undo all the edge turns and leave the dialog box open; **Undo Last** to undo the last edge turn; or **Cancel** to undo all of the edge turns and close the dialog box.

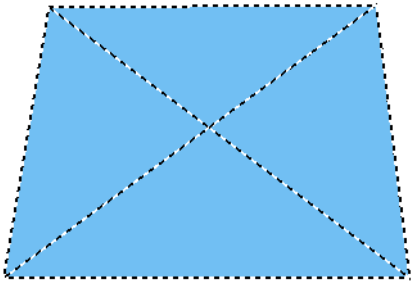
Tessellate Tool



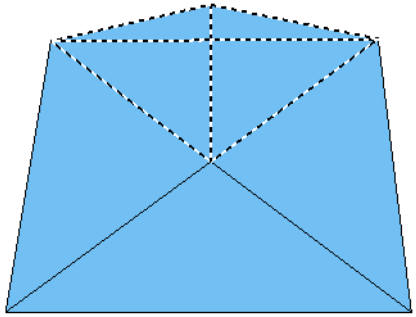
The **Tessellate** tool triangulates a selected polygon. A new triangle is formed for each edge. All triangles meet at the center point of the polygon.

The new polygons created retain all of the attributes of the original polygon, including color, material, and texture indices. The new vertices created retain all of the attributes of the original vertices, such as color, normal, and texture coordinates.

You can tessellate a polygon to create more faces and vertices to add more detail, for example.



The square was tessellated into four triangles



In this example, one of the triangles was raised using the **Peak** tool

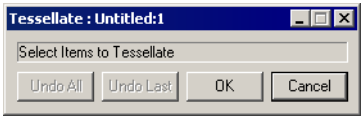
To Tessellate Faces



Step 1. Open the **Modify Face** toolbox.



Step 2. Click the **Tessellate** tool. The *Tessellate* dialog box displays and prompts you to select a face in the Graphics view that you want to tessellate.



Step 3. Select a face in the Graphics view with the right mouse button. The face will be triangulated. You can continue selecting any of the faces to tessellate.

Step 4. Click **OK** to save your changes, **Undo All** to undo all of your changes, **Undo Last** to undo the last change, or **Cancel** to close the dialog box without saving changes.

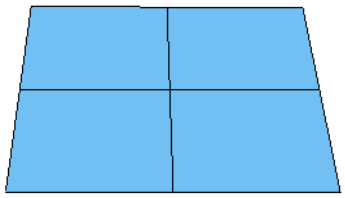
Divide Quads Tool



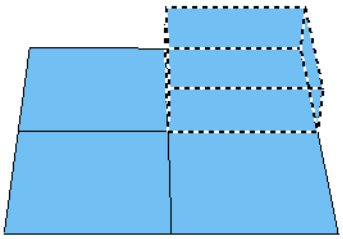
The **Divide Quads** tool divides a four-sided polygon (quadrilateral) into four new quadrilateral polygons. All quadrilaterals meet at the center point of the polygon.

The new polygons created retain all of the attributes of the original polygon, including color, material, and texture indices. The new vertices created retain all of the attributes of the original vertices, such as color, normal, and texture coordinates.

You can divide a polygon to create more faces and vertices to add more detail, for example.



The square was divided into four quadrilaterals



In this example, one of the quadrilaterals was raised using the **Wall** tool

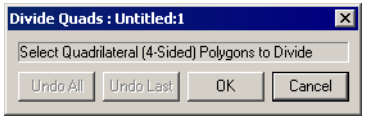
To Divide Faces



Step 1. Open the **Modify Face** toolbox.



Step 2. Click the **Divide Quads** tool. The *Divide Quads* dialog box displays and prompts you to select a face in the Graphics view that you want to divide.



Step 3. Select a face in the Graphics view with the right mouse button. The face will be divided into quads. You can continue selecting any of the faces to divide.

Step 4. Click **OK** to save your changes, **Undo All** to undo all of your changes, **Undo Last** to undo the last change, or **Cancel** to close the dialog box without saving changes.

Orient Faces Tool



The **Orient Faces** tool reverses faces that are facing the opposite direction from a selected face. The tool searches and analyzes each connected face to see if its vertex normals point in the opposite direction. The faces are selected recursively in the database to find all faces that connect to the selected face by common edges. When faces are found that face backwards, the order of those vertices is reversed.

Imported geometry in other formats, such as DXF, or 3DS, often have faces that are not oriented uniformly. Many faces are backfaced compared to their neighboring faces. The **Orient Faces** tool reorients these backfaced faces quickly and efficiently.

To Orient Faces

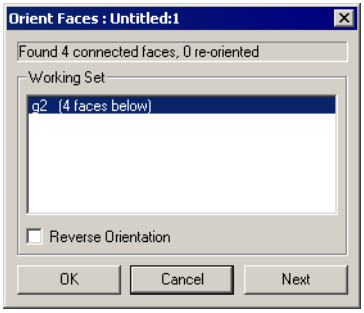
Step 1. Select a face with the direction that you want to use as a reference.



Step 2. Open the **Modify Face** toolbox.



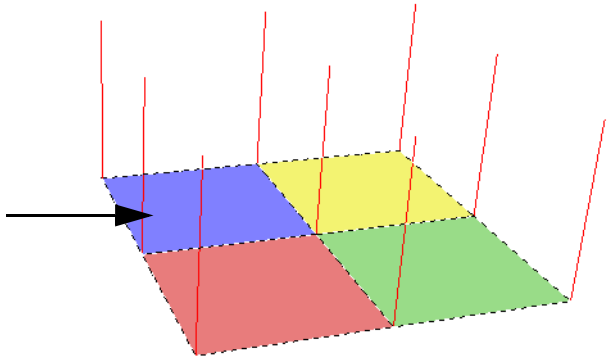
Step 3. Click **Orient Faces** to display the *Orient Faces* dialog box. All connected faces will be selected in the Graphics view. The nodes in the hierarchy associated with the connected faces are listed in hierarchical order in the *Orient Faces* dialog box. A message at the top of the dialog box displays the total number of connected faces that were selected.



Step 4. Set **Reverse Direction** to reverse the direction of the selected faces.

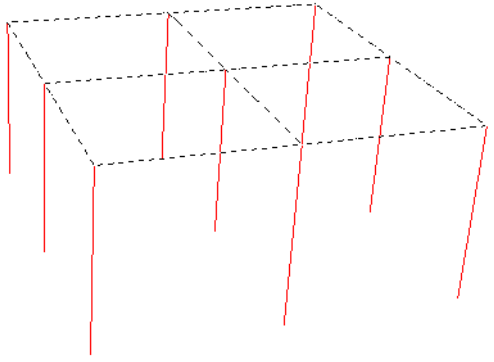
All connected faces to the reference face are selected.

The vertex normals point in an upward direction in this example.



When **Reverse Direction** is set, all selected faces are reversed.

The vertex normals point in an opposite (downward) direction



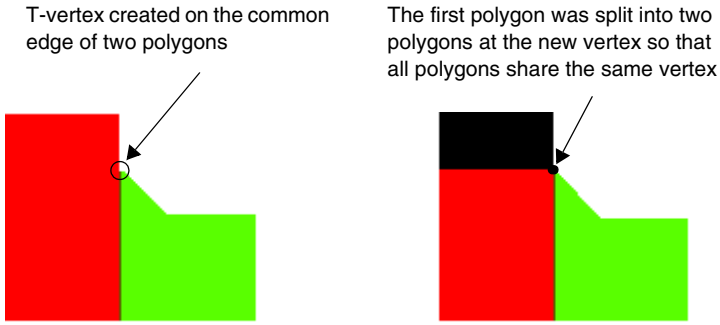
Step 5. Click **OK** to save changes and close the dialog box, **Cancel** to cancel the operation and close the dialog box, or **Next** to save the changes and keep the dialog box open so that you can select a different reference face. The new reference face must be different from one of the reversed faces.

T-Vertex Killer Tool



The **T-Vertex Killer** tool finds and corrects T-vertices within a database. A T-vertex occurs where two or more adjacent polygons share an edge, and the polygons do not share a common vertex on that edge. Rendering artifacts such as cracks can appear along the common edge.

Using the **T-Vertex Killer** tool, you can specify tolerance values for determining how close a vertex needs to be to an edge to be considered a T-vertex. For a greater level of control, you can choose the individual T-vertices that you want corrected. The tool corrects T-vertices by adding a vertex on the edge at the T-vertex location. You can use the **Slice** tool or **Triangulate** tool if you want to split the polygon into two polygons at the new point.



If you have previously selected polygons when you start the tool, the tool will operate only on the vertices and edges of the selected items. If you have not selected any polygons when you start the tool, the entire database will be searched for T-vertices.

This tool works best on non-transformed geometry. It will not correctly detect T-vertices under instances, external references, or any other geometry that has been transformed.

Note: Multiple vertices can correspond to the same T-vertex.

To Correct T-Vertices

Step 1. Select the polygons in the Graphics view that you want corrected. If you do not select any polygons, the **T-Vertex Killer** tool will search the entire database for T-vertices.

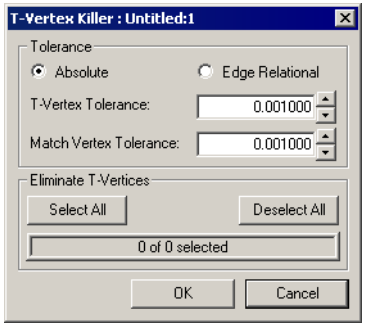


Step 2. Open the **Modify Face** toolbox.



Step 3. Click **T-Vertex Killer** to display the *T-Vertex Killer* dialog box. You will notice T-vertices highlighted in the Graphics view.

You can select or deselect individual T-vertices by holding the Ctrl key and clicking the left mouse button. Deselected vertices become purple. A message at the bottom of the window indicates the number of selected T-vertices in relation to the total number found.



Step 4. Select either **Absolute** or **Edge Relational** as the tolerance distance state. **Absolute** tolerance distances are interpreted as the same physical distance everywhere in the database. **Edge Relational** tolerance distances are affected by the relationship between the user-specified tolerance values and the length of the edges.

Step 5. Enter the **T-Vertex Tolerance** value. T-vertex tolerance is the maximum distance between a vertex and an edge for determining if a T-vertex exists. Any vertex that is within this distance of an edge will be considered a T-vertex. This value is affected by the tolerance distance state. The default value is .001000, but you can change it to a different positive value.

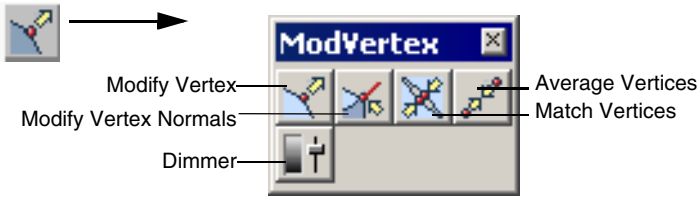
Step 6. Enter the **Match Vertex Tolerance** value. **Match Vertex Tolerance** is the minimum distance between two T-vertices on the same edge. If two or more T-vertices on the same edge exist and are within this tolerance distance of each other, their positions will be averaged, and only one new vertex on the edge will be created. The default value is .001000, but you can change it to a different positive value.

Step 7. You can click **Select All** to select all T-vertices, or **Deselect All** to deselect all T-vertices. This is useful if you have selected or deselected individual vertices, and you want all T-vertices considered instead.

Step 8. Click **OK** to correct the T-vertices, or **Cancel** to cancel operations and close the window.

Modify Vertex Tools

Modify Vertex tools move selected vertices and alter direction of the normals associated with them when Calculate Shading is performed.



Modify Vertex Tool



Modify Vertex (Shift+M) moves selected vertices to a new coordinate. If **Freeze Flags** have not been set, all vertices move to the coordinate you enter. However, it is useful to set one or two **Freeze Flags** and modify only the unfrozen component of the vertices. For example, if you freeze *x* and *y*, **Modify Vertex** changes only the *z* component of the selected vertices.

To Modify a Vertex

Step 1. Set **Freeze Flags**, if necessary.



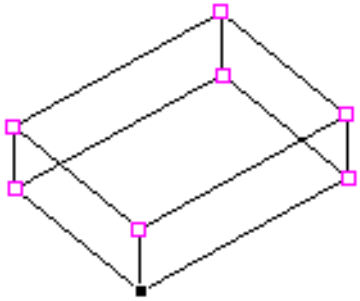
Step 2. Click **View Panel** in the Toolbar.



Step 3. Click **Draw Vertex Large** to make vertices easier to select.



Step 4. In **Vertex** modeling mode, select the vertex to be modified.

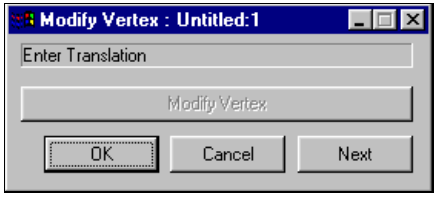




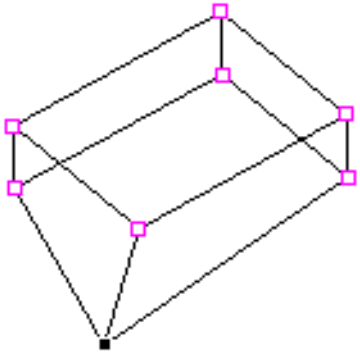
Step 5. Open the **Modify Vertex** toolbox.

Step 6. Click the **Modify Vertex** tool. The *Modify Vertex* dialog box opens and prompts you to select a new position for the vertex.





Step 7. Select a new position for the vertex and click **OK** to accept the new vertex position, **Next** to modify another vertex, or **Cancel** to quit the operation.



Match Vertices Tool

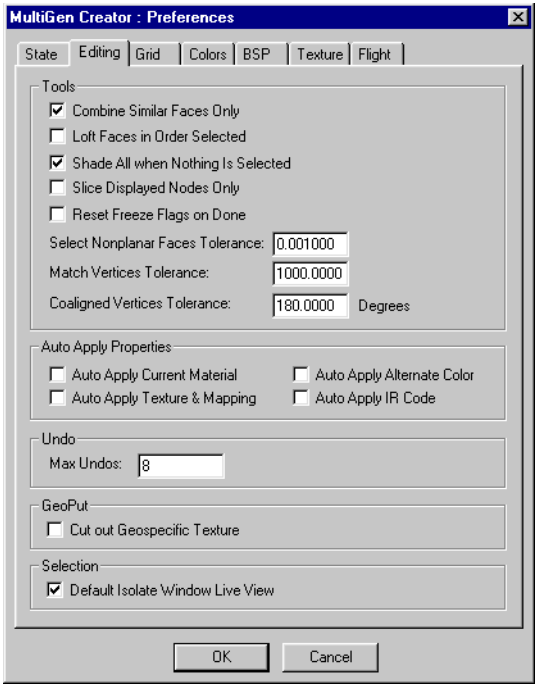


Match Vertices searches selected items for sets of "very close" vertices, that is, vertices that are no farther apart from one another than the distance specified by the **Match Vertices Tolerance** Editing preference. The vertices in each such set are moved to the average coordinate of the set. **Match Vertices** is particularly good for fixing small "holes" in models.

Note: To determine whether two or more vertices are close, but not matched, set the mode to Vertex, select the vertices, and select **View/Fit on Select** (V). If the vertices are not matched, the view will zoom into the area.

To Set the Match Vertices Tolerance Preference

Step 1. Select **Info/Preferences**, and then click the **Editing** tab to display the Editing preferences.



Step 2. Enter a tolerance, in database units and click **OK**.

To Match Vertices



Step 1. Open the **Mode** toolbox.



Step 2. Click the **Vertex Mode** tool to change to **Vertex** modeling mode.

Step 3. Select the vertices you want to match.



Step 4. Open the **Modify Vertex** toolbox.



Step 5. Click the **Match Vertices** tool.

Note: An alternate method is to choose **Edit/Match Vertices**.

Average Vertices Tool



Average Vertices calculates the average point of all selected vertices and joins them together at that point. This tool is useful for adjoining separate faces at their midpoint or for creating a corner from two vertices on the same face. The *uv* coordinates on a textured face are remapped when the vertices are averaged.

To average vertices:



Step 1. Open the **Mode** toolbox.



Step 2. Click the **Vertex Mode** tool to change to **Vertex** modeling mode.

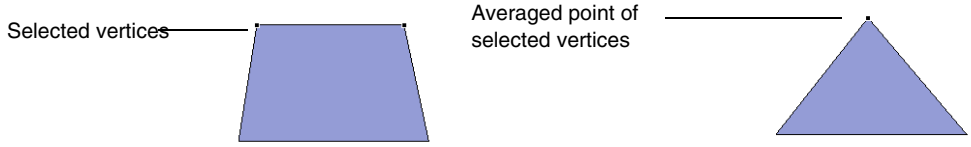
Step 3. Select the vertices you want to average.



Step 4. Open the **Modify Vertex** toolbox.



Step 5. Click the **Average Vertices** tool.



Creating the Average Point of Vertices on a Face

Dimmer Tool



The **Dimmer** tool allows you to lighten and darken vertex colors by changing the color intensity and transparency on the vertices. You can gradually adjust the colors using sliders, or you can enter Intensity Delta or Transparency Delta values.

To adjust vertex color intensities:

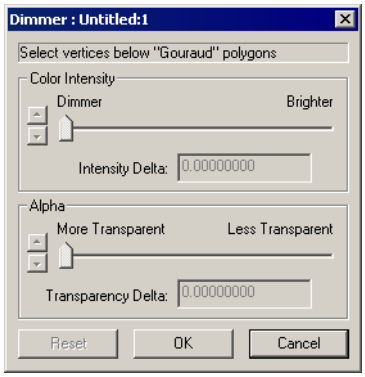
Step 1. Select vertices with vertex colors that you want to adjust.

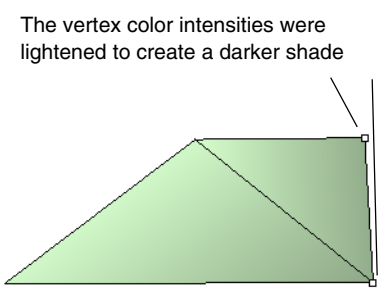


Step 2. Open the **Modify Vertex** toolbox.



Step 3. Click the **Dimmer** tool to display the *Dimmer* dialog box.





Step 4. Drag the **Color Intensity** slider to adjust the vertex color intensity. Alternatively, you can enter an **Intensity Delta** value to apply to the color intensities.

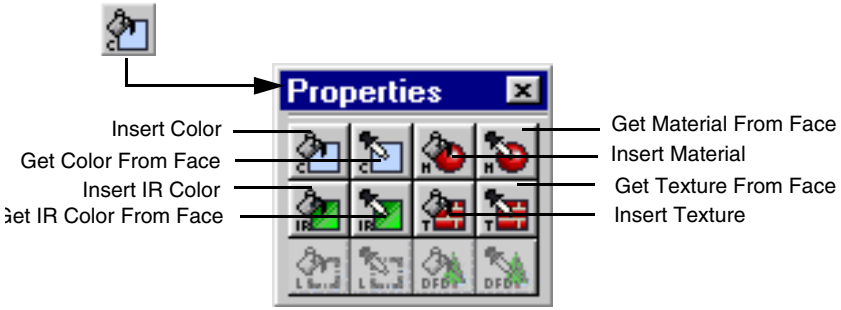
Step 5. Drag the **Alpha** slider to adjust the vertex transparency intensity. Alternatively, you can enter a **Transparency Delta** value to apply to the transparency intensities.

Step 6. You can select additional vertices for which to adjust color intensities.

Step 7. Click **OK** to save the changes and close the dialog box; **Reset** to reset the default values; or **Cancel** to close the dialog box without saving changes.

Properties Tools

Tools in the **Properties** toolbox apply color, texture and material characteristics to selected faces in the database.



Insert Line Style Tool



The **Insert Line Style** tool in the Properties toolbox assigns the line style selected in the Line Style palette to selected faces. The **Insert Line Style** tool is part of the Instrumentation Option.

To Use Insert Line Style

Step 1. Open the Line Style palette (**Palettes/Line Style**) and select the style to be applied.



Step 2. In Face mode, select the target face or faces.



Step 3. Click **Insert Line Style** in the **Properties** toolbox.

Get Line Style From Face Tool



The **Get Line Style From Face** tool in the **Properties** toolbox selects the line style in the Line Style palette that matches the line style assigned to the selected face.The **Get Line Style From Face** tool is part of the Instrumentation Option.

To Transfer a Line Style to Another Face



Step 1. In Face mode, select the face.

Step 2. Click **Line Style From Face** in the **Properties** toolbox.



Step 3. Select the target face and click **Insert Line Style**.

Paste GeoFeature Attributes Tool



Transfers the GeoFeature attributes that you copied using **Copy GeoFeature Attributes** to all selected cultural features that are of the same category (point, areal, or linear).

This tool is included with the Terrain Pro option.

To Copy and Paste GeoFeature Attributes

You can assign identical attributes to multiple features using the **Copy GeoFeature Attributes** and **Paste GeoFeature Attributes** tools in the **Properties** toolbox. All features involved in the operation must be in the same category of point, areal, or linear. You cannot copy GeoFeature attributes from a linear feature, for example, and paste them to an areal feature. To use these tools:

Step 1. Make sure you are in Object mode.

Step 2. In either the Graphics or Hierarchy view, select a single feature whose attributes you want to copy.



Step 3. Open the **Properties** toolbox.



Step 4. Click **Copy GeoFeature Attributes**.

Step 5. Select the features to which you want to copy the attributes.

You can fence a wide area, if you want, because all selections that are not in the same category as the copied feature will be ignored.



Step 6. Click **Paste GeoFeature Attributes**. The attributes of all the selected features are changed to match the copied feature.

If the feature you are copying and pasting is not centered at the origin, it will be offset from the position where you pasted it. See “Centering a Model at the Origin” on page 352 for details.

Copy GeoFeature Attributes Tool



Copies the geofeature attributes of the selected cultural feature (without affecting the Graphics Buffer or Text Buffer), in preparation for transferring the attributes to other features using **Paste GeoFeature Attributes**.

This tool is included with the Terrain Pro option.

To Copy and Paste GeoFeature Attributes

You can assign identical attributes to multiple features using the **Copy GeoFeature Attributes** and **Paste GeoFeature Attributes** tools in the **Properties** toolbox. All features involved in the operation must be in the same category of point, areal, or linear. You cannot copy GeoFeature attributes from a linear feature, for example, and paste them to an areal feature. To use these tools:

Step 1. Make sure you are in Object mode.

Step 2. In either the Graphics or Hierarchy view, select a single feature whose attributes you want to copy.



Step 3. Open the **Properties** toolbox.



Step 4. Click **Copy GeoFeature Attributes**.

Step 5. Select the features to which you want to copy the attributes.

You can fence a wide area, if you want, because all selections that are not in the same category as the copied feature will be ignored.

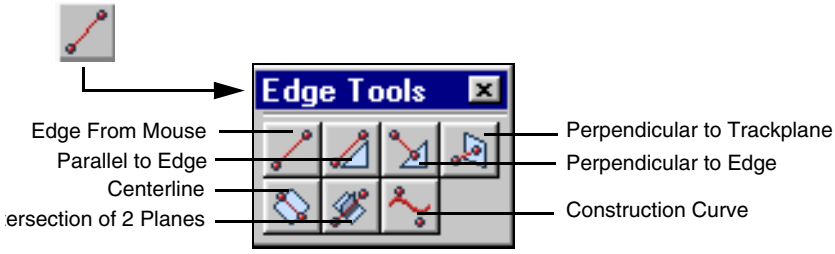


Step 6. Click **Paste GeoFeature Attributes**. The attributes of all the selected features are changed to match the copied feature.

If the feature you are copying and pasting is not centered at the origin, it will be offset from the position where you pasted it. See “Centering a Model at the Origin” on page 352 for details.

Edge Tools

Edge tools create construction edges in the database that help you place new elements during modeling.

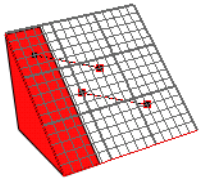


You can hide and unhide construction edges using the **Draw Constructions** toggle in the View Panel.

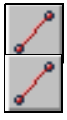
Edge From Mouse Tool



Edge From Mouse creates construction edges from pairs of vertices that you enter with the mouse.



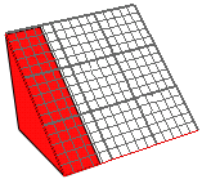
To Create Construction Edges Using Edge from Mouse

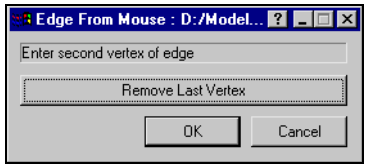


Step 1. Open the **Edge** toolbox.

Step 2. Click **Edge From Mouse**.

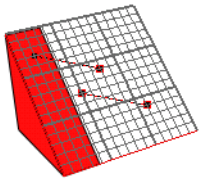
The *Edge From Mouse* dialog box prompts you to enter vertices.



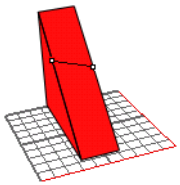


Step 3. Enter vertices with the mouse. An edge is created between each pair of vertices.

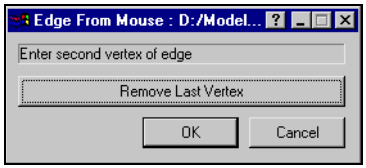
Left-click to enter vertices on the tracking plane. Middle-click to reference existing vertices or edges in the database.



Vertices entered by left-clicking the tracking plane



Vertices entered by middle-clicking edges



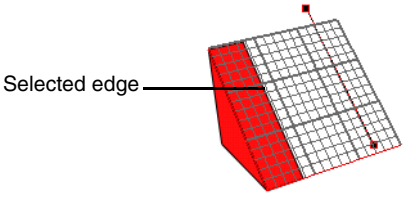
Step 4. If you make a mistake, click **Remove Last Vertex** to undo vertices, one at a time.

Step 5. Click **OK** to accept the construction edges and close the dialog box, or **Cancel** to discard all new edges.

Parallel to Edge Tool



Parallel to Edge creates a construction edge parallel to an existing edge in the database, in the plane of the face that is most nearly parallel to the tracking plane.



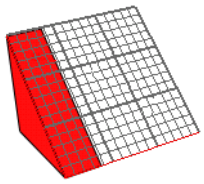
To Create a Construction Edge Parallel to an Edge

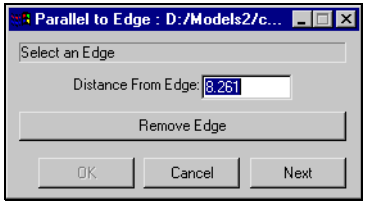


Step 1. Open the **Edge** toolbox.

Step 2. Click **Parallel to Edge**.

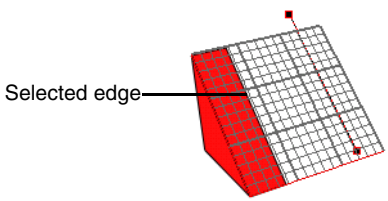
The *Parallel to Edge* dialog box prompts you to select an edge.

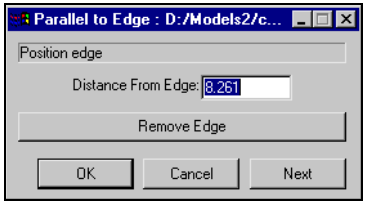




Step 3. Select an edge in the database.

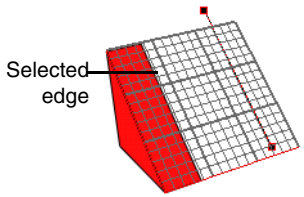
A construction edge appears parallel to the selected edge and the *Parallel to Edge* dialog box prompts you to position the edge.

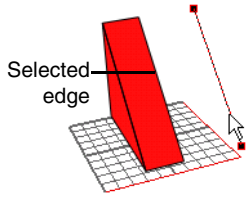




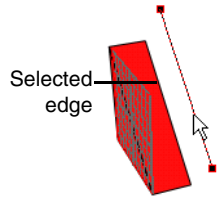
Step 4. Position the construction edge by editing **Distance From Edge** in the *Parallel to Edge* dialog box or by dragging the pointer in the Graphics view.

The construction edge stays in the plane of the face that is most nearly parallel to the tracking plane.





The two construction edges on the left remain parallel to the top face, because the tracking plane is horizontal.



The right edge remains parallel to the side face, because the tracking plane is vertical.

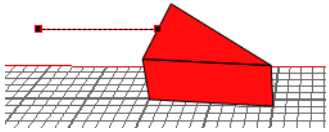
Step 5. If you make a mistake, click **Remove Edge** to undo the edge without closing the dialog box.

Step 6. Click **OK** to accept the construction edge and close the dialog box, **Next** to create another construction edge, or **Cancel** to discard all new edges.

Perpendicular to Edge Tool



Perpendicular to Edge creates one or more construction edges perpendicular to a selected edge and parallel to the tracking plane.



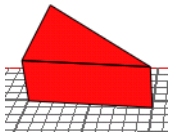
To Build Construction Edges Perpendicular to an Edge

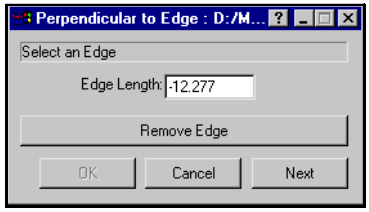


Step 1. Open the **Edge** toolbox.

Step 2. Click **Perpendicular to Edge**.

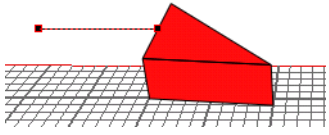
The *Perpendicular to Edge* dialog box prompts you to select an edge.

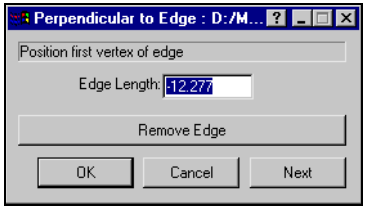




Step 3. Right-click an existing edge in the Graphics view.

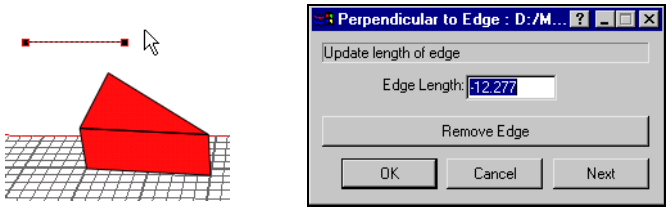
A construction edge appears perpendicular to the selected edge and parallel to the tracking plane, and the *Perpendicular to Edge* dialog box prompts you to position the first ve rtex of the edge.



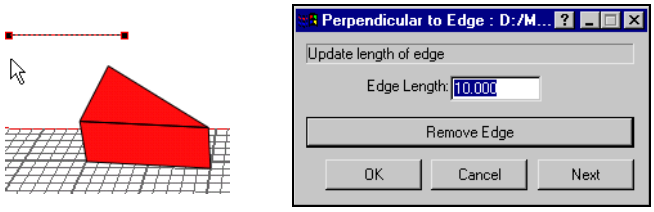


Step 4. Position the construction edge along the line defined by the selected edge by dragging the pointer anywhere in the Graphics view.

Perpendicular to Edge prompts you to update the length of the edge.



Step 5. Update the length of the construction edge by editing the **Edge Length** value or by dragging the pointer in the Graphics view.



Step 6. If you want to discard this construction edge without closing the dialog box, click **Remove Edge**.

Step 7. Click **OK** to accept the construction edge, **Next** to accept the edge and create another construction edge, or **Cancel** to discard all new edges.

Perpendicular to Trackplane Tool



Perpendicular to Trackplane creates construction edges perpendicular to the tracking plane using a starting point and length that you specify. The *Perpendicular to Trackplane* dialog box prompts you to enter the first vertex of the edge and specify its length.

To make the edge perpendicular to a face, set the tracking plane to the face before entering these vertices.

To Build Construction Edges Perpendicular to the Tracking Plane

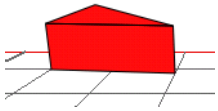


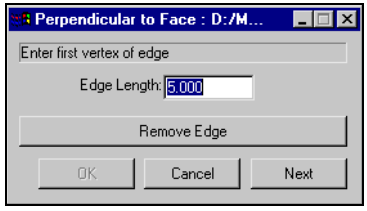
Step 1. Open the **Edge** toolbox.



Step 2. Click **Perpendicular to Trackplane**.

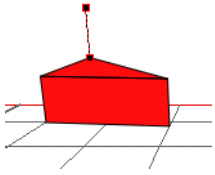
The *Perpendicular to Trackplane* dialog box prompts you for the first vertex of the construction edge.

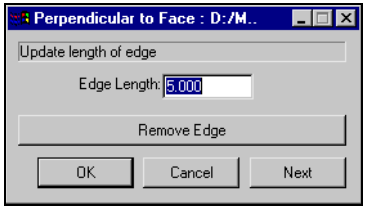




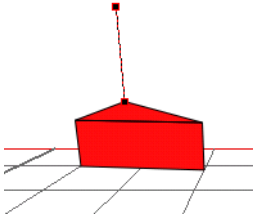
Step 3. Enter the first vertex of the edge.

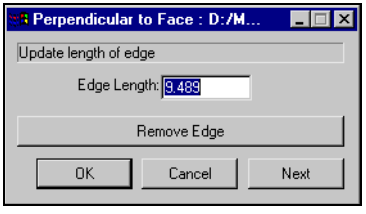
A construction edge appears at the point you specify, and *Perpendicular to Trackplane* prompts you to update the length of the edge.





Step 4. Update the length of the construction edge by editing the **Edge Length** value or by dragging the pointer in the Graphics view.





Step 5. If you want to discard this construction edge without closing the dialog box, click **Remove Edge**.

Step 6. Click **OK** to accept the construction edge, **Next** to accept the edge and create another construction edge, or **Cancel** to discard all new edges.

Centerline Tool



Centerline creates a construction edge joining the midpoints of two selected edges that belong to the same face. You cannot use **Centerline** to connect edges from different faces.

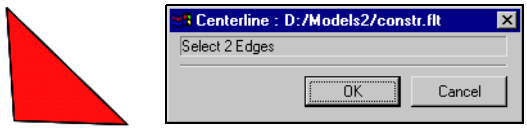
To Build Construction Edges Using Centerline



Step 1. Open the **Edge** toolbox.

Step 2. Click **Centerline**.

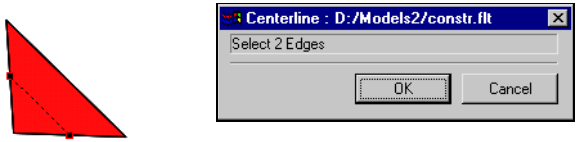
The *Centerline* dialog box prompts you to select two edges.



Step 3. Middle-click two edges that belong to the same face.



Because each edge is shared by two faces, you may find it easier to select the edges if you turn on **Shrink mode** in the View panel.



Step 4. To create more construction edges, continue selecting edges in the database. A construction edge is added between each pair of edges you select.

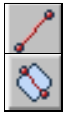
Step 5. Click **OK** to accept the construction edges and close the dialog box, or **Cancel** to discard all new edges.

Intersection of Two Planes Tool



Intersection of Two Planes creates a construction edge at the intersection of the planes defined by two selected faces. If you have not already selected the two planes, the *Construction Edge Centerline* dialog box prompts you to do so.

To Create Construction Edges at the Intersection of Two Planes



Step 1. Open the **Edge** toolbox.

Step 2. Click **Intersection of Two Planes**.

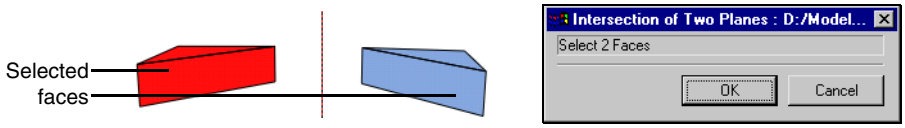
The *Intersection of Two Planes* dialog box prompts you to select two edges.





Step 3. Middle-click two faces that are not parallel to each other in the database.

The construction edge may appear far away from the selected faces if they are nearly parallel.



Step 4. To create more construction edges, continue selecting faces in the database. A construction edge is added between each pair of non-parallel faces you select.

Step 5. Click **OK** to accept the construction edges and close the dialog box, or **Cancel** to discard all new edges.

Construction Curve Tool



Builds curved construction edges.



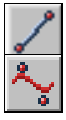
The **Construction Curve** tool is used as a construction device when creating geometries. Construction curves are not saved with the database; they are simply devices to aid in creating complex geometries. The curves created in this mode are colored red in the Graphics view with construction points that you can select and modify.



To hide or show the control points for a curve in the Graphics view, click **Draw Control Points** in the View panel. Control points allow you to determine the shape of each construction curve.

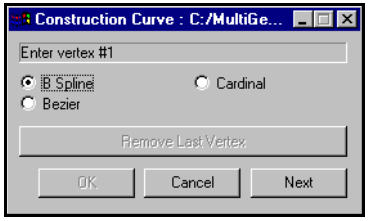
You can use a construction curve with edge tools such as **Edge from Mouse, Perpendicular to Edge**, and other tools that use the middle mouse button during construction. The middle mouse button references the curve to set the vertices onto the curve.

To Create Construction Curves

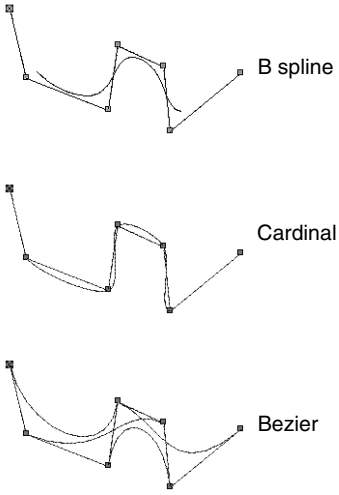


Step 1. Open the **Edge Tools** toolbox.

Step 2. Click **Construction Curve** to display the *Construction Curve* dialog box.



Step 3. Select the type of curve: **B spline**, **Cardinal**, or **Bezier**.



Step 4. In the Graphics view, click at least four points to place the control points for the curve. These control points determine the size and shape of the curve. Use **Remove Last Vertex** to erase the last control point you set.

Note: If the curve is not visible, click **Draw Control Points** in the View panel.

Step 5. Click **Next**. to create another curve, click **OK** to finish creating curves, or click **Cancel** to quit.

Step 6. Use the **Edge** tools as described in "Edge Tools" on page 209. These tools will set the vertices on the construction curve.

To Clear Construction Curves

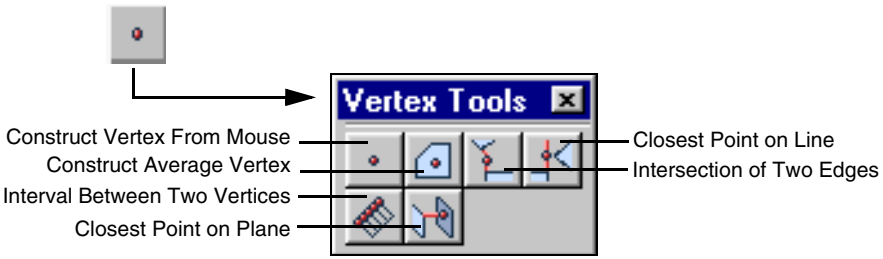
Step 1. Select the construction curve in the Graphics view.

Step 2. Choose **Edit/Clear Construction**.

Note: If nothing is selected when you choose **Edit/Clear Construction**, all construction geometry is cleared.

Vertex Tools

Vertex tools create construction vertices in the database.



You can hide and unhide construction vertices using the **Draw Constructions** toggle in the View Panel.

Construct Vertex From Mouse Tool



Vertex From Mouse places construction vertices on the tracking plane with the mouse.

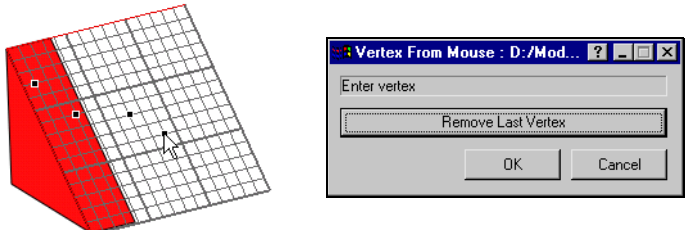
To create a vertex on the tracking plane:



Step 1. Click **Vertex From Mouse**.

Step 2. Click a location in the Graphics view to place the vertex. To remove the vertex, click **Remove Last Vertex**.

Step 3. Click another location on the tracking plane to place another vertex. When you are done creating vertices, click **OK**.



Construct Average Vertex Tool



The **Construct Average Vertex Tool**, in the **Vertex** toolbox, creates a construction vertex that is the average of all selected vertices. You do not need to select vertices to average individually; they can be part of any selected item in the database. This tool is often used to insert a construction vertex at the center of a selected face.

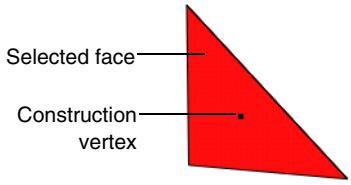
To create an average construction vertex:

Step 1. In Vertex mode, select at least two reference vertices on a face or on different faces.



Step 2. Click Average Vertex. A construction vertex will be created at an average distance between the reference points.





Averaging the Vertices of a Face

Intersection of Two Edges Tool

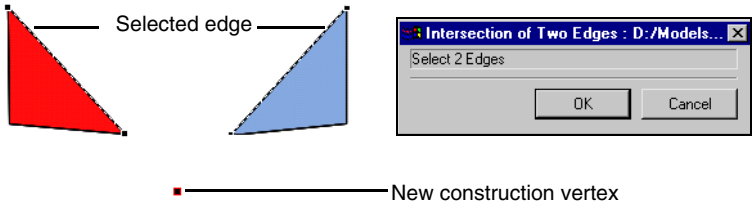


Intersection of Two Edges creates a construction vertex at the intersection of the lines defined by two edges that you select in the database.

To create a vertex at the intersection of two edges:

Step 1. Click **Intersection of Two Edges**. The *Intersection of Two Edges* dialog box opens and prompts to you to select two edges.

Step 2. Select two edges that intersect at some point. A new construction vertex is created at the point where the edges intersect.



Closest Point on Line Tool



Closest Point on a Line creates a construction vertex as close as possible to a selected vertex, along the line defined by the edge.

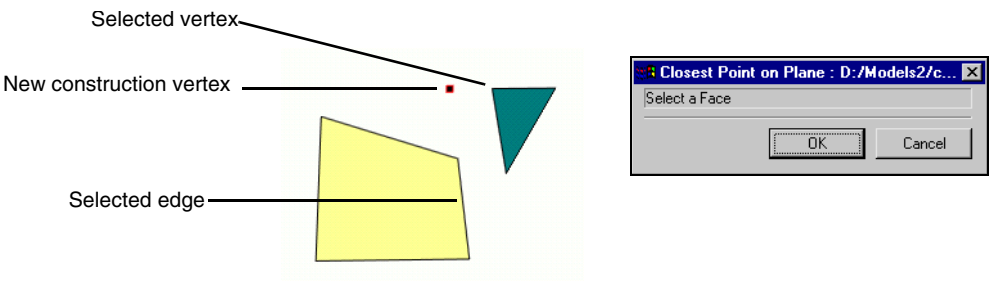
To create a vertex on an edge at the closest point to another vertex:



Step 1. Select **Closest Point on a Line**. The *Closest Point on Line* dialog box opens and prompts you to select an edge.

Step 2. Select a reference edge. The construction vertex will be placed along a line defined by this edge. The dialog box prompts you to select a vertex.

Step 3. Select a reference vertex. A construction vertex is created at a point along the line defined by the reference edge at its closest point to the reference vertex.



Interval Between Two Vertices Tool



Interval Between Two Vertices creates construction vertices at equal intervals along a line segment between two vertices you select.

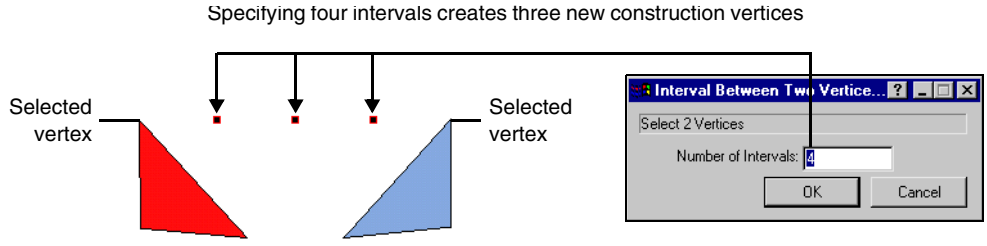
To create vertices that are evenly spaced between two vertices:



Step 1. Select **Interval Between Two Vertices**. The *Interval Between Two Vertices* dialog box opens.

Step 2. Enter a number in the **Number of Intervals** field. This is the number of times the line will be divided with a vertex placed at each division point. The number of intervals is one more than the number of vertices that will be created.

Step 3. Select two reference vertices defining the line. The new vertices will be placed along the line at evenly spaced intervals



Closest Point on Plane Tool



Closest Point on Plane places a construction vertex on a plane you indicate by selecting a face.

To create a vertex at the closest point on a plane:

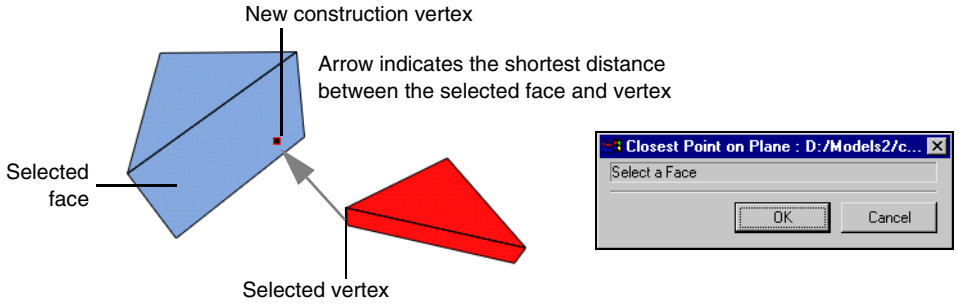


Step 1. Select **Closest Point on Plane**. The *Closest Point on Plane* dialog box prompts you to select the reference face.

Step 2. Select the face that the vertex will be created on.

The *Closest Point on Plane* dialog box prompts you to select a vertex.

Step 3. Select the reference vertex. A construction vertex is created at the closest point on the face to the reference vertex.



Saving and Restoring Construction Vertices and Edges

After you create construction vertices and edges, they are not automatically saved with the database when you close the database window. You can save construction vertices and edges to a separate file, and open this file in either the same or a different database. The construction vertices and edges retain the same database coordinates and can be reused.

To Save Construction Vertices and Edges

Step 1. Create a construction edge or vertex using a **Vertex** or **Edge** tool.

Step 2. Select **Constructions/Save Constructions** from the **File** menu. The **Save As** dialog box appears and prompts you to enter the name of the construction (.cst) file. The default directory in which to save the file is either the directory where the database is saved or the Temp directory for an unsaved database.

To Restore Construction Vertices and Edges

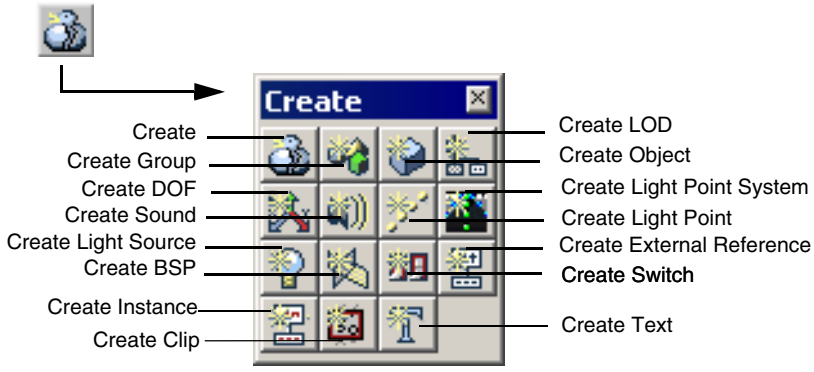
Step 1. Open either a new database file or an existing database.

Step 2. Select **Constructions/Open Constructions** from the **File** menu. The **Open** dialog box appears and prompts you for the construction (.cst) file that you want to open.

Step 3. After you click the **Open** button, the construction vertices and edges appear in your database with their original database coordinates. You can apply the same changes to them as any other construction vertex or edge.

Create Tools

Tools in the **Create** toolbox create new database nodes that define the database items being created.



Create Tool



Create (Shift+C) creates a new node in the current modeling mode and attaches it to the database parent. If the database parent is not valid and if the Flight preference **Allow Default Parent** is selected, the new node is attached to the **Default** node in the hierarchy.

You do not need **Create** to add faces or vertices to a database. Faces and vertices are created by the **Face Tools**. (See "Face Tools" on page 85.)

There are several ways you can create new nodes:

- Click **Create** to create one node in the current modeling mode.
- Hold down Ctrl and click **Create** to create multiple group or object nodes.
- Use other Create tools. For example, **Create Group** or **Create Object**.

Newly-created nodes are placed under the current Parent in the hierarchy.

To Create a New Node

Step 1. Set the parent for the new node by choosing an appropriate node in the Hierarchy view and then clicking the **Parent** button in the lower-left corner of the application window.

Step 2. Choose a mode tool from the **Mode** toolbox, or select a modeling mode from the Mode list in the Toolbar.

Step 3. Open the **Create** toolbox and click **Create** (Ctrl+C). The type of the new node is determined by the current modeling mode.



Note: You can create multiple nodes at once by holding Ctrl while clicking **Create**. Creator prompts you to enter the number of nodes to create.

Create Group Tool



Create Group creates a group node under the current parent node (if legal). You do not need to be in Group mode to use this tool. Hold down Alt while clicking **Create Group** to create a group and simultaneously set it as the current parent.

Create Object Tool



Create Object creates an object node under the current parent. You do not need to be in Object mode to use this tool. Hold down Alt while clicking **Create Object** to create an object and simultaneously set it as the current parent.

Create LOD Tool



Creates a Level-of-Detail (LOD) node attached to the current database parent.

See “Creating LODs” on page 707 for complete information.

Create DOF Tool



Creates a Degree-of-Freedom (DOF) node attached to the current database parent.

See “Degrees of Freedom” on page 735 for complete information about DOFs.

Create Sound Tool



Creates a sound node attached to the current database parent.

See “Audio Modeling” on page 669 for complete information about sound nodes.

Create Light Point Tool



Creates sets of non-radiant light points in the database.

See “Light Points and Light Strings” on page 631 for complete information.

Create Light Point System Tool



Creates a group node attached to the current database parent for light points. The Vega Prime loader uses this node to identify light points.

See “Light Points and Light Strings” on page 631 for more information.

Create Light Source Tool



Creates a light source node attached to the current database parent.

See “Light Sources and Database Shading” on page 599 for complete information.

Create BSP Tool



Creates a BSP (Binary Separating Plane) node attached to the current database parent. This tool requires the BSP option.

See “Binary Separating Planes” on page 907 for complete information.

Create Switch Tool



Creates a switch node attached to the current database parent.

See “Switch Nodes and Switch Attributes” on page 861.

Create External Reference Tool



Creates an external reference node, attached to the current parent, which references geometry in another database.

See “External References” on page 370 for more information.

Create Instance Tool



Changes the selected node into an instance.

See “Instances” on page 368 for more information.

Create Clip Node Tool



Creates a clip node. This tool is part of the Instrumentation option. See “Clipping Nodes and Planes” on page 802 for more information.

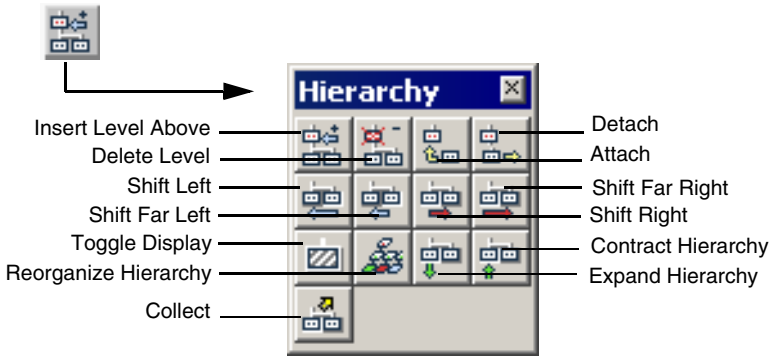
Create Text Node Tool



Creates a text node for adding dynamic text to a database. This tool is part of the Instrumentation option. See “3D Text Tool” on page 119 for more information.

Hierarchy Tools

The **Hierarchy** toolbox contains tools for executing specific editing actions, such as moving nodes to different areas of the hierarchy. To use the Hierarchy tools, select the node or nodes you want to edit and choose one of the following tools:



Insert Level Above Tool



Creates and inserts a new parent node for each selected node. The type of the new parent is determined by the current modeling mode.

Delete Level Tool



Deletes the selected node and attaches its children to the selected node’s parent.

Attach Tool



Moves the selected node to the current parent.

Detach Tool



Separates the selected node from its parent, creating an orphan.

Shift Left Tool



Exchanges the selected node with the sibling on its immediate left.

The effects of **Shift Left** can be restricted by the Flight preference **Check Relative Priority When Shifting Nodes Left or Right**. When this preference is set, Creator checks the value of each shifted node's **Relative Priority (Fix Listing)** attribute. A node cannot be put in a position that allows it to be processed ahead of nodes with a higher relative priority value.

For more information on relative priority and database traversal (the processing of nodes), see "Draw Order" on page 682 and "Database Traversal Order" on page 688.

Shift Far Left Tool



Makes the selected node the leftmost child of its parent.

The effects of **Shift Far Left** can be restricted by the Flight preference **Check Relative Priority When Shifting Nodes Left or Right**. When this preference is set, Creator checks the value of each shifted node's **Relative Priority (Fix Listing)** attribute. A node cannot be put in a position that allows it to be processed ahead of nodes with a higher relative priority value.

For more information on relative priority and database traversal (the processing of nodes), see "Draw Order" on page 682 and "Database Traversal Order" on page 688.

Shift Right Tool



Exchanges the selected node with the sibling on its immediate right.

The effects of **Shift Right** can be restricted by the Flight preference **Check Relative Priority When Shifting Nodes Left or Right**. When this preference is set, Creator checks the value of each shifted node's **Relative Priority (Fix Listing)** attribute. A node cannot be put in a position that allows it to be processed ahead of nodes with a higher relative priority value.

For more information on relative priority and database traversal (the processing of nodes), see "Draw Order" on page 682 and "Database Traversal Order" on page 688.

Shift Far Right Tool



Makes the selected node the rightmost child of its parent.

The effects of **Shift Far Right** can be restricted by the Flight preference **Check Relative Priority When Shifting Nodes Left or Right**. When this preference is set, Creator checks the value of each shifted node's **Relative Priority (Fix Listing)** attribute. A node cannot be put in a position that allows it to be processed ahead of nodes with a higher relative priority value.

For more information on relative priority and database traversal (the processing of nodes), see "Draw Order" on page 682 and "Database Traversal Order" on page 688.

Toggle Display Tool



Turns the display of selected items on or off in the Graphics view.

Reorganize Hierarchy Tool



Rebuilds the hierarchy attached to all levels of detail in a database to optimize for realtime culling.

See "Editing the Hierarchy Structure" on page 287.

Expand Hierarchy Tool



Displays child nodes by level, or displays child nodes of the selected parent.

See “Expanding and Collapsing Hierarchy Levels” on page 281.

Contract Hierarchy Tool



Hides child nodes by level, or hides child nodes of the selected parent.

See “Expanding and Collapsing Hierarchy Levels” on page 281.

Collect Tool



Selects nodes in the hierarchy based on a search requirement that you specify, and automatically rearranges them under a new group or object node. The default name of the node is **collect**, which you can change in the *Collect Tool* dialog box.

You first set a group node as a parent. You can select nodes manually either in the Graphics or Hierarchy view, or Creator can recursively search and select all the faces within the group node that are connected to the selected face, or all the connected faces that share manifold edges. See “Select Connected Faces” on page 337 and “Select Non-Manifold Faces” on page 338 for explanations of connected faces, and manifold faces and edges.

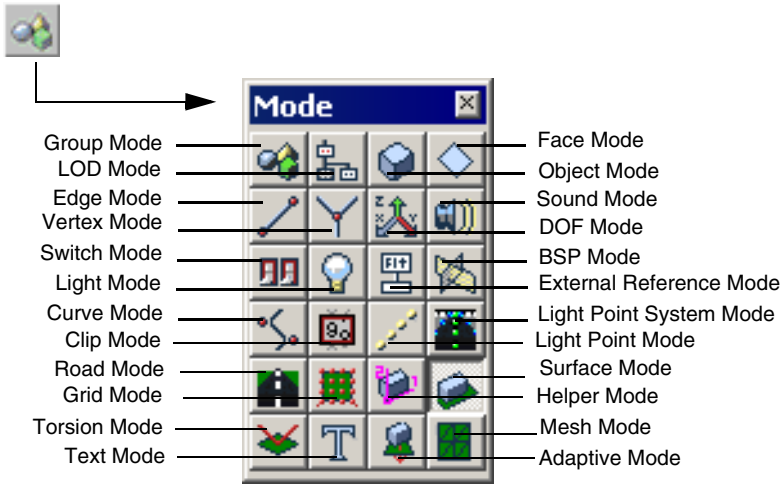
You can collect nodes for only one new node, or continue collecting for several new nodes while in the tool.



Mode Tools

Mode tools reflect the levels of organization available in the database hierarchy. When you select a tool in the **Mode** toolbox, you set the corresponding modeling and selection modes. One of these modes is active (or current) at all times.

The unlabeled tools in this toolbox appear when Creator Pro options are installed.



Mode tools affect many modeling and selection functions. To change the mode in which an item is selected, select a new modeling mode in the **Mode** toolbox and choose **Select/ Change Mode** (Ctrl+Shift+C). For example, if you select an object in Object mode, change to Vertex mode (Shift+V), and choose **Change Mode**, the vertices of the original object will be selected.

Group Mode Tool



Groups can contain other groups (or group-like nodes), levels of detail, and objects. You must be in Group mode (Shift+G) to select groups in the Graphics view. Group IDs are color-coded red in the Hierarchy view.

Collecting sets of logically connected objects into groups makes them easier to manipulate in the database. The number and location of objects included in each group also affect culling and drawing in the realtime system.

LOD Mode Tool



LOD Mode is used to create levels of detail and to select them in the Graphics view.

See "Selecting LODs" on page 711.

Object Mode Tool



Object mode (Shift+O) is used to create objects and to select them in the Graphics view. An object is a collection of polygons.

Face Mode Tool



A *face* is a collection of ordered, coplanar vertices describing a surface. Faces can be attached to objects or groups. You must be in Face mode (Shift+F) to select faces in the Graphics view.

Edge Mode Tool



When Creator is in Edge mode, you can select edges in the Graphics view. An edge is the line segment between two consecutive vertices in the same face. Edges are not defined as part of the database hierarchy, however several edge functions are provided to facilitate modeling.

Vertex Mode Tool



A vertex is a coordinate point in the database. To select vertices in the Graphics view you must be in Vertex mode.

Vertices are created during a number of Creator edit functions, usually as part of the process of creating a face. Construction vertices are also used, but these are not saved with the database.

DOF Mode Tool



DOF Mode is used to create degrees of freedom and to select them in the Graphics view.

See “Degrees of Freedom” on page 735.

Sound Mode Tool



The **Sound Mode** tool in the Mode toolbox is used to create sound nodes and to select sound nodes in the Graphics view.

See “Sound Nodes” on page 672.

Switch Mode Tool



The **Switch Mode** tool in the Mode toolbox is used to create switches and select them in the Graphics view.

See “Switch Nodes and Switch Attributes” on page 861.

Light Mode Tool



The **Light Mode** tool in the Mode toolbox is used to create light sources and select light sources in the Graphics view.

See “Light Sources” on page 599.

External Reference Mode Tool



The **External Reference Mode** tool in the Mode toolbox is used to create external reference nodes and select external references in the Graphics view.

See “External References” on page 370.

BSP Mode Tool



BSP Mode is used to manually create BSP planes in the database.

See “Manually Separating a Database” on page 922.

Curve Mode Tool



Curve Mode is used to create and select geometry and construction curves.

See “Curve Tool” on page 116.

Clip Mode Tool



Clip Mode is used to select the clipping nodes you create with the Instruments option.

See “Clipping Nodes and Planes” on page 802 for more information.

Light Point Mode Tool



Light Point Mode is used to create light points and select light points in the Graphics view.

See “Light Point Tools” on page 660.

Light Point System Mode Tool



Light Point System Mode is used to create light points in the Graphics view for a light point system node.

See “Light Point Tools” on page 660.

Road Mode Tool



Road Mode allows you to select road sections. Requires the Road Tools option.

Grid Mode Tool



Grid Mode sets the current mode to **Grid** and allows you to display the attributes of a SmartScene Grid node.

Helper Mode Tool



Helper Mode sets the current mode to **Helper** and allows you to display the attributes of a SmartScene Helper node.

Surface Mode Tool



Surface Mode sets the current mode to **Surface** and allows you to display the attributes of a SmartScene Surface node.

Torsion Mode Tool



Torsion Mode sets the current mode to **Torsion** and allows you to display the attributes of a SmartScene Torsion node.

Text Mode Tool



Text Mode allows you to select 3D text in the Graphics view.

Adaptive Mode Tool



Adaptive Mode is used with Continuous Adaptive Terrain (CAT). When selected, you can create and select an adaptive attribute that can be attached to any level of the hierarchy. CAT is part of the TerrainPro option.

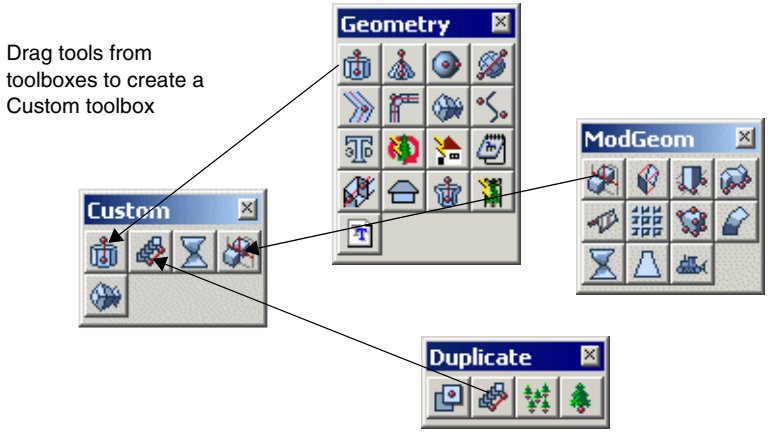
Mesh Mode Tool



Mesh Mode sets the current mode to **Mesh** and allows you to generate mesh for models and terrain to render quickly. Individual Face nodes combine into a single Mesh node.

Custom Toolbox

You can create a custom Toolbox with the tools that you commonly use. This is useful when you want to display only one Toolbox. Your custom Toolbox is saved when you close Creator.



To Create a Custom Toolbox



Step 1. Click the **Custom Toolbox** tool.

Step 2. Open the Toolboxes that have the tools that you want to add.

Step 3. Tear both the Custom Toolbox and the source Toolboxes from the Toolbox caddy with the left mouse button.

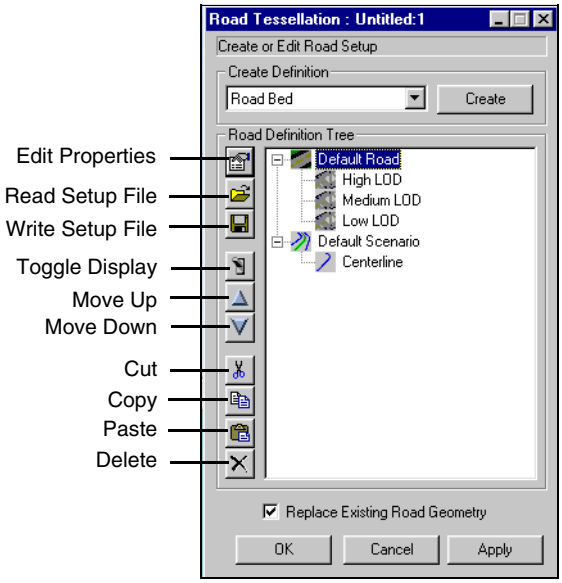
Step 4. Hold down the Shift key and drag tools from the source Toolboxes with the left mouse button onto the Custom Toolbox palette.

Note: You cannot add **Mode** tools to your custom Toolbox.

Step 5. Delete tools by dragging them off of the Custom Toolbox with the left mouse button and holding down the Shift key.

Tessellation Tools

Tessellation tools, which appear as buttons in the *Road Tessellation* dialog box, create and edit road attributes. See "Road Tessellation" on page 1252.



- **Edit Properties**



Opens the setup definition of the selected road component.

- **Read Setup File**



Loads a saved road definition setup file (*.rds).

- **Write Setup File**



Saves the current road definition in a setup file (*.rds).

- **Toggle Display**



Turns the display of the selected road component on or off.

- **Move Up**



Moves the selected road component to the next higher position in the hierarchy (if it is legal).

- **Move Down**



Moves the selected road component to the next lower position in the hierarchy (if it is legal).

- **Cut**



Removes the selected road component from the hierarchy and places it on the clipboard.

- **Copy**



Copies the selected road component and places the copy on the clipboard.

- **Paste**



Places the road component on the clipboard at the specified location in the **Road Definition Tree**.

- **Delete**



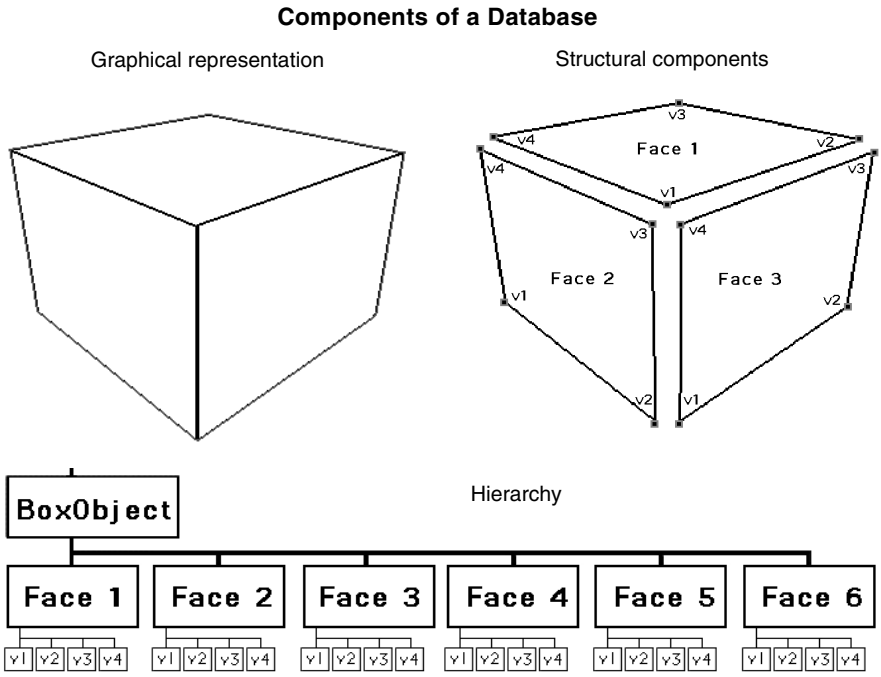
Removes the selected road component from the **Road Definition Tree**.

5 Working with Database Files

Creator database files use the OpenFlight scene-description format for general purpose modeling. These files can be copied or moved to other directories, or renamed, using standard system tools.

An OpenFlight database uses geometry, hierarchy, and attributes to define models used in realtime applications. Each component provides a unique contribution to the model.

- *Geometry* defines the model in three dimensions as a set of ordered coordinate points. For example, each side of a box is made up of four coordinate points, or vertices.
- *Hierarchy* defines additional structures that combine the vertices of the model into logical units that are easily edited and displayed. The vertices of each side of a box make one *face* (or polygon), and the faces of the box make one object. The term *node* refers to each of these structural elements in the hierarchy (see "Database Node Types" on page 292).
- Each node in a database has a unique name or *identifier* (ID) that has a maximum length of 512 characters. IDs are assigned automatically, but can be changed by editing the node attribute (see "Database Identification Names" on page 289), or by choosing **Attributes/Rename** (Ctrl+J). You cannot rename vertex IDs, however.
- *Attributes* provide additional data to define the graphical, physical, and theoretical characteristics of each node (see "OpenFlight Nodes and Attributes" on page 825).

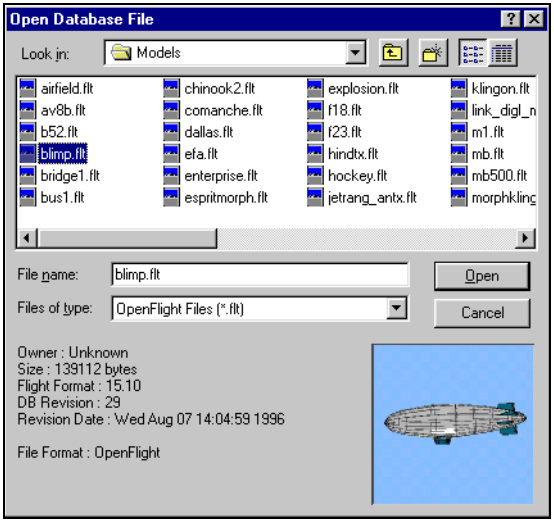


Opening Database Files



To open an existing database file, choose **File/Open** (Ctrl+O) or click the **Open Database File** button on the Toolbar. Use the controls in the *Open Database File* dialog box to select and open a file. The lower-left portion of this window displays information about the

selected file. The lower-right corner may show a graphical thumbnail snapshot of the file's contents (see "Using Snapshot" on page 248).



Note: The *Open Database File* dialog box initially shows the directory defined by the FLTPATH environment variable (see "Environment Variables" on page 43).

By default, only files that have the .flt extension are shown. To see all of the files in the current directory, choose **Files of type/All Files (*)**. However, only files having the OpenFlight format will open with **File/Open**. To open files of a different file format, choose **File/Import**.

Opening Recent Files

You can also open recently used files by choosing **File/Recent Files** and then choosing a file from the **Recent Files** submenu.

Creating New Database Files



To create a new database file, choose **File/New** (Ctrl+N) or click the **New Flight File** button. A new Database window appears, and the file is given a default file name.

Undoing Work



As you perform operations in Creator, each operation is posted in the **Edit/Undo History** submenu. These operations form a *stack*. Each new operation is added to the top of the stack, moving previous ones down. You can undo the most recent operation by choosing its name from the top of the list in the **Undo History** submenu, by choosing **Edit/Undo** (Ctrl+Z), or by clicking **Undo** in the Toolbar.

You can also choose to undo any operation listed in **Edit/Undo History**. However, any operation you undo using **Edit/Undo History** will undo all operations above it. *There is no command to restore undone operations; you must manually redo the work to restore it.*

The **Max Undos** Editing preference sets the number of operations that can appear in the **Edit/Undo History** submenu. The **Max Undos** setting should reflect the amount of memory available on your computer. The amount of memory required to save each undo operation depends on the scope of the operation; for example, deleting a large portion of a database requires much more memory than changing the color of a face.

Saving Database Files

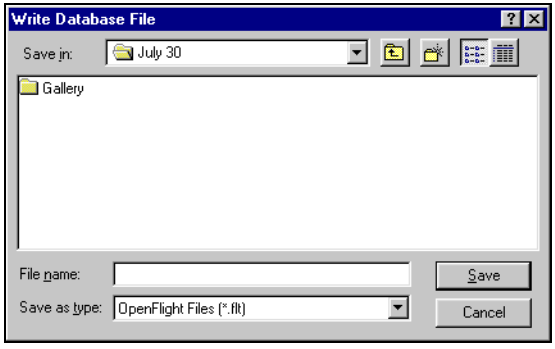


To save a database file, choose **File/Save** (Ctrl+S) or click the **Save Flight File** button on the Toolbar. The file is saved only if changes have been made since the last save. Files are saved in the OpenFlight format.

You can set Creator to save open database files regularly, and to back up database files automatically on opening. See “To Save and Back Up Files Automatically” on page 245.

To Save a New File

Step 1. Choose **File/Save** (Ctrl+S), or click the **Save Flight File** button to open the *Write Database File* dialog box. This dialog box will open only if changes have been made to the new file.



Step 2. Choose a folder in which to save the file.

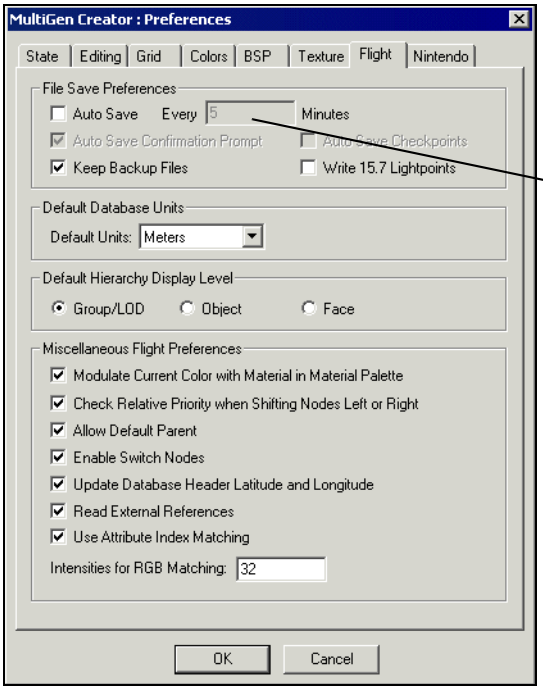
Step 3. Enter a name for the file in the **File name** text field, ending with .flt.

Step 4. Click **Save** to save the file and close the *Write Database File* dialog box.

To Save and Back Up Files Automatically

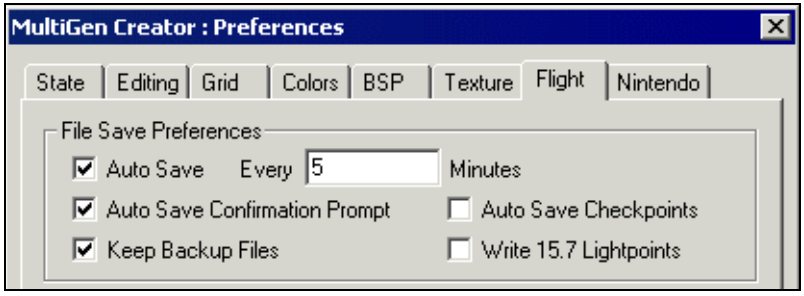
Step 1. Choose **Info/Preferences** (Ctrl+Shift+P) to open the *Preferences* window.

Step 2. Click the **Flight** tab to go to the Flight panel.



The text field for setting the Auto Save interval is dimmed until Auto Save is set.

Step 3. Set **Auto Save**. The **Every N Minutes** field becomes available, where *N* is a variable representing the value in the preference's text field.



Step 4. Enter a number in the **Every N Minutes** text field. This value determines the interval (in minutes) between automatic file saving operations. The

database is automatically saved every *N*minutes if it has not been manually saved and if a change has been made to the contents of the database.

Note: Using a very small number may cause Creator to save too often, possibly interfering with your computer's performance.

Step 5. Set **Auto Save Confirmation Prompt** if you want Creator to generate an informational message each time an automatic save is done. This message appears in the Status bar and is also logged.To see logged messages, choose **Info/Status Log**.

Step 6. To create backup copies of database files automatically, set **Keep Backup Files**. When a database is saved for the first time after being opened, this preference causes Creator to save a copy of the original database with the .bak extension.

Step 7. Set **Auto Save Checkpoints** if you want to save checkpoint files at regular intervals. This option is useful if you want the possibility of reverting to previous versions of a database file; yet it requires considerable disk space, since it results in numerous copies of the file being written to your hard drive. The checkpoint file receives the same name as the original file, with a numerical extension added to the filename. The new file then becomes the database in the Creator window, as you continue your work.

The **Write 15.7 Lightpoints** option is provided for compatibility with applications still using OpenFlight specs with 15.7 light points. Unless you specifically need this option leave it unchecked. (If you are using Vega Prime, be sure that this box remains unchecked.)

Saving With a New Name

To save a database file with a new name, choose **File/Save As** and type the name of the new file in the *Write Database File* dialog box's **File name** text field. End the file name with .flt.

Saving Portions of a Database

Using **Write Select**, you can save selected portions of a database file into a set of new database files. For more information on using **Write Select**, see "To Save With Write Select" on page 296.

Saving Checkpoint Files

To save a database file as a checkpoint file, choose **File/Save Checkpoint**, and click **Yes** when the *Verify* message displays. The checkpoint file receives the same name as the original file, with a numerical extension added to the filename (001-999). This new file becomes the database in the Creator window, as you continue your work.

Using Snapshot

You can save a thumbnail image of the Graphics view in a file with the same name as the database file, but with the extension .icn. This image is displayed in the lower-right corner of the *Open Database File* dialog box when you select the database file in the dialog box's file list.

To Take a Snapshot

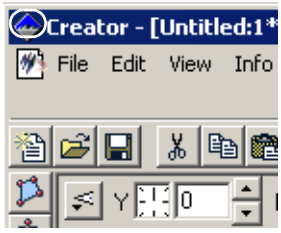
Step 1. Arrange the Graphics view to show a representative view of the database. For more information on how to work in the Graphics view, see "Working in the Graphics View" on page 251.

Step 2. Choose **View/Snapshot**.

Closing a Database

To close a database file, choose **File/Close** (Ctrl+W). If the database contains unsaved changes, a dialog box prompts you to save your work. You can choose to save before closing, to close without saving, or to cancel the close operation.

You can also close the Database window by clicking its standard Windows **Close** button or by double-clicking its **Document** menu icon.



Windows

The Document Menu Icon (Circled)

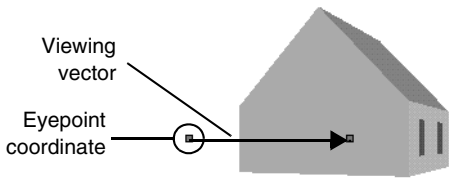
Note: If you close the Application window, the size and position of the window, including positions of torn-off toolboxes, are restored when you reopen the Application window.

6 Working in the Graphics View

The Graphics view provides a three-dimensional drawing of your database. The following topics describe controls to set viewing and drawing parameters and extend the usability of the Graphics view. In general, these controls affect only the active pane.

Adjusting The Eyepoint

The *eyepoint* is the spatial point in the Graphics view from which you observe the database drawing. As its name suggests, the eyepoint is where the observer's eye is located. The *viewing vector* is a line segment that extends from the eyepoint to the point in the database at which the viewer is looking. The Graphics view is the graphical representation of a database as it appears from the eyepoint coordinate. This coordinate is shown in the **X**, **Y**, and **Z** text fields of the Control bar.

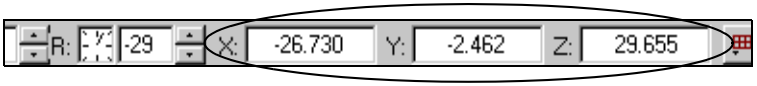


Changing the Eyepoint Position

Locating and changing the eyepoint position is one of the most common operations you are likely to perform in Creator. There are several ways of changing the eyepoint position.

The Eyepoint Coordinates

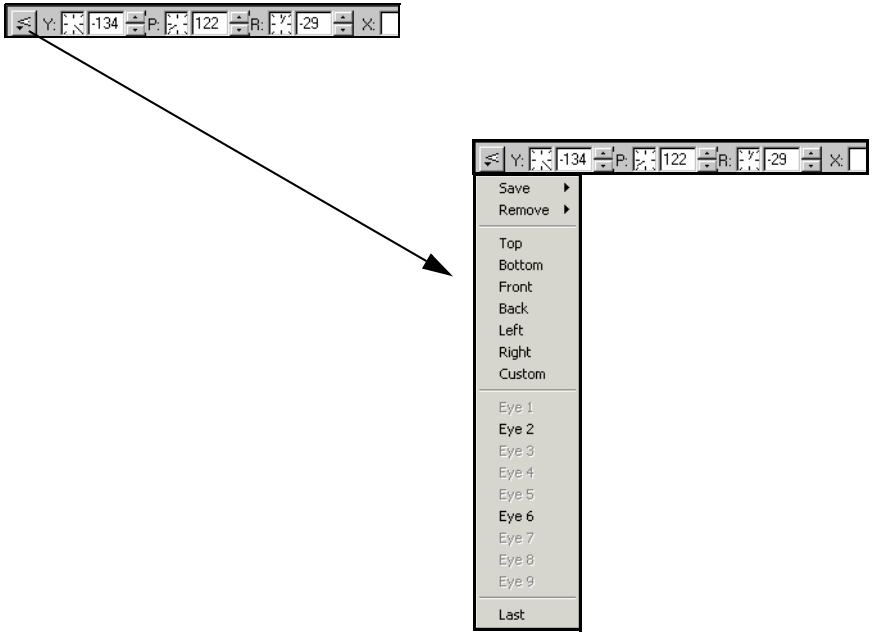
The current eyepoint position is shown as coordinates in the **X**, **Y**, and **Z** text fields of the Control bar. When the eyepoint position changes, its coordinates are immediately updated in the coordinate text fields.



To change the eyepoint position to a specific point, you can enter new values for the *x*, *y*, and *z* coordinates in the **X**, **Y**, and **Z** text fields of the Control bar. You can use the Tab key to move forward through these fields, and Shift+Tab to move back.

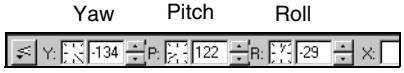
The Eyepoint Menu

You can change the eyepoint position by choosing any one of seven preset views from the **Eyepoint** menu. To open the **Eyepoint** menu, click the **Eyepoint** menu button on the Control bar.



The **Eyepoint** menu contains **Top**, **Bottom**, **Front**, **Back**, **Left**, and **Right** positioning choices. Each choice yields an orthographic view of the selected item from the point of view indicated by its name. The view is centered on the selected items. If nothing in the Graphics view is selected, then the database origin is taken as the center.

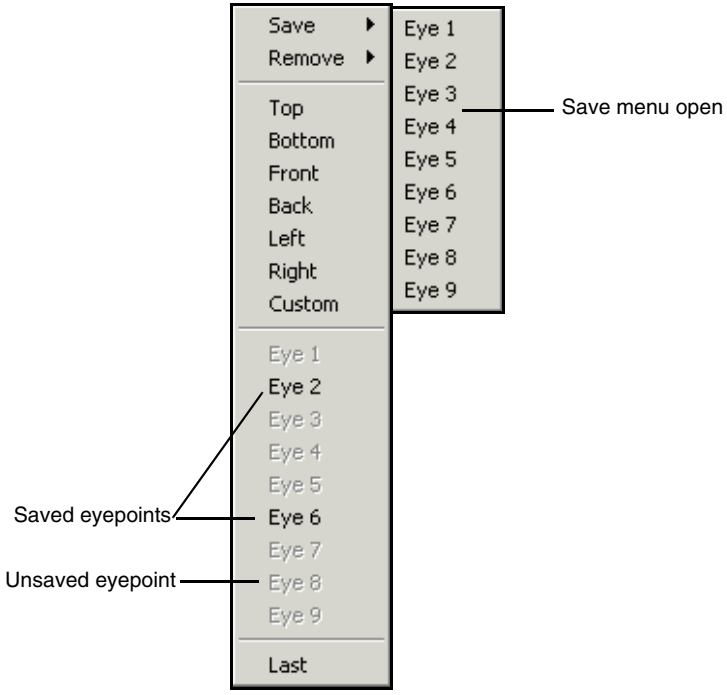
Choosing **Last** returns the eyepoint to the last eyepoint position. Choosing **Custom** from the **Eyepoint** menu changes the view from orthographic to perspective view, so you can rotate the eyepoint as usual. Rotating the eyepoint using the **Yaw**, **Pitch**, and **Roll** controls on the Toolbar also changes into perspective view.



See “Rotating the Eyepoint” on page 255.

Storing, Recalling, and Removing Eyepoint Positions

You can store up to nine custom eyepoint positions in the **Eyepoint** menu. To store an eyepoint position, choose **Eyepoint/Save/Eye k**, where *k* is a number from one to nine. The active pane’s eyepoint position is saved as **Eye k**, which becomes available as a choice in the Eyepoint menu.



To recall an eyepoint position, choose **Eyepoint/Eye k**, where *k* is a number from one to nine.

You can also store and recall eyepoint positions using the keyboard numbers. You cannot use the keypad to store and recall eyepoint positions.

- To store an eyepoint position, press Shift+number.
- To recall an eyepoint position, press the number.
- To recall the last eyepoint position, press 0.

To remove a custom eyepoint position, choose **Eyepoint/Remove/Eye k**, where *k* is the number of the eyepoint you want to remove. If you want to remove all custom eyepoints, choose **Eyepoint/Remove/All**.

Zooming in the Graphics View

Zooming in the Graphics view is the same as zooming in the Hierarchy view. Using **Zoom View**, you can zoom the eyepoint in (toward) or out (away from) the database. When **Zoom View** is active, the pointer turns into a magnifying glass containing a plus sign (zoom in) or a minus sign (zoom out). For using **Zoom View** in the Hierarchy view, see "Zooming in the Hierarchy View" on page 285.

To use **Zoom View**:



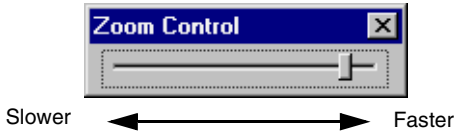
Step 1. Click **Zoom View** in the Toolbar or simultaneously hold down the Shift key and the right mouse button. The pointer turns into a magnifying glass.

Step 2. Drag the magnifying glass down to zoom in, or drag it up to zoom out.

Note: Holding down the middle mouse button while zooming makes the zoom action continuous, making it unnecessary to continue moving the magnifying glass.

Zoom Speed Control

The speed of **Zoom View** can be set by choosing **View/Zoom Control** and adjusting the slider in the *Zoom Control* window. Moving the slider to the right speeds up the zooming action, while moving it to the left slows down the zooming action.



Panning in the Graphics View

Using **Pan View**, you can move the eyepoint up, down, left, and right to bring other parts of the database into view. When **Pan View** is active, the pointer turns into a cross with arrows. For using **Pan View** in the Hierarchy view, see "Panning in the Hierarchy View" on page 286.

To use **Pan View**:



Step 1. Click **Pan View** in the Toolbar or simultaneously hold down the Ctrl key and the right mouse button. The pointer turns into a cross with arrows.

Step 2. Drag the pointer in the direction you want to move.

Note: Holding down the middle mouse button while panning makes the pan action continuous, making it unnecessary to continue moving the pointer.

Rotating the Eyepoint



Rotating the eyepoint lets you view your work from different points in space. You can rotate the eyepoint by dragging the pointer when the **Rotate View**button in the Toolbar is selected.

When **Rotate View** is not selected, you can rotate the eyepoint by *right-dragging* (holding down the right mouse button and dragging the pointer in the Graphics view).

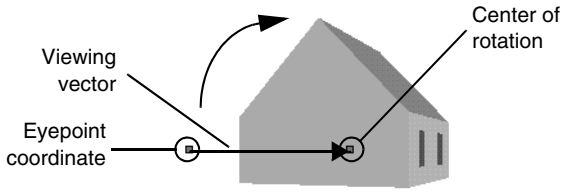
Pressing and holding the middle mouse button while right-dragging rotates the eyepoint continuously, making it unnecessary to repeatedly click and drag the pointer.

Note: In a large database, the eyepoint may rotate more smoothly and quickly if you turn on **Dynamic Wireframe** in the View panel. This feature temporarily changes the drawing mode to wireframe when the eyepoint is in

motion, which reduces the computing resources needed to display the scene.



The eyepoint can rotate in any direction around the *center of rotation*. The center of rotation is usually (but not always) the point you are looking at in the drawing, which is the end of the viewing vector.



One Possible Eyepoint Rotation.

By default, the center of rotation is at the origin of a newly opened database (point coordinates 0,0,0). To see how this works, try the following:

Step 1. Open a new database file and turn on the grid in the View panel. By default, the tracking plane starts with its origin at the database origin.

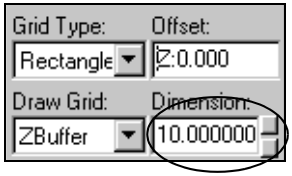


Leave the View panel open to use for some of the following steps.

Note: You can also alternately hide and restore the View panel by pressing F5.

Step 2. Make the grid small enough so that the entire grid is only about one third the width of the pane you are working in. Make the grid units large enough so that you can easily see the grid's origin.





Step 3. Create a small cube just outside the grid.

Step 4. Right-drag with the mouse to rotate the view, and note that the cube is rotating about the grid's origin (which currently is at the same point as the database origin).

Step 5. Choose **Trackplane from Face** in the View panel. The *Trackplane from Face* dialog box opens. Make sure the **Exit When Face Selected** checkbox is set.



Step 6. Choose **Face** from the **Mode** menu and select a face on the cube. The *Trackplane from Face* dialog box closes.

Step 7. Rotate the eyepoint and note that the grid's origin is no longer the center of rotation, which remains at the database origin.

The Yaw, Pitch, and Roll Clocks

You can rotate the eyepoint around a particular axis by using the **Yaw**, **Pitch**, and **Roll** clocks on the Control bar. *Yaw*, *pitch*, and *roll* refer to rotation around the *z*, *x*, and *y* axes, respectively.

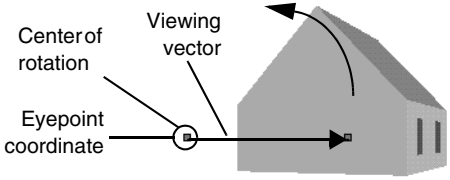
Changing the Center of Rotation

There are several ways to reposition the center of rotation, all of which involve changing the position of the eyepoint.

> **Note:** The center of rotation does not change when you change the eyepoint position using the **Yaw**, **Pitch**, and **Roll** controls.

When you change the **X**, **Y**, and **Z** text fields in the Control bar, the eyepoint becomes the center of rotation. In this case, you can rotate the database drawing around the eyepoint.

There are other view options that can reposition the center of rotation. To see how some of them work, try the following:



The Eyepoint is the Center of Rotation.



Step 1. Open a new database file and turn on the grid in the View panel. By default, the tracking plane starts with its origin at the database origin.

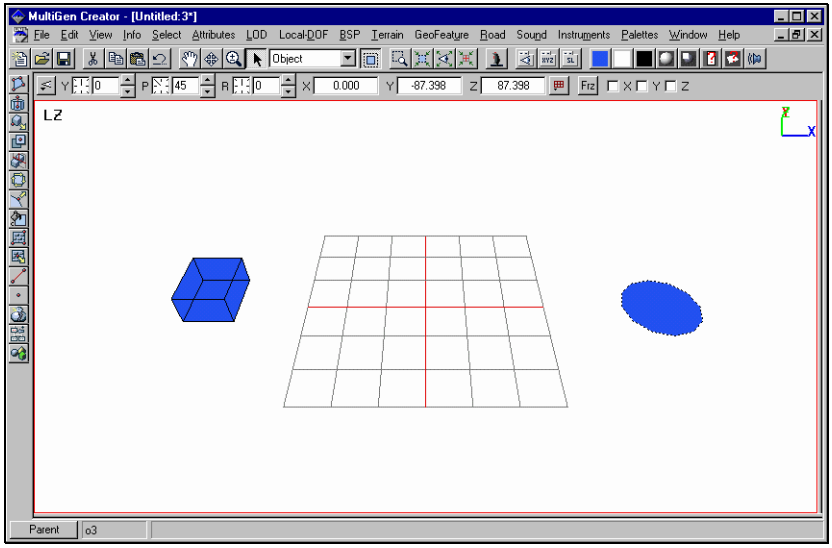
Leave the View panel open to use for some of the following steps.

Note: You can also alternately hide and restore the View panel by pressing F5.



Step 2. Make the grid small enough so that the entire grid is only about one third the width of the pane your are working in. Make the grid units large enough so that you can easily see the grid's origin.

Step 3. Create a small cube and a circle just outside and on opposite sides of the grid.



Step 4. Right-drag with the mouse to rotate the view, and notice that the cube and circle are rotating about the grid's origin (which currently is at the same point as the database origin).

Step 5. With both the circle and the cube in view, and the **Mode** list set to **Object**, select the circle.

Step 6. Choose **View/Fit on Pick** (Ctrl+F), or click the **Fit on Pick** control in the Toolbar, and drag a fence around a portion of the cube. The Graphics view centers the cube and zooms in on it, but the cube is not selected.



Step 7. Rotate the eyepoint and note that the center of rotation is at the center of the cube and that the circle is still selected. (You may need to zoom out to see both the cube and the circle.)

Step 8. Choose **View/Fit on Select** (V), or click the **Fit on Select** control in the Toolbar. Note that the circle is now centered in the Graphics view and that the center of rotation is at the center of the circle.



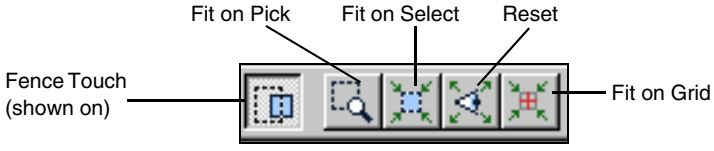
Step 9. Choose **View/Fit on Grid** (Ctrl+Shift+V), or click the **Fit on Grid** control in the Toolbar. Note that the grid is now centered in the Graphics view and that the center of rotation is back at the grid's origin.



View Options



View menu options produce a particular database view while also repositioning the center of rotation. **View** menu options that depend on selecting a database item are affected by the **Fence Touch** control. If **Fence Touch** is off, the selection fence must contain an entire item to select it. **Fence Touch**, and some of the **View** menu options, are available in the Toolbar.



Fit on Pick



Centers and scales the view on a database item without selecting the item. Items that are already selected will remain selected.

To use **Fit on Pick**, choose **View/Fit on Pick** (Ctrl+F) or click the **Fit on Pick** button in the Toolbar. The pointer changes to a cross. Pick an item by dragging a fence around it in the Graphics view. The type of item you can pick depends on the current modeling

mode. The item you pick is centered and enlarged in the active pane. The center of rotation moves to the center of the picked item.

Fit on Grid



Centers and scales the view on the grid, with the center of rotation at the grid's origin. Selected items remain selected.

To perform this operation, choose **View/Fit on Grid** (Ctrl+Shift+V) or click the **Fit on Grid** button in the Toolbar.

Fit on Select



Centers and scales the view of the selected database item, so that it just fits into the view, and places the center of rotation at the item's center.

To perform this operation, select the item to fit on, and then choose **View/Fit on Select** (V) or click the **Fit on Select** button in the Toolbar.

Center Select

Centers the selected database item in the view but does scale it (as in **Fit on Select**), and places the center of rotation at the item's center.

To perform this operation, select the item to center on, and then choose **View/Center Select** (C).

Reset View



Restores the default database view specified in the **Graphic View Defaults** State preferences (see "Understanding the Default Eyepoint Position" on page 262).

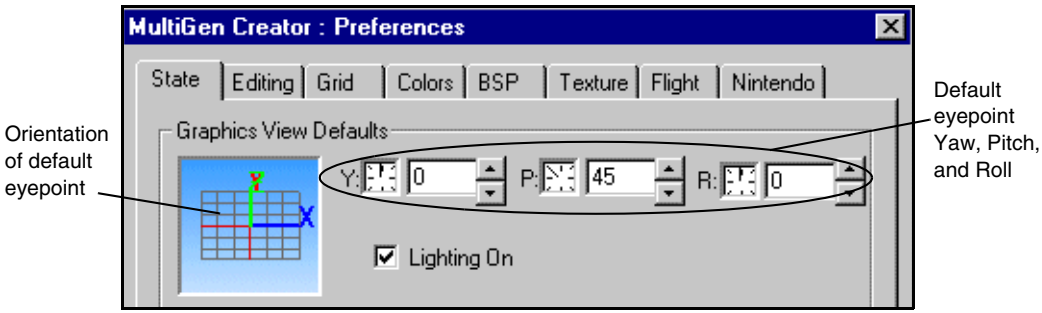
To perform this operation, choose **View/Reset** (U) or click the **Reset** button in the Toolbar.

Understanding the Default Eyepoint Position

The default eyepoint position is set using the **Graphics View Defaults**, located in the State preferences panel. To open the State preferences panel, choose **Info/Preferences** and click the **State** tab.



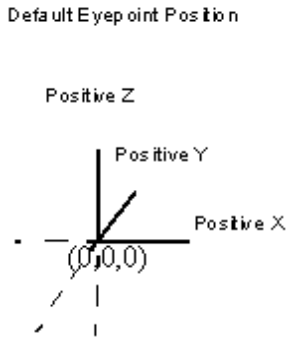
The default eyepoint position is used when a Graphics view is first opened and when you choose **View /Reset** (U) or click **Reset View** in the Toolbar.

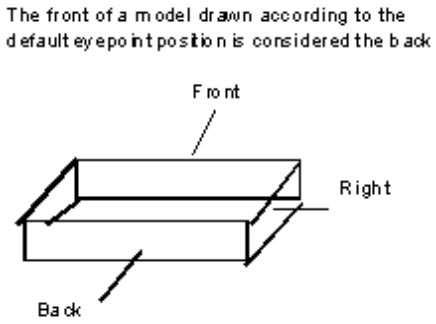


In the **Graphics View Defaults** area of the State preferences panel, the small coordinate axes to the left of the **Yaw**, **Pitch**, and **Roll** controls show the default eyepoint position relative to the database coordinates.

The orientation of the default eyepoint is considered to be "z-up," where the x axis is horizontal, the z axis is vertical and perpendicular to the x axis, and the y axis intersects both the x and z axes at a 45 degree angle. In the **Eyepoint** menu, **Front** is looking at the model from the positive y axis, **Right** is looking at the model from the positive x axis, and **Top** is looking at the model from the positive z axis. **Back, Left, and Bottom** are all looking at the model from the corresponding negative axes.

When you create a model in the Graphics view according to the default eyepoint position (positive x to the right, positive y in back, and z-up), the front of the model that faces you is considered the back of the model in Creator, since **Front** looks at the model down the positive y axis.

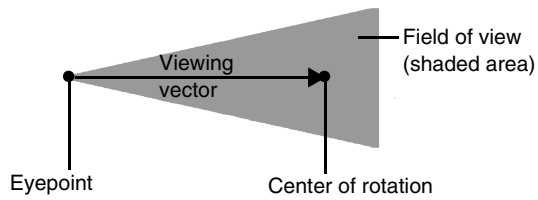




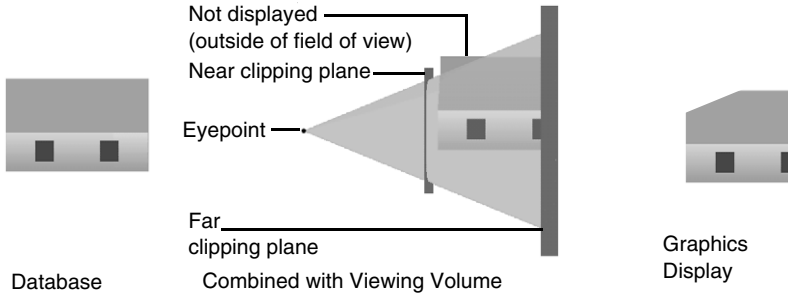
Understanding the Viewing Volume

The *viewing volume* determines how much of the database you see from a given eyepoint. The viewing volume has three components: an angle and two planes.

The angle, called *field of view*, defines a cone of visibility around the viewing vector. The angle's vertex is at the eyepoint. Parts of the database that are outside this cone are hidden.



The two planes that are perpendicular to the viewing vector (and parallel to each other) determine how much of the database is displayed within the field of view. The near clipping plane defines the viewing limit nearest to the eyepoint, while the far clipping plane defines the viewing limit farthest from the eyepoint. Creator displays only the portion of the database that lies between the near and far clipping planes and within the field of view.

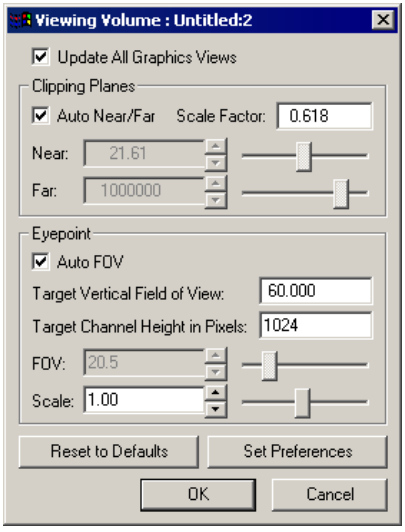


Controlling the Viewing Volume

For a discussion of the viewing volume, see "Understanding the Viewing Volume" on page 263.

You can adjust the viewing volume parameters to control how much of the database is visible in the Graphics view. If you are working with only a subset of what is visible in the Graphics view, then reducing the view to that subset can improve your system performance. This is especially true for large, dense databases. When using z-buffering, it is essential to optimize your clipping plane positions to improve the graphic card's ability to resolve objects that are close together (see "Z-buffer Drawing" on page 686).

Choose **View/Viewing Volume** menu to open the *Viewing Volume* window. In this window, you can set the parameters that control the viewing volume in the Graphics view of the database.



Viewing Volume Parameters

The *Viewing Volume* window has the following parameters:

Update All Graphics Views

Applies the changes made in this dialog box to all open Graphics views in the current Database window. If this checkbox is cleared, the changes you make apply only to the last active view when you chose the **Viewing Volume** function.

Auto Near/Far

Calculates near and far clipping planes based on the distance of the bounding box (that is around the geometry) from the current eyepoint. This is an alternative to defining the near and far clipping planes based on the distances from the eyepoint.

Scale Factor

When **Auto Near/Far** is set, adds a buffer distance to the bounding box. You can increase the default value of the scale factor if the clipping planes are clipping out geometry that you want displayed.

Near

Defines the distance from the eyepoint to the near clipping plane.

Far

Defines the distance from the eyepoint to the far clipping plane.

Auto FOV

Recalculates the field of view based on the **Target Vertical Field of View** and the **Target Channel Height in Pixels** values to simulate the pixel size of the display channel in your runtime system.

Target Vertical Field of View

The size (in degrees) of the vertical field of view in your runtime system.

Target Channel Height in Pixels

The height (in pixels) of the display channel in your runtime system.

FOV

Defines the viewing angle (field of view) of the viewing frustrum from the eyepoint. As field of view is increased, objects in the corresponding Graphics view appear to shrink in size. This is because more of the database is displayed in the same area of the screen.

This is similar to a zoom lens on a camera. A wide field of view is like a wide angle lens, and a narrow field of view is like a telephoto lens.

Scale

Increases or decreases the apparent size of objects within the field of view without changing the eyepoint coordinates.

Reset to Defaults

Restores the default viewing volume parameter values.

Set Preferences

Makes the current viewing volume parameter values the new default.

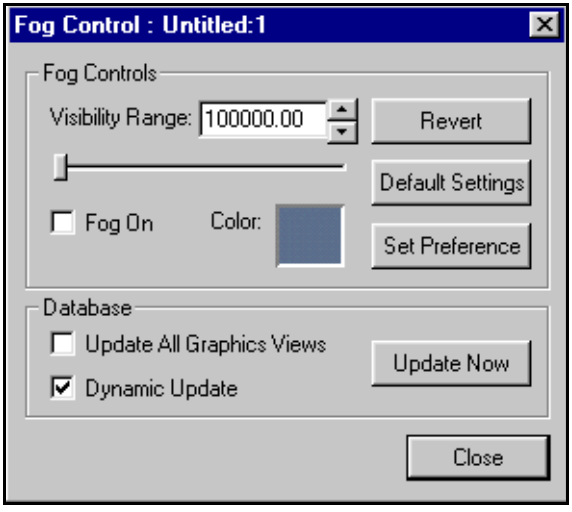
Controlling the Graphics View Appearance

You can control the general appearance of a database by using Creator fog, sky color, and background image controls.

Creating Fog

By turning on fog and setting the fog parameters, you can create the appearance of fog in the Graphics view. As you move the eyepoint through the Graphics view, objects fade in and out to simulate the effects of real fog.

To define the color and density of fog, choose **View/Fog Controls** and set the parameters in the *Fog Control* window.



Fog Control Parameters

The following settings control the appearance of fog.

Visibility Range

Controls the density of fog by setting the distance from the eyepoint at which an item becomes completely obscured. The current distance is shown in the **Visibility Range** text

field. This parameter can be set by entering a value in the text field, using the slider, or clicking the **Up** and **Down** buttons.

Revert

Reloads **Fog Color** and **Visibility Range** settings from your preferences.

Default Settings

Reloads the default Creator **Fog Color** and **Visibility Range** settings.

Set Preference

Saves the current **Fog Color** and **Visibility Range** with your preferences. These are loaded if you click **Revert**.

Fog On

Turns the display of fog on or off.

Color

Displays the current fog color. Clicking this color button opens the *Color Palette* window for editing the fog color. For more information on the *Color Palette* window, see "Color Palette Window" on page 399.

Update Now

Applies any changes to the fog parameter values to all open Graphics views of the database.

Update All Graphics Views

Explicitly applies changes to the fog parameter values to the affected Graphics views.

Dynamic Update

Instantly applies changes to the fog parameter values to the current Graphics views. To reduce drawing overhead on your computer, turn off **Dynamic Update** and use **Update Now** instead.

Close

Closes the *Fog Control* window.

Setting Sky Color

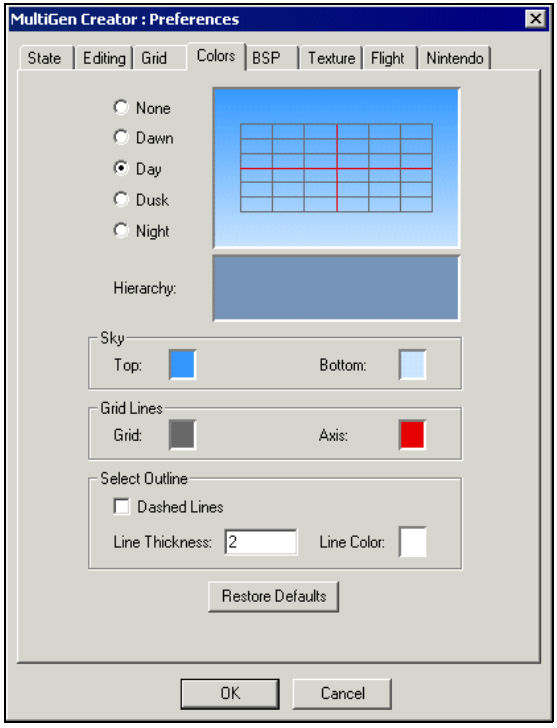
Newly opened database files use the default sky color. The ambient color of the sky in the Graphics view can be set using one of five "time-of-day" choices from the Color Preferences panel: **None, Dawn**, **Day**, **Dusk**, **Night**, and **Time of Day**.

To change a file's sky color, choose **View/Sky Color**, and then select the time of day you want to display from the secondary menu. **None** changes the Graphics view color to a solid color (the default setting is blue); **Time of Day** interpolates current sky color definitions for **Dawn**, **Day**, **Dusk** and **Night** and draws the sky colors according to the current system time during your modeling session.

You can change any of the default sky colors by choosing **View/Sky Color Controls** or **Info/Preferences** and clicking the **Color** tab.

Changing Default Sky Colors

Step 1. Choose **View/Sky Color Controls** to open the Color Preferences panel. You can also open the Color Preferences panel by choosing **Info/Preferences** and clicking the **Color** tab.



Step 2. Click the time-of-day sky color you want to edit. It appears in the preview window.

Step 3. Click the **Top** color sample to open the Color palette. Choose a new color, and click **OK** to close the Color palette. the preview window updates with the new **Top** color.

Step 4. Repeat Step 3 to change the **Bottom** color.

Step 5. Click **OK** to accept the new sky colors, or click Restore Defaults to reset the colors to their original default settings.

Colors Preferences (Sky Colors)

The following settings in the Colors preference panel affect sky color.

None, Dawn, Day, Dusk, Night, Time of Day

Shows which time-of-day sky color definition is currently selected for editing. **None** displays the sky color in a solid color. **Time of Day** takes the current definitions for **Dawn**, **Day**, **Dusk** and **Night** and draws the sky colors according to the user's current system time during a modeling session. **Dawn**, **Day**, **Dusk** and **Night** blend the Top and Bottom color definitions.

Bottom

Defines the sky color at the bottom of the Graphics view.

Top

Defines the sky color at the top of the Graphics view.

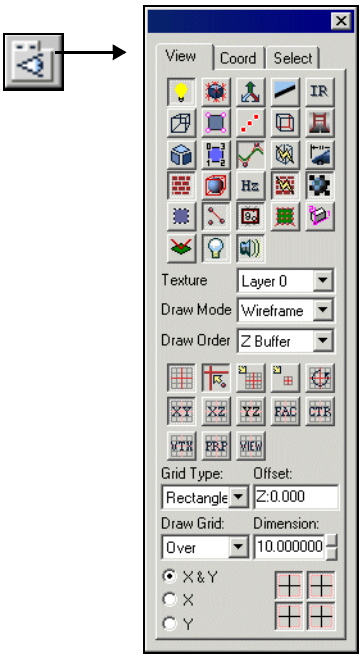
Restore Defaults

Resets all Colors preference definitions to their original default colors.

Drawing Controls



Drawing controls determine the appearance of the Graphics view. Some drawing controls are in the **View** menu, but most are located in the View panel. To open the View panel, click the **View panel** button in the Toolbar.



Hidden Faces

Hidden faces are faces in which the **Hidden** face attribute is enabled. The **View/Hidden Faces** submenu contains options that control the display of hidden faces.

Hidden Faces Menu Options

The following options are available in the **View/Hidden Faces** submenu:

Hide

Suppresses the display of hidden faces in the Graphics view.

Draw Normal

Displays hidden faces in the Graphics view.

Draw Wireframe

Displays hidden faces in wireframe regardless of the Graphics view drawing mode (see "Draw Mode" on page 321).

Changing the Database View

Any pane in the Database window can be changed to show a Graphics view. In addition, a large single-window containing the Graphics view is also available for overview or screen shot purposes.

View/Graphics

To change the view in a pane to the Graphics view, activate the pane and choose **View/Graphics**. The Graphics view is the default view in a newly opened database file. You can toggle between the Graphics and Hierarchy views by pressing S.

Select/Isolate

To work more efficiently, you can open an isolated view of selected items in your database. This allows you to more easily focus on a relevant part of your database.

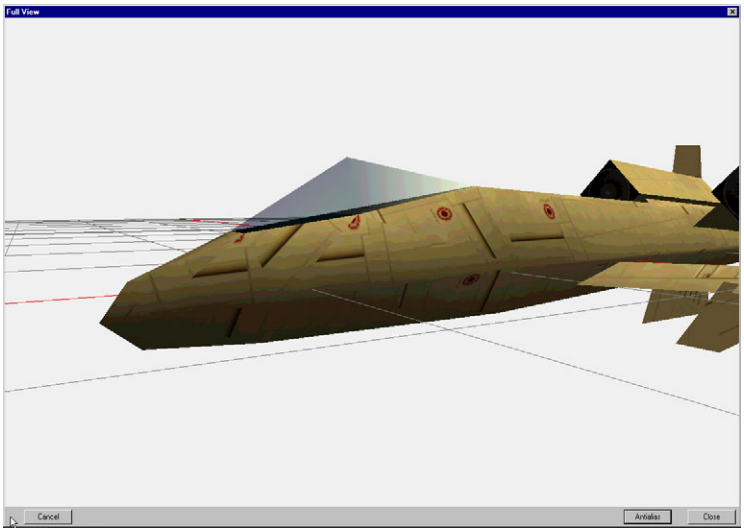
To open an isolated view, select one or more items in your database and then choose **Select/Isolate** (Ctrl+I). For complete information about this feature, see "Isolate" on page 332.

View/Full View

Choosing **View/Full View** fills the entire desktop with the Graphics view of the database. This is often done prior to capturing screen shots of a database. To produce an antialiased image, click the **Antialias** button in the lower-right corner of the screen. (It may take some time to compute.) You can change the eyepoint in full view mode with the right mouse button (see "Adjusting The Eyepoint" on page 251).

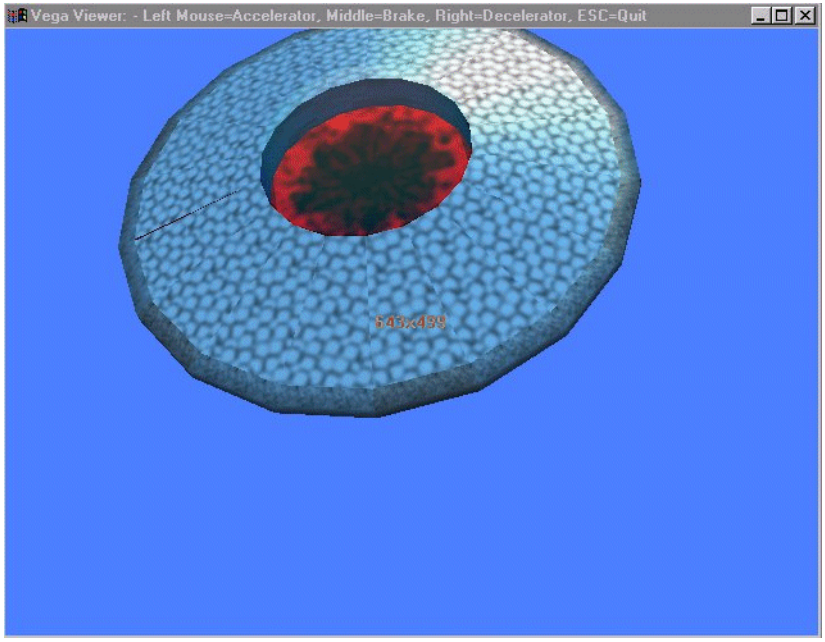
To return from Full View mode to regular editing mode, click **Close** in the lower right corner of the screen.

Full View mode



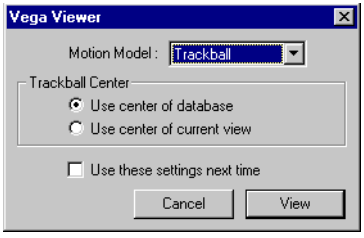
Using Vega Viewer

Vega Viewer lets you quickly preview your database simulation to check its performance. It plays the Vega .adf file that is written when you save your OpenFlight file. The .adf file specifies the initial conditions for a Vega application. Vega Viewer is automatically installed when you choose to install Creator using the default installation.



To use Vega Viewer:

Step 1. Choose **View/Vega Viewer**. The *Vega Viewer* dialog box opens.



Step 2. Set the type of motion you want to use. If your motion mode is **Trackball**, set the location of the trackball center, and then click **View**.

Step 3. The *Vega Viewer* window opens with your current database displayed. You control the motion using mouse buttons. Depending upon the **Motion Model** setting, the mouse buttons have different actions. The actions are displayed in the window's title bar.

Step 4. When you are done viewing your model, press ESC to quit Vega Viewer.

Note: If you later make changes to your model in Creator, you must first save your model in Creator before you can see the changes in Vega Viewer.

Vega Viewer Parameters

Motion Model

Sets the way you want your model to move. For example, for a moving car or tank, you would choose Drive. Options are:

- Trackball - yaw, pitch, roll, zoom in/out
- Drive - ground movement
- UFO - Similar to Fly, but with hovering capabilities
- Fly - High altitude flight simulation

Trackball Center

Sets the center of motion for the trackball. Use Center of Database sets the center of the trackball motion at 0, 0, 0. Use Center of View sets the center of the trackball motion at the center of the current view.

Use These Settings Next Time

Preserves the options you have set.

Using Interceptor 3D Viewer

To be completed. The Interceptor 3D Viewer will have its own online help system.

7 Working in the Hierarchy View

This information supplements the chapter *Structuring a Database* in the *MultiGen Creator Modeler's Guide*.

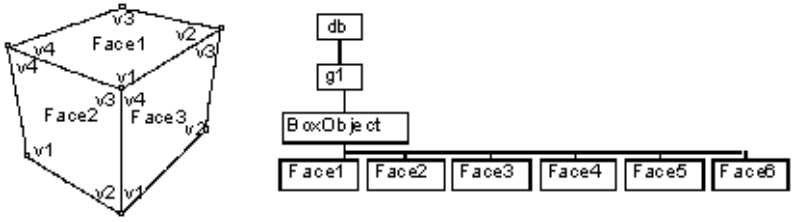
The OpenFlight format uses a multilevel hierarchy to define the relationships between items in the database. The *Graphics* view provides a three-dimensional representation of the database components so that you can work with models in a more intuitive way. The *Hierarchy* view lets you see and manipulate the organizational structure of these components.

Models used in realtime must be defined in terms not only of geometry, but also of structure, or *hierarchy*. The Hierarchy view is where you see the organized components of your database. The Hierarchy view appears as levels of nodes, connected in one or more tree structures.

To understand database hierarchy, you should be familiar with the following terms.

- *Node* - A database item that appears as a labeled box in the Hierarchy view. Some common types of nodes are: group, object, face, and vertex.
- *Bead* - Obsolete synonym for node.
- *Parent* - The point of attachment of any given node, or the node attached just above the given node.

  In the figure below, **BoxObject** is the parent of **Faces 1-6**, and each face is the parent of four vertices (not shown in the hierarchy).



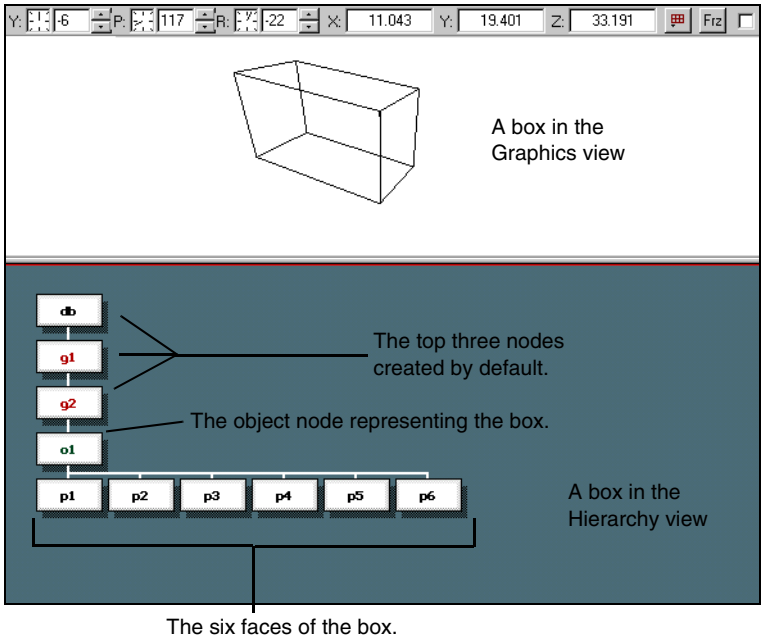
Graphics and Hierarchy of a Box

- *Child* A node attached below another node in the database tree. In the figure above, **Faces 1** through **6** are the children of **BoxObject**.

- *Siblings* - Two or more nodes attached to the same parent. In the figure above, **Faces 1** through **6** are siblings.
- *Descendant* - A node that is reached by traveling down a series of parent-child relationships.
- *Ancestor* - A node that is reached by traveling up through a series of child-parent relationships.
- *Database tree* - The entire hierarchy of the database.
- *Subtree* - Any node, together with all of its descendants.
- *Database header* - The root or topmost node in the database hierarchy. The database header is created automatically with each new file and is represented by the node labeled **db** in the Hierarchy view.
- *Level* - The number of parent-child relationships between the database header and each node. A child is one level lower than its parent. Typically, it is the relative depth of nodes, rather than their specific levels, that is of interest.
- *Orphan* - A node that has no point of attachment in the hierarchy, but may have descendants. *Orphans are not saved with the database.*

New database files open with a default hierarchy consisting of three nodes labeled **db**, **g1**, and **g2**. **g1** and **g2** are group nodes, and can be renamed or deleted. **db**, the database header, always exists and cannot be renamed. For more information on these node types, see "Database Node Types" on page 292.

New models you create in the Graphics view appear as nodes in the Hierarchy view, and are automatically attached to the three original nodes. For example, if you build a box in the Graphics view, new levels of nodes appear in the Hierarchy view to represent that box.



To see a pane containing the Hierarchy view after you first open a database, choose **Window/Standard View** (Shift+F9) or drag the bottom split bar up.

To change the Graphics view in an active pane to the Hierarchy view, choose **View/Hierarchy**.

The following topics show you how to use tools and controls to create efficient databases, how to modify the database hierarchy, and how to control its appearance. This information supplements the descriptions of each node type in “OpenFlight Nodes and Attributes” on page 825.

Expanding and Collapsing Hierarchy Levels

The Hierarchy view can be controlled to display as many or as few levels as are needed using the **Expand Hierarchy** and **Contract Hierarchy** tools. This allows you to work with as much of the hierarchy as you need to see. You can also quickly maneuver the Hierarchy view into a more convenient position.

Note: Alternatively, you can click with the middle mouse button to expand the hierarchy or hold down Alt and click with the middle mouse button to contract the hierarchy.

To Expand Specific Nodes

Step 1. Select the node or nodes you want to expand.



Step 2. Open the Hierarchy toolbox and click **Expand Hierarchy** to expand the selected nodes.

Step 3. To expand the entire hierarchy below the selected node, hold down Ctrl and click **Expand Hierarchy**, or continue clicking the middle mouse button to expand the selected nodes' children until you have reached the desired level.

To Expand The Database By Levels

Step 1. Make sure that no nodes are selected.



Step 2. Click **Expand Hierarchy** in the **Hierarchy** toolbox to expand the database by one level. Every node that can expand does so, simultaneously.

Step 3. To expand the entire hierarchy, hold down Ctrl and middle click anywhere in the Hierarchy view, or continue to click **Expand Hierarchy** until you have reached the desired level.

To Collapse Specific Nodes

Step 1. Select the node or nodes you want to collapse.



Step 2. Click **Contract Hierarchy** in the **Hierarchy** toolbox to collapse the selected nodes.

Step 3. To contract the entire hierarchy up to the selected node, hold down Ctrl while clicking **Contract Hierarchy**, or continue to click **Contract Hierarchy** to collapse the selected nodes' children until you have reached the desired level.

To Collapse The Database By Levels

Step 1. Make sure that no nodes are selected.



Step 2. Click **Contract Hierarchy** in the **Hierarchy** toolbox to collapse the database by one level. Every node that can collapse does so, simultaneously.

Step 3. To contract the entire hierarchy, hold down Alt while middle clicking anywhere in the Hierarchy view, or continue to click **Contract Hierarchy** until you have reached the desired level.

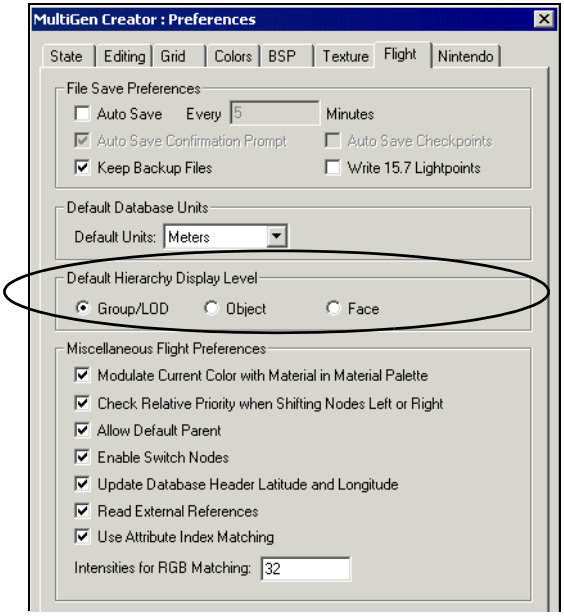
Resetting the View

You can instantly expand or collapse the database tree to a default level by choosing **View/Reset** (U).

To Set The Default Reset Level

Step 1. Choose **Info/Preferences** (Ctrl+Shift+P) to open the *Preferences* window.

Step 2. Click the **Flight** tab to go to the Flight panel.



Step 3. Select one of the levels shown in the **Default Hierarchy Display Level** group.

Changing the Hierarchy View Color

You can change the Hierarchy view's background color in the **Info/Preferences/Color** panel.

Step 1. Choose **Info/Preferences**. The *Preferences* window opens.

Step 2. Click the **Color** tab to display Color preferences.

Step 3. Click the **Hierarchy** color sample to open the Color palette. Choose a new color, and click **OK**.

Step 4. Click **OK** to close the *Preferences* window.

Selecting Nodes

To select a node, click it or drag a fence around at least part of the node.

To select a number of nodes, hold down the Shift key and click each node individually, or drag a fence around the nodes you want to select.

Note: You cannot select both a parent node and its children, since selecting the parent implies that the children are selected as well. If you drag a fence around a parent and its children, only the parent will appear to be selected, and an error message will appear.

Selecting nodes using the fence can unintentionally initiate drag-and-drop editing (see "Drag-and-Drop Editing" on page 287). This happens when the fence is started with the pointer directly over a node. Avoid this situation by holding down the Shift key before dragging a fence with the pointer directly over a node.

Viewing Selected Nodes

To center a selected node in your Hierarchy view, choose **View/Center Select** (C). This is useful for moving to a chosen node in a database that is too large to fit in a pane.

For example, if you create a sphere and expand the Hierarchy view to show all of the sphere's polygons, or faces, it is likely that many of the faces will be out of view. To go to a particular face's node:

Step 1. Select the face in the Graphics view.

Step 2. Press the Tab key until the Hierarchy view pane is active.



Step 3. Choose **View/Center Select** (C). The node appears in the center of the pane.

This method can be used to center any type of node or groups of nodes.

Zooming in the Hierarchy View

Zooming in the Hierarchy view is similar to zooming in the Graphics view. Using **Zoom View**, you can zoom in and out to see more or less of the database in the Hierarchy view. When **Zoom View** is active, the pointer turns into a magnifying glass containing a plus sign (zoom in) or a minus sign (zoom out).

To use **Zoom View**:



Step 1. Click **Zoom View** in the Toolbar or simultaneously hold down the Shift key and the right mouse button. The pointer turns into a magnifying glass.

Step 2. Click in the Hierarchy view with the right mouse button. This gives input focus to that view.

Step 3. Drag the magnifying glass down to zoom in, or drag it up to zoom out.

Note: Holding down the middle mouse button while zooming makes the zoom action continuous, making it unnecessary to continue moving the magnifying glass.

Panning in the Hierarchy View

Using **Pan View**, you can pan up, down, left, and right to bring other parts of the database into view. When **Pan View** is active, the pointer turns into a cross with arrows.

To use **Pan View**:



Step 1. Click **Pan View** in the Toolbar or simultaneously hold down the Ctrl key and the right mouse button. The pointer turns into a cross with arrows.

Step 2. Drag the cross in the direction you want to go.

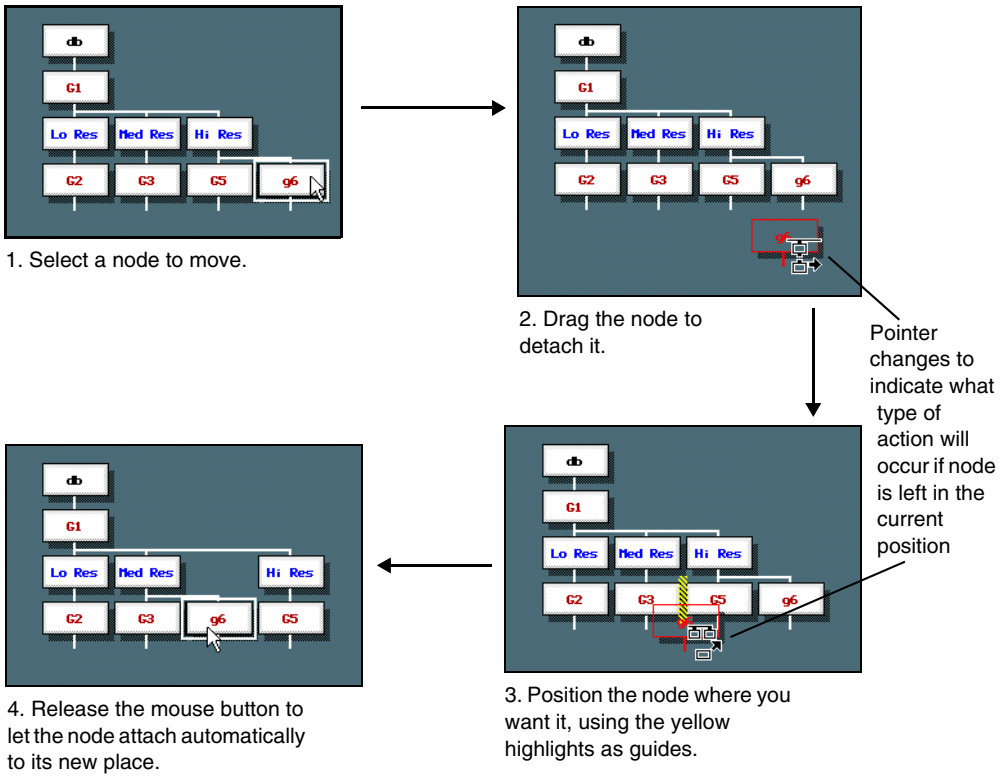
Note: Holding down the middle mouse button while panning makes the pan action continuous, making it unnecessary to continue moving the cross.

Editing the Hierarchy Structure

You can change the hierarchical relationships in your database by editing the database in the Hierarchy view. Individual nodes, groups of nodes, subtrees, and levels can be manipulated. Creator has two graphical methods for editing the database hierarchy's structure: drag-and-drop editing, and hierarchy tools. These methods are discussed in the next two sections.

Drag-and-Drop Editing

To change the point of attachment of any node in the Hierarchy view, simply *drag* it to a new position and *drop* it by releasing the mouse button. This is referred to as *drag-and-drop editing*. As you drag, the pointer changes to show the effect of dropping the node and a yellow highlight shows the potential point of attachment. If you drop the node at an illegal point of attachment, the node returns to its original position.



If more than one Hierarchy view of the database is open, you can drag and drop between views by holding down both the left and right mouse buttons as you cross the boundary between panes. This is useful when the original point of attachment and the new point of attachment are far apart in the hierarchy.

Priority and Shifting Nodes

The hierarchy tools used to shift nodes left or right can be restricted by the Flight preference **Check Relative Priority When Shifting Nodes Left or Right**. When this preference is set, Creator checks the value of each shifted node's **Relative Priority (Fix Listing)** attribute. A node cannot be put in a position that allows it to be processed ahead of nodes with a higher relative priority value.

For more information on relative priority and database traversal (the processing of nodes), see "Draw Order" on page 682 and "Database Traversal Order" on page 688

Database Identification Names

Database identification names, also called *Database identifiers (IDs)* or node IDs, are alphanumeric names (up to 512 characters) that uniquely identify each node in the database. OpenFlight IDs are case sensitive. Default IDs are assigned automatically for each node created in the database. Each node is named with a letter indicating the node type followed by a sequential number. Examples of default IDs are **g5** (group 5), **l3** (level of detail 3), **o121** (object 121), **p223** (polygon 223), and **v328** (vertex 328).

The first seven characters of each node ID appear in the box that represents the node in the Hierarchy view. The full name appears in the editable **ID** attribute of each node in the database. The full identifier of each node also appears in the Select list when the node is selected for processing.

Renaming nodes can make objects and groups easier to identify in the Hierarchy view. Some examples of modeler-assigned IDs are:

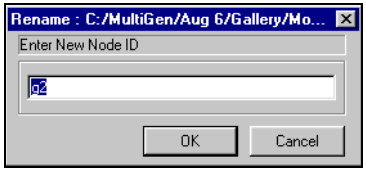
- **HouseGrp**, a group node whose subtree contains all the nodes that make up a model of a house.
- **RoofObj**, an object node whose subtree contains all of the nodes that make up a model of a house's roof. **RoofObj** could be a child of **HouseGrp**.
- **GrndObj**, an object node whose descendents make up the ground in a model.

For more information on types of nodes, see "Database Node Types" on page 292.

To Rename a Node

Step 1. Select the node to rename.

Step 2. Choose **Attributes/Rename** (Ctrl+J) to open the *Rename* dialog box. This dialog box has a text field containing the current ID of the node.



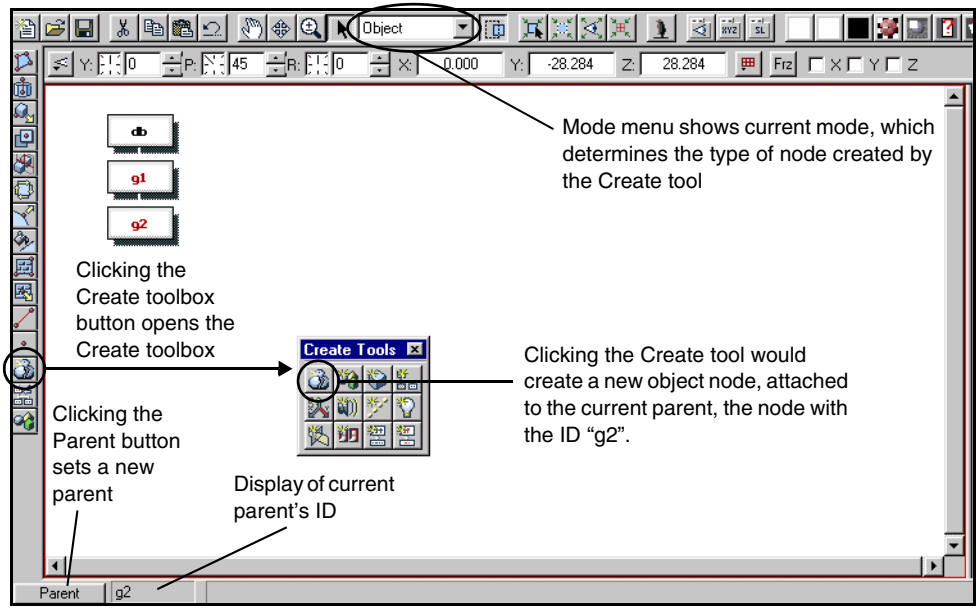
Step 3. Enter a new ID for the node and click **OK**.

Node IDs can also be renamed in the node's attribute window. For more information about node attributes, see "OpenFlight Nodes and Attributes" on page 825.

Database Parent



The database parent (or simply parent) is the current point of attachment in the database hierarchy. Many modeling functions require you to set a parent. For example, you can use the **Create** (Shift+C) tool to create certain types of nodes which are automatically attached to the current parent.



Setting the Parent

The ID of the current parent is displayed in the field next to the **Parent** button, near the lower-left corner of the application window. To set a new parent, do one of the following:

- Select a node in the Hierarchy view or an item in the Graphics view and click the **Parent** button.
- Press the Alt key and click a node or an item in the Graphics view. In the Hierarchy view, the node you click becomes the parent without being selected. In the Graphics view, the item selected when you click depends on the current modeling mode. The selected item becomes the database parent.

To clear the database parent, press Ctrl and click the **Parent** button.

Note: If you attempt to set a database parent or create a new node but fail, check the Status bar for an error message. You may have set an invalid parent.

Default Parenting

In automatic parenting mode, Creator uses a default group node as the database parent during any operation that requires a parent, but for which the database parent specified is not valid. Default parenting is enabled by setting the **Allow Default Parent** Flight preference.

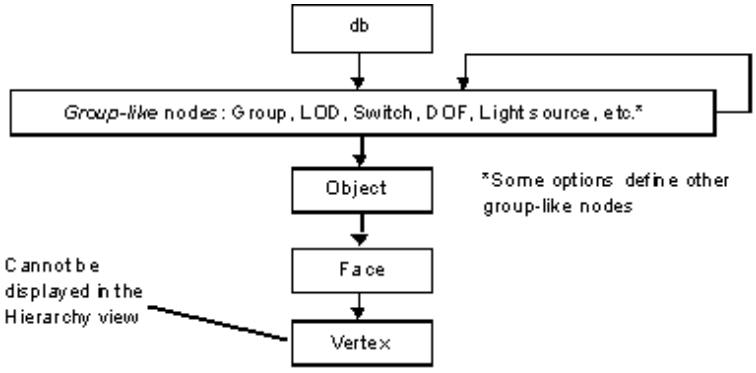
The first time a default parent is required, Creator creates a group, usually named **Default**, and attaches it to the database header. This node continues to be the default parent, even if you move it to another point in the hierarchy. If you delete the default parent node, it is recreated the next time it is needed.

If you clear the **Allow Default Parent** Flight preference, automatic parenting is disabled. If you try to create a node when a parent node is not set, Creator displays an error message and does not create the node.

If the ID **Default** is already in use when the default parent node is created, an arbitrary group ID is used as its identifier.

Database Node Types

A database hierarchy's levels consist of one or more nodes. At the top level is a unique node called the *database header*. At the lowest level are objects made up of faces, which are, in turn, made up of vertices. Between these two levels are a number of different types of group-like nodes that are attached to each other in an organized way.



A general database hierarchy consists of several types of nodes, organized in layers

Each node has a set of descriptive data (options, settings, flags, numeric values, and text) that are saved with the node in the database hierarchy. This data is referred to as the node's *Database attributes*. Each type of node (group, object, face, etc.) can have a different set of Database attributes, although some attributes are common to more than one node type. For example, the node ID is an attribute common to all nodes.

Database Header

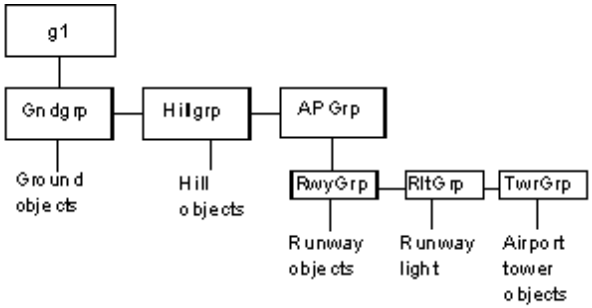
The *Database Header* node is at the top of the database hierarchy. It is created automatically for each new file and is labeled **db** in the Hierarchy view. Information about the database, such as database units, creation and modification dates, and terrain projection information are stored in the database header. The database header cannot be deleted or explicitly selected.

For a discussion of header attributes, see "Header Attributes" on page 831.

Groups

Collecting sets of logically-connected objects into groups makes them easier to manipulate in the database. Group nodes serve as the point of attachment for such logically organized objects, and can even include other group nodes. Group node IDs are color-coded red in the Hierarchy view. Default group node IDs always start with **g**.

In the figure below, the top-level group (**g1**) contains one group each for the ground (**Gndgrp**), some hills (**Hillgrp**), and an airport (**APGrp**). The airport group is further divided into groups representing the runway (**RwyGrp**), runway lights (**RltGrp**), and airport tower (**TwrGrp**).



Using Groups to Organize Hierarchy

Imagine you want to move the tower. With the organization as it is, you only need to select **TwrGrp** to select all the geometry defining the tower. If the runway objects, runway light objects, and tower objects were attached directly to **APgrp**, you might find it difficult to select the tower objects.

For a discussion of group attributes, see “Group Attributes” on page 835.

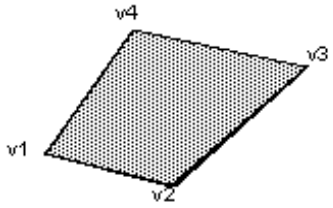
Objects

An *object* is a collection of polygons. Objects created in the Graphics view appear as object nodes in the Hierarchy view. Object IDs are color-coded green in the Hierarchy view. Default object IDs always start with **o**.

For a discussion of object attributes, see “Object Attributes” on page 839.

Faces

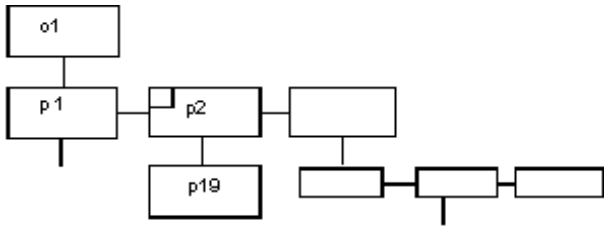
A *polygon,* or *face*, is a collection of ordered, coplanar vertices describing a surface. Faces created in the Graphics view appear as face nodes in the Hierarchy view. A face node’s color reflects the color assigned to the face in the Graphics view, and its ID color is either black or white, depending on which is more visible. Default face node IDs always start with **p**.



A face, with its vertices displayed

A *subface* (or *nested face*) is a face attached to another face in the hierarchy. Subfaces can also be the parents of subfaces. Each subface should lie within, and on the same plane as, its parent. Subfaces provide a useful means of specifying drawing order on z-buffered systems, because in z-buffered drawing mode, every subface draws on top of its parent.

A face node with a subface for a child has a small square in the upper-left corner of the rectangle that represents it in the Hierarchy view.

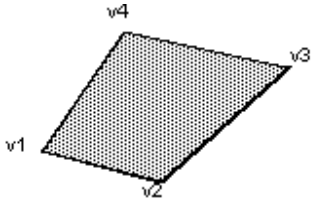


p19 is a subface of p2

For a discussion of face attributes, see "Face Attributes" on page 840.

Edges

An *edge* is the line segment between two consecutive vertices in the same face. Edges are not defined as part of the database hierarchy, and so do not appear as nodes in the Hierarchy view. However, several edge functions are provided to facilitate modeling in the Graphics view. See "Edge Tools" on page 209.



A face with four edges, each defined by two vertices

Vertices

Vertices are coordinate points in the database. The unit of the database coordinate system is determined by the **Database Units** attribute of the database header node. Vertices are created during a number of Creator edit functions, usually as part of the

process of creating a face. Creator also uses construction (reference) vertices that are not saved with the database. See “Edge Tools” on page 209 and “Vertex Tools” on page 220.

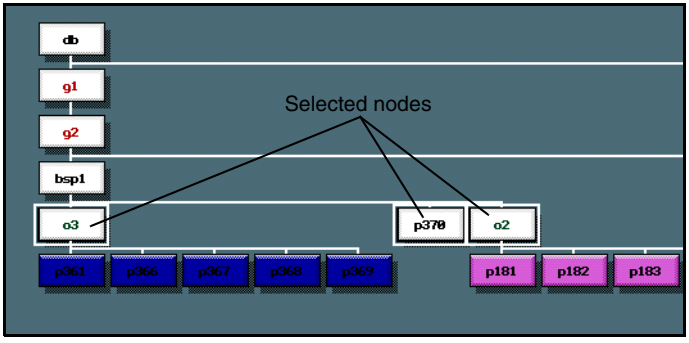
For a discussion of vertex attributes, see “Vertex Attributes” on page 848.

Saving Selected Nodes

If you want to save part of a database into a new database file, you can use the **Write Select** feature. The highest-level selected node or nodes are each saved in a separate file, and each node’s subtree is saved with it.

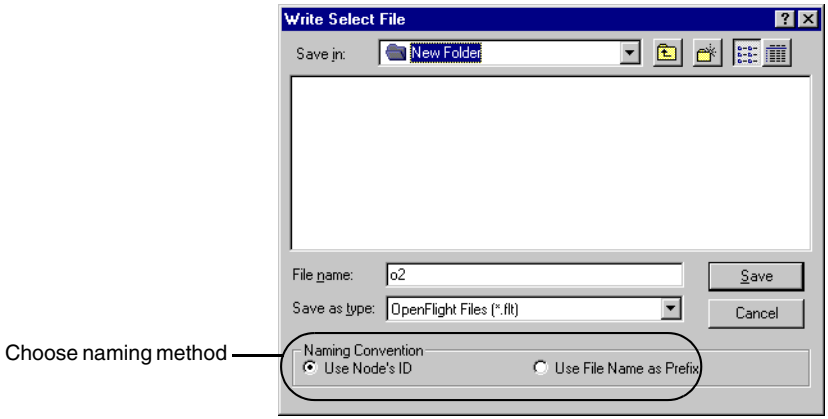
To Save With Write Select

Step 1. Select a portion of the database to save, either in the Hierarchy view or the Graphics view.



Selected nodes’ descendents, such as P181, are saved with their parents

Step 2. Choose **File/Write Select** to open the *Write Select File* dialog box.

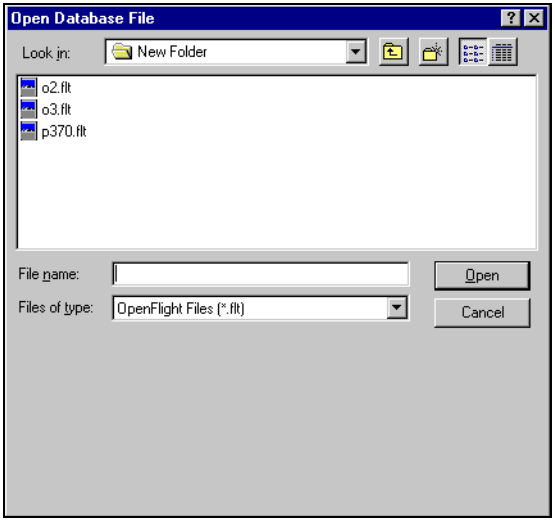


Step 3. Choose the folder to save the new files in.

Step 4. Click **Use Node's ID** to name the new files with the highest-level node ID saved in that file, or click **Use File Name as Prefix** to use the name you enter in the **File name** text field. With **Use File Name as Prefix**, an incremented number is automatically appended to the file name prefix as each file is saved.

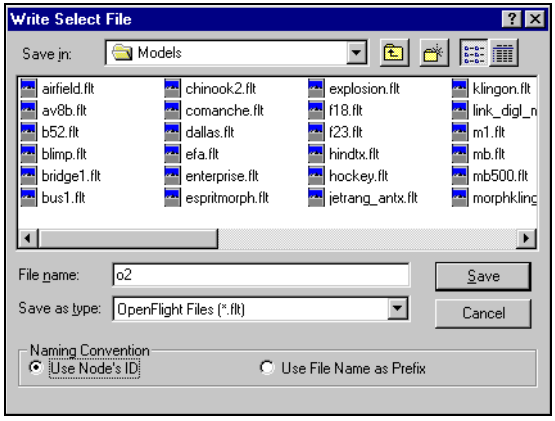
Step 5. Click **Save** to save the new files. Once they are saved, the new files can be opened as standard OpenFlight databases.

In this case, three files were saved using the **Use Node's ID** naming method: **02.flt**, **o3.flt**, and **p370.flt**.



Write Select Naming Conventions

Write Select names the nodes being written out in one of two ways: by using node IDs, or by using a custom prefix. You can set the naming convention in the *Write Select* dialog box.



Use Node's ID

Saves each selected node in a database file named using the node identifier, plus the extension .flt. For example, if **o23** and **o24** are selected, the new databases are called o23.flt and o24.flt.

Use File Name as Prefix

Saves each selected node in a database file using the name entered in the **File name** field, plus a counter. For example, if two nodes are selected and the selection field contains the string **newdb.flt**, the new databases are called newdb1.flt and newdb2.flt.

Designing An Efficient Database

The Hierarchy view of a database shows the structure formed by the interrelationships of the nodes that make up the database. The efficiency of a database's structure can have a great impact on how well the database performs in a realtime system. A database structure can be rearranged as needed, but this becomes more difficult as the database grows in size. The structure of the database should be worked out at the beginning of a modeling project.

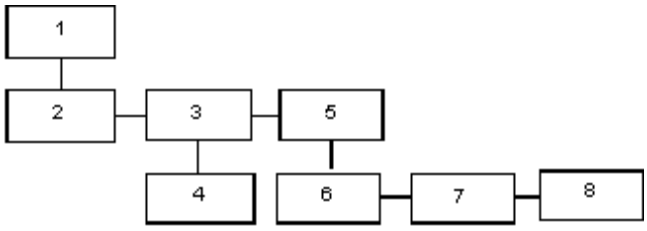
The database structure is important because each time the database is used in a realtime system, it will be traversed in a particular order. In an efficient database, the needed items will be found quickly; a minimum amount of time will be spent unnecessarily parsing subtrees.

For more information on optimizing a database for a realtime system, see "Realtime Considerations" on page 681.

Database Traversal

Database traversal refers to the order in which nodes in the database hierarchy are processed for drawing. Some functions, such as fixed list drawing and the rendering of face transparency, use the order of nodes in the database traversal to determine the final appearance of the Graphics view.

In OpenFlight, the database hierarchy is traversed from top to bottom and then from left to right. For more information on database traversal, see "Database Traversal Order" on page 688.



A sample database hierarchy, showing the order of traversal

Object and face nodes can also have a **Relative priority** attribute. This attribute is a value that fixes the hierarchical position of each node relative to its siblings. For more information on **Relative priority**, see "Relative Priority" on page 688.

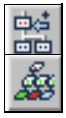
Database Hierarchy Reorganization

Many realtime systems are able to improve drawing performance by *culling* (ignoring) groups and objects that are not visible from the current eyepoint. To optimize realtime culling from a large database, you can organize each level of detail into a grid of evenly distributed groups.

The **Reorganize Hierarchy** tool in the **Hierarchy Tools** toolbox allows you to rebuild the hierarchy attached to all levels of detail in a database to optimize for realtime culling.

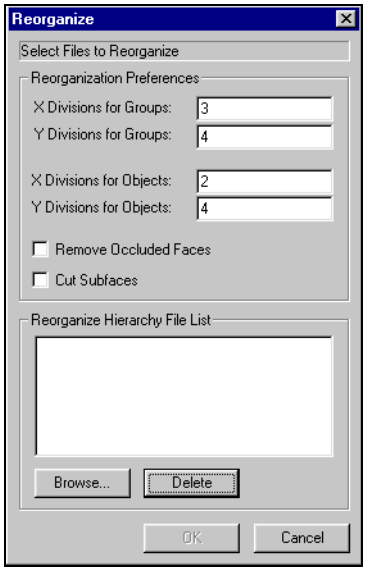
The **Reorganize Hierarchy** tool discards the groups and objects below each level of detail in the selected database and collects the orphan faces into grids of evenly distributed objects and groups.

To Reorganize a Database Hierarchy



Step 1. Open the **Hierarchy Tools** toolbox.

Step 2. Click **Reorganize Hierarchy**.



Step 3. In the *Reorganize* dialog box, enter the number of groups per level of detail in the **X Divisions for Groups** and **Y Divisions for Groups** fields.

Step 4. Enter the number of objects per group in the **X Divisions for Objects** and **Y Divisions for Objects** fields.

Step 5. Select the database files to reorganize with the **Browse** button.

Step 6. Click **Done** to reorganize the selected database files.

Reorganize Parameters

These parameters let you control how the databases are reorganized.

X Divisions for Groups

Defines the number of groups to create along the *x* axis for each level of detail.

Y Divisions for Groups

Defines the number of groups to create along the *y* axis for each level of detail.

X Divisions for Objects

Defines the number of objects to create along the *x* axis of each group.

Y Divisions for Objects

Defines the number of objects to create along the *y* axis of each group.

Remove Occluded Faces

Deletes terrain faces that are entirely covered by non-terrain faces from the final database.

Cut Subfaces

Merges subfaces into their parent faces by removing the area under the subface and triangulating the remaining area of the face.

Reorganize Hierarchy File List

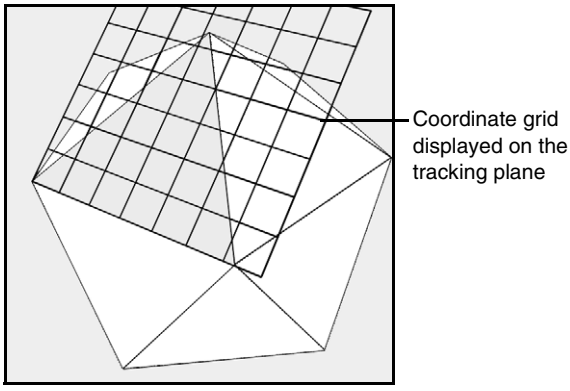
Displays a list of database files to process. **Browse** adds files to the list. **Delete** removes files from the list.

8 The Tracking Plane and Grid

In Creator, *z* is a vertical axis and *x* and *y* are the horizontal and vertical axes that are perpendicular to *z*. When you edit the database, Creator projects points entered with the mouse into this three-dimensional coordinate system and places them on the *tracking plane*. Your computer display's surface is two-dimensional; you use the tracking plane to accurately place database components in three dimensions.

The tracking plane has two components:

- A plane, defined anywhere within the three-dimensional coordinate space of the database. Ordinarily, coordinates entered with the left mouse button are automatically placed on the tracking plane. (This is not true of coordinates entered with the middle mouse button. See "Referencing a Vertex or Edge" on page 355.)
- A two-dimensional grid, located on the tracking plane. The grid can be defined in Cartesian or radial coordinates and can be visible or invisible (see "Grid and Trackplane Controls" on page 306). The grid can be in the XY, XZ, or YZ tracking plane.



Defining the Grid

The grid shows distance increments to assist you when entering coordinates and allows you to see the current position of the tracking plane. Each Graphics view of the

database has its own tracking plane and grid. Grid controls are located in the View panel.

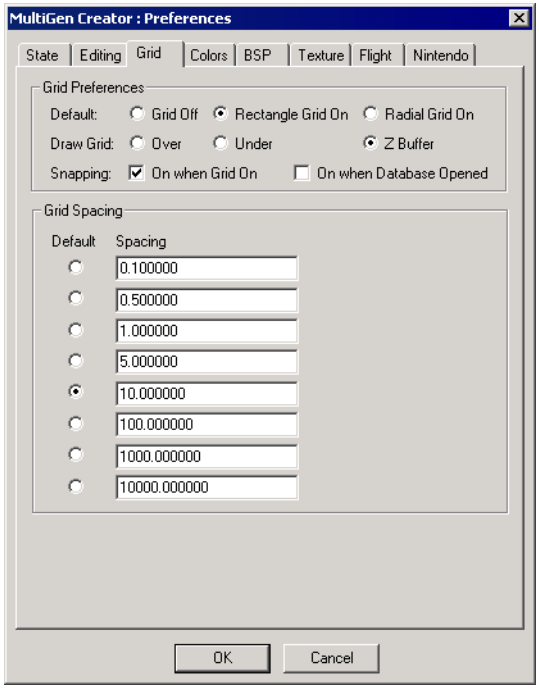
The tracking plane is always defined, whether or not you see the grid.

Setting Grid Preferences

Default grid settings are located in the **Grid** Preferences panel. To display these preferences, choose **Info/Preferences** (Ctrl+Shift+P), and select the **Grid** tab.

Grid Preferences

Grid preferences set the default grid characteristics for each new database.



Default

Defines the type of grid to be displayed on the tracking plane.

- *Grid Off* does not display a grid
- *Square Grid On* displays a grid defined in Cartesian coordinates
- *Radial Grid On* displays a grid defined in radial coordinates

Drawing

Controls the default way the grid is drawn, as follows:

- **Grid Over** draws the grid in front of the database scene
- **Grid Under** draws the grid behind the database scene
- **Grid Z-Buffer** draws the grid Z-buffered with the database scene and puts the Graphics view in Z-buffer drawing mode

Snapping

Controls grid snap. When grid snap is on, any point you place in the Graphics view is automatically "snapped" to the nearest grid intersection.

- **On when Grid On** links grid snap to the **Grid On/Off** option in the View panel (see "Grid and Trackplane Controls" on page 306). If **On when Grid On** is set, grid snap is on only when the grid display is on.

  You can also set and clear grid snap manually using the **Grid Snap** control in the View panel.
- **On when Database Opened** controls the default setting of grid snap. If this option is set, grid snap is turned on each time you open a database. You can then turn it off and on using the controls in the View panel.

Grid Spacing

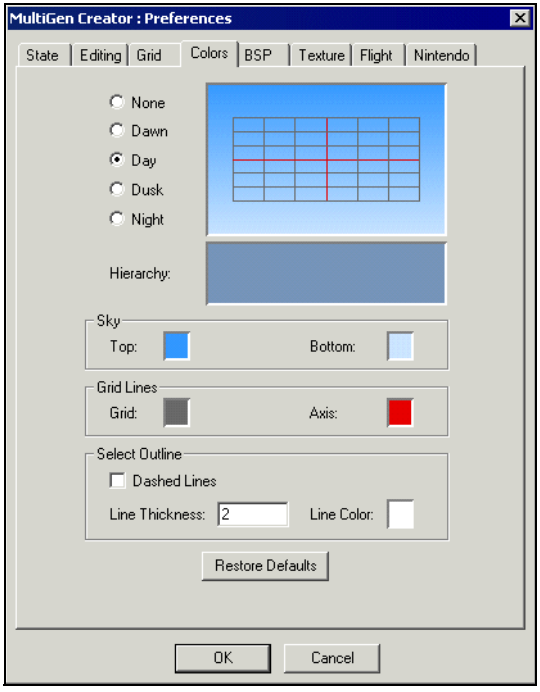
Controls the distance between grid lines. The unit of measurement is determined by the **Default Spacing** defined in the Flight preferences panel.

Setting Default Grid Colors

You can set the default colors for the grid and the axis lines in the Colors preference panel.

Step 1. Choose **Info/Preferences** to open the *Preferences* window.

Step 2. Click the **Color** tab to display the Colors panel. The panel shows a miniature sample of the Graphics and Hierarchy views in their current color schemes.



Step 3. In the Lines section, click the **Grid** color sample to open the Color palette.

Step 4. Choose a new grid color, and click **OK**.

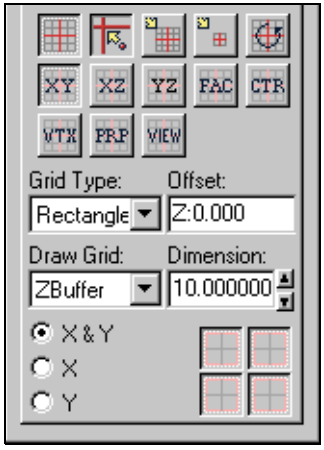
Step 5. Repeat Steps 3 and 4 to change the color of the axis lines, if you wish.

Step 6. Click OK to update the Preferences with the new colors, or click **Restore Defaults** to reset the colors to their original defaults

Grid and Trackplane Controls



The View panel contains grid controls that manipulate the grid for the current database. To open the View panel click **View Panel** in the Toolbar or choose **View/Grid Controls** from the **View** menu.



Grid On/Off (G)



Grid On/Off controls the display of the grid on the tracking plane. The **Grid Off**, **Rectangle**, and **Radial** options in the Grid preferences control the default appearance of the grid. See "Defining the Grid" on page 303.

Grid Snap



Grid Snap sets and clears the grid snap function. When **Grid Snap** is on, coordinates entered with the left mouse button automatically snap to the closest point of intersection among grid lines. This does not affect coordinates entered with edge or vertex references. (See "Referencing a Vertex or Edge" on page 355.) The default grid snap features are set by the **Snapping** preference in the Grid preferences.

Note: Grid Snap can be turned on even when grid display is turned off.

Grid Larger



Grid Larger doubles the size of the grid along each axis. To reduce drawing time on your computer, it is a good idea to limit the grid display to a small area of interest.

Grid Smaller

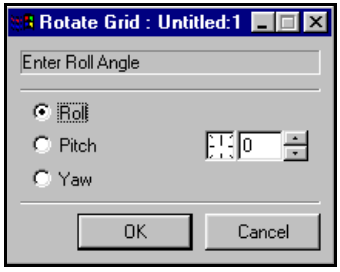


Grid Smaller reduces the size of the grid by half along each axis. Hold down the Ctrl key and click **Grid Smaller** to reduce the grid to its smallest size.

Rotate Grid



Rotate Grid changes the orientation of the grid and tracking plane. In the *Rotate Grid* dialog box, **Roll** rotates the tracking plane about its own y-axis, **Pitch** rotates the tracking plane about its own x-axis, and **Yaw** rotates the grid about the z-axis without changing tracking plane definition. To change the grid, edit the value in the text field, drag the hands of the clock, or drag the grid to rotate it in the Graphics view.



Coordinate Plane Buttons (XY, XZ, YZ)

These buttons set the tracking plane to the designated coordinate plane.



Trackplane From Face (F)



Trackplane From Face opens the *Trackplane From Face* dialog box and sets the tracking plane to the next face you select in the database.

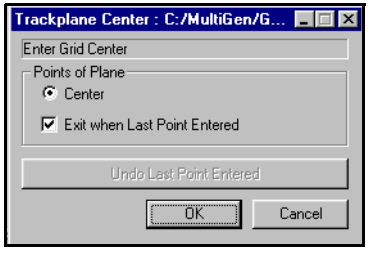


If **Exit When Face Selected** is set, the *Trackplane From Face* dialog box closes automatically when you select the face; otherwise it remains open until you click **OK**.

Trackplane Center



Trackplane Center opens the *Trackplane Center* dialog box and sets the grid to the next face you select in the database.

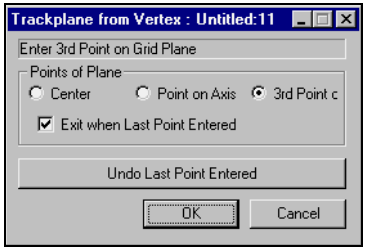


If **Exit When Last Point Entered** is set, the *Trackplane Center* dialog box closes automatically when you enter the **Center** point; otherwise it remains open until you click **OK**.

Trackplane From Vertex



Trackplane From Vertex sets the tracking plane using one, two, or three existing vertices that you select in the Graphics view. If you hold down the Alt key while you select a vertex, the tracking plane aligns with the face.



In the *Trackplane From Vertex* dialog box, **Center** defines the center of the grid. A line from **Center** to **Point on Axis** defines the x-axis of the grid, and **3rd Point on Plane** defines the new tracking plane.

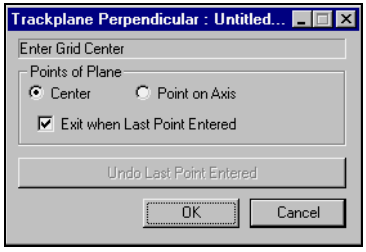
To move the center of the grid without changing its orientation, enter a new **Center** point and click **Done**.

If **Exit When Last Point Entered** is set, the *Trackplane From Vertex* dialog box closes automatically when you enter the **3rd Point on Plane**; otherwise it remains open until you click **OK**.

Perpendicular Tracking Plane (P)



Perpendicular Tracking Plane is used to redefine the tracking plane perpendicular to its previous orientation. In the *Trackplane Perpendicular* dialog box, **Center** defines the center of the grid, and the line from **Center** to **Point on Axis** defines the x-axis of the grid.

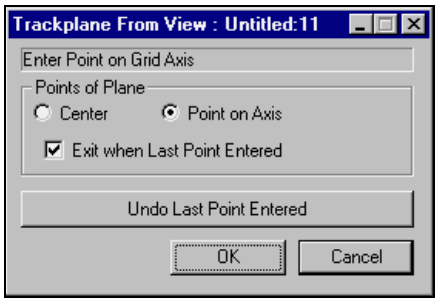


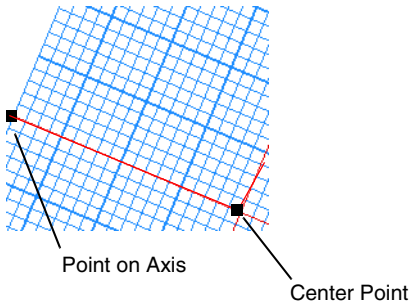
If **Exit When Last Point Entered** is set, the *Trackplane Perpendicular* dialog box closes automatically when you enter the **Point On**; otherwise it remains open until you click **OK**.

Trackplane From View



Trackplane From View orients the tracking plane based on two points. The *Trackplane From View* dialog box prompts you to select a **Center** point and a **Point on Axis** on the tracking plane. The grid origin is set at the indicated center point and aligns a grid quadrant at the point you select on the plane.





If **Exit When Last Point Entered** is set, the *Trackplane From View* dialog box closes automatically when you enter the **Point on Axis**; otherwise it remains open until you click **OK**.

Grid Type

Selects the type of grid to display. The current grid type is set by **Draw Grid**.

Offset

Offsets the tracking plane from the plane that defines its orientation. If the offset is relative to a coordinate plane, this field contains the label **Z:**, **Y:**, or **X:**. The label **N:** indicates the offset is in the direction of the normal to the defining plane.

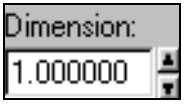
Draw Grid

Selects the drawing order method used to display the grid. The current drawing order is set by **Draw Grid**.

- **Over** draws the grid in front of the database scene.
- **Under** draws the grid behind the database scene.
- **Z-Buffer** draws the grid z-buffered with the database scene. Selecting this option puts the Graphics view in z-buffer drawing mode.

Dimension

Specifies the distance between grid lines. The grid preference **Grid Spacing** specifies the grid's default spacing. To cycle the grid dimensions through the **Grid Spacing** preferences, click the scroll arrows on the side of the list box. To select the maximum or minimum grid spacing defined in the preferences, hold down Ctrl while clicking the scroll arrows.



Grid Spacing Axes

Control the axis or axes to which grid changes will be applied. Changes can be applied to the x-axis, y-axis, or both the x- and y-axes.

Quadrants On/Off



Quadrants On/Off independently turns each grid quadrant on or off.

Placing a Coordinate on the Tracking Plane

To place a coordinate on the tracking plane, point to the place you want in the database and click the left mouse button. Some tools (such as **Wall** and **Peak**) override this

convention and let you specify the length of a construction edge with the left mouse button.

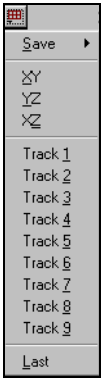
Storing Custom Tracking Planes

The **Tracking Plane** menu in the Control bar stores up to nine custom tracking planes for quick recall during modeling. You can also use it to change the grid orientation from **XY** to **YZ** or **XZ**.

To Store a Tracking Plane



Step 1. Click the **Tracking Plane Menu** button to display the menu.



Step 2. Click **Save** and choose a tracking plane number in the menu.

To Recall a Tracking Plane

Choose a tracking plane number from the **Tracking Plane** menu, or choose **Last** to recall the most recent tracking plane.

To Store and Recall Both Eyepoint and Tracking Plane

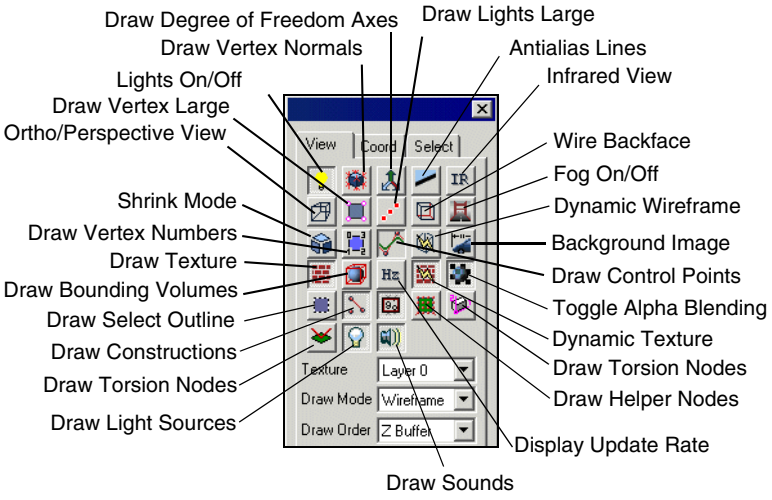
You can also store and recall the eyepoint position, with or without the tracking plane position, using the number keys or keypad with Num Lock on.

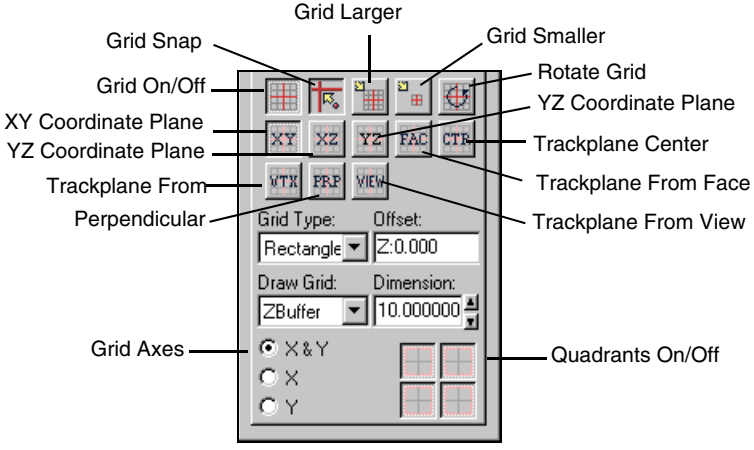
- To store an eyepoint position, press Shift+a number key (Shift+2, for example).
- To recall an eyepoint position, press the number key (2, for example).
- To store an eyepoint position, press Ctrl+Shift+the number key (Ctrl+Shift+3, for example).
- To recall the eyepoint and tracking plane position, press the number (32, for example).

9 The View Panel



View controls show and hide elements in the Graphics view.You can open the View panel from the Toolbar, or by choosing **View/Grid Controls** from the **View** menu.





Lights On/Off (L)



The **Lights On/Off** control in the View panel turns the effects of lighting on and off. When lighting is on, an L appears in the upper left corner of the Graphics view. The keyboard shortcut to toggle lights on or off is the L key. Lighting effects cannot be seen, even when this control is toggled on, until you calculate shading on faces. See "Database Shading" on page 620.

Draw Vertex Normals



The **Draw Vertex Normals** control in the View panel displays vertex normals when the Graphics view is in wireframe or wireframe-solid mode. See "Viewing Vertex Normals" on page 626.

Draw Degree of Freedom Axes



The **Draw Degrees of Freedom Axes** control in the View panel displays degree of freedom (DOF) axes in the Graphics view, making them easier to see and select. The positions of the DOF axes control the direction of movement of the geometry below the DOF node. See "Degrees of Freedom" on page 735.

Antialias Lines



The **Antialias Lines** control in the View panel displays the database using antialiased (smoother) lines on computers that support antialiasing.

Infrared View



The **Infrared View** control in the View panel displays the database using infrared colors and materials. See "Infrared View" on page 415.

Ortho/Perspective View



A *perspective view* is one in which faces in the foreground of a scene are drawn larger than faces in the background. An *orthographic view* is one in which faces are all drawn to the same scale. The **Ortho/Perspective View** control toggles the Graphics view between these options.

You may find it useful to select something in the database and choose **View/Center Select** (C) before switching to orthographic mode, because the scale of objects in the scene may change substantially.

Draw Vertex Large



Draw Vertex Large exaggerates the size of vertices in the Graphics view, making them easier to see and select.

Draw Lights Large



The **Draw Lights Large** control in the View panel controls the display size of light points in the Graphics view. See “Light Point Tools” on page 660.

Draw Wire Backface



The **Draw Wire Backface** control in the View panel suppresses the display of the wireframe outline around backfacing polygons when the Graphics view is in wireframe or wire solid drawing mode (see “Draw Mode” on page 321). A **B** appears in the upper-left corner of the Graphics view pane when **Draw Wire Backface** is on.

Fog On/Off



The **Fog On/Off** control in the View panel turns fog feature on and off. See “Creating Fog” on page 267.

Shrink Mode



The **Shrink Mode** (A) control in the View panel shrinks each face in the Graphics view about its center. This is helpful for selecting individual vertices and edges when more than one item occupies the same space. An **A** appears in the upper-left corner of the Graphics view when **Shrink Mode** is on. You can toggle **Shrink Mode** on or off by right-clicking in the Graphics view (to return input focus to the window) and pressing A. (Pressing Shift is not required).

Draw Vertex Numbers



The **Draw Vertex Numbers** control in the View panel displays the vertex numbers for each face. Vertex numbers start at 0 and increase, counterclockwise, when the face is viewed from the front.

Draw Control Points



The **Draw Control Points** control in the View panel displays or hides control points for a curve in the Graphics view. See “Editing a Curve” on page 117.

Dynamic Wireframe



The **Dynamic Wireframe** control in the View panel temporarily switches the Graphics view to wireframe draw mode while the eyepoint changes. This reduces drawing overhead, thus speeding up eyepoint movement. See “Changing the Eyepoint Position” on page 251.

Background Image



Turns background image display on and off. See “Using Background Images” on page 361.

Draw Bounding Volumes



The **Draw Bounding Volume** control in the View panel displays bounding volumes in the Graphics view. *Bounding volumes* are simplified shapes that approximate the shape of a model, providing the boundaries for determining when models collide. See "Bounding Volumes" on page 694.

Display Update Rate



The **Display Update Rate** control in the View panel displays the rate in Hertz (Hz) at which the Graphics view is being redrawn. This information appears in the lower-left corner of the Graphics view.

Draw Select Outline



The **Draw Select Outline** control in the View panel toggles highlighting of selected items on and off. When **Draw Select Outline** is turned off, selected items do not appear highlighted in the Graphics view; however, the nodes appear highlighted in the Hierarchy view and are displayed in the Select List.

Draw Constructions



The **Draw Constructions** control in the View panel toggles constructions on and off. When **Draw Constructions** is turned off, constructions are hidden in the Graphics view, giving you a clearer view of your model; when **Draw Constructions** is turned on, constructions are visible in the Graphics view.

Draw Clip Nodes



Turns the display of clipping polygons on or off in the Graphics view of the database. Clipping nodes (and their graphics view representations, clipping polygons) are created with the Instruments option.

Draw Grid Nodes



Displays and hides the Smart Grids that have been designed into models using the SmartEditor option.

A Smart Grid is a series of equally-spaced snap points. It provides a "peg board" of potential surfaces to which another model with the corresponding helpers can be snapped once the models are inside the SmartScene environment.

Draw Helper Nodes



Displays and hides the SmartScene helpers that have been designed into models using the SmartEditor option.

Helpers are like magnets, and *surfaces* are what they are attracted to. When you grab an object in the SmartScene environment, its helper(s) become active and "snap" to another object when brought in close proximity to a surface with the same attribute name.

Draw Torsion Nodes



Displays and hides the SmartScene torsions that have been designed into models using the SmartEditor option.

Torsions are alignment vectors that control the position of a model when it is "snapped" to a surface inside the SmartScene environment.

Draw Light Sources



Displays and hides light sources in the Graphics view. This is automatically set to be on when the State preference Draw Light Sources is set. See "Light Sources" on page 599.

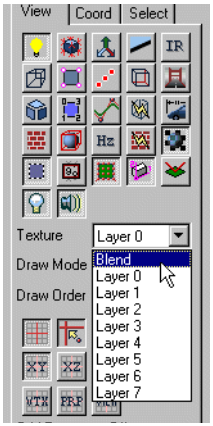
Draw Sounds



Displays and hides sound nodes in the Graphics view. Sound nodes are represented in the Graphics view by a small speaker which can be selected and repositioned. See “Audio Modeling” on page 669.

Texture Layers

The **Texture** list box in the View panel selects the current layer for applying a texture, and allows you to apply and blend MultiTexture layers on a face.



You select a layer in the View panel before you apply a texture to a face, or you can choose **Blend** in the list box to blend the textures according to the Texture Environment algorithm that you select for each texture in the *Texture Attributes* window.

Draw Mode

The **Draw Mode** (D) list box in the View panel contains three Creator drawing modes (see “Draw Mode Option Menu” on page 322). You can step through these modes by activating the Graphics view and pressing the keyboard shortcut D. The default drawing mode for each Graphics view is determined by the **Draw Mode** State preference.

Draw Mode Option Menu

The following options in the **Draw Mode** list box control how objects are displayed.

Solid

Objects are displayed by filling each face with its assigned color. Backfacing polygons are not displayed.

Wireframe

Objects are displayed by drawing each edge in the color (or material color) assigned to the face. Face color is not displayed.

Wire/Solid

Objects are displayed by filling each face with its assigned color and drawing each edge in black. Backfacing polygons are not filled, but the edges of all faces are displayed, unless **Draw Wire Backface** is turned on (see "Draw Wire Backface" on page 317).

Grid On/Off (G)



Controls the display of the grid on the tracking plane. The **Grid Off**, **Rectangle**, and **Radial** options Grid preferences control the appearance of the grid. See "Grid and Trackplane Controls" on page 306.

Grid Snap



Sets and clears the grid snap function. See "Grid and Trackplane Controls" on page 306.

Grid Larger



Doubles the size of the grid along each axis. See "Grid and Trackplane Controls" on page 306.

Grid Smaller



Reduces the size of the grid by half along each axis. See “Grid and Trackplane Controls” on page 306.

Rotate Grid



Changes the orientation of the grid and tracking plane. See “Grid and Trackplane Controls” on page 306.

Coordinate Plane Buttons (XY, XZ, YZ)

Set the tracking plane to the designated coordinate plane. See “Grid and Trackplane Controls” on page 306.



Trackplane From Face



Sets the tracking plane to the same orientation as a specified face. See “Grid and Trackplane Controls” on page 306.

Trackplane Center



Centers the grid at a selected location in the database without changing its orientation. See “Grid and Trackplane Controls” on page 306.

Trackplane From Vertex



Sets the tracking plane position using one, two, or three existing vertices. See “Grid and Trackplane Controls” on page 306.

Perpendicular Tracking Plane (P)



Sets the tracking plane perpendicular to its previous orientation. See “Grid and Trackplane Controls” on page 306.

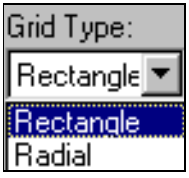
Trackplane From View



Orients the tracking plane based on a specified center point and a point on the plane. See “Grid and Trackplane Controls” on page 306.

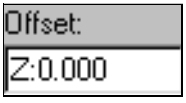
Grid Type

Selects the type of grid to display. The default grid type is set by the **Draw Grid** preference. See “Grid and Trackplane Controls” on page 306 and “Setting Grid Preferences” on page 304.



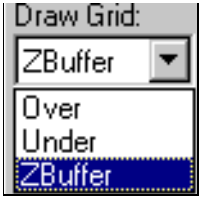
Offset

Sets the number of units to separate the tracking plane from the plane that defines its orientation. See “Grid and Trackplane Controls” on page 306.



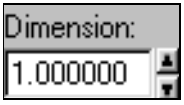
Draw Grid

Selects the drawing order method used to display the grid. The default drawing order is set by a Grid preference. See “Grid and Trackplane Controls” on page 306 and “Setting Grid Preferences” on page 304.



Dimension

Sets the distance between grid lines. The default dimension is specified by the **Grid Spacing** preference. See “Grid and Trackplane Controls” on page 306 and “Setting Grid Preferences” on page 304.



Grid Axes

Control the axis or axes to which grid changes will be applied. See “Grid and Trackplane Controls” on page 306.



Quadrants On/Off



Turns each quarter of the grid on or off. See “Grid and Trackplane Controls” on page 306.

10 Selecting

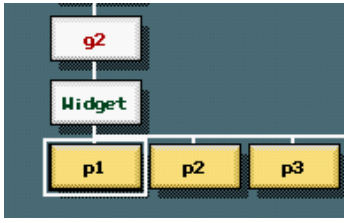


Before you can edit an item, you must select it. To select an item in the Graphics or Hierarchy view of a database you must be inPick mode. Click the **Pick Mode** button in the Toolbar to activate it, and then click an item in the database. Selected items are drawn with a dotted outline in the Graphics view and a solid outline in the Hierarchy view.

Item selected in the Graphics view



Node p1 selected in the Hierarchy view



- In the Graphics view, what you select when you click an item depends on the current modeling mode. For example, in Vertex mode, clicking a vertex of a house selects the vertex; in Face mode, it selects the faces that share the vertex, in Object mode, it selects the object that contains the vertex; and so on.
- In the Graphics view, faces and edges must be *frontfacing* (that is, they must face the eyepoint) to be selected by clicking. To select both frontfacing and backfacing faces, hold down the mouse button and drag a *fence*.
- In the Hierarchy view you can select items by clicking or fencing without regard to the modeling mode. To select a number of nodes, hold down the Shift key and click each node individually. If you want to select multiple nodes by dragging a fence, do not point directly at a node as you start to drag the fence, as this activates drag-and-drop editing ((see "Drag-and-Drop Editing" on page 287). To force Creator to start a fence from any point in the Hierarchy view, hold down the Shift key as you begin to drag the fence.
- You cannot select both a parent node and its children, since selecting a node automatically selects its children. If you drag a fence around a node and its children, only the parent will appear to be selected. The error message, "`Attempted recursive select`" appears in the status bar.

- To select only the children of a node, select the node in the Hierarchy view, choose **Select/Change Mode**, and then select a new mode from the Mode list on the Toolbar. For example, to select all the faces that are children of a Group node, select the Group node, choose **Select/Change Mode**, and then select Face from the Mode list.
- If you hold down the Alt key when selecting in either the Graphics or Hierarchy view, the selected item becomes the parent (see "Setting the Parent" on page 291).
- Clicking a shared edge or vertex often causes more than one item in the database to be selected. To isolate individual faces and vertices, click **Shrink Mode** in the View panel. The keyboard shortcut for **Shrink Mode** is "A."



Fence Selection



Selection results are governed by the **Select/Fence Touching** menu item. If **Fence Touching** is on, everything inside of or touched by a fence is selected. If the option is off, only items completely inside the fence are selected. You can also turn fence touching on or off by clicking **Fence Touch On** in the Toolbar.

You must position the mouse pointer very close to an edge or vertex in order to select it with the left mouse button. Using the middle mouse button allows a wider tolerance. To select a vertex or an edge, click it with the middle mouse button.

Customizing Selection Highlights

You can change the color and line appearance of the highlight that indicates which items are selected in the Graphics and Hierarchy views. This also changes the line type color, and thickness of the outline that appears when you do a fence selection.

Creator preferences control the color and line type of the selection highlights. To change either of these properties:

Step 1. Choose **Info/Preferences** to open the *Creator Preferences* window.

Step 2. Click the **Colors** tab to display Color preferences.

Step 3. In the Select Outline section, set the **Dashed Lines** option to highlight selected items with a *dashed* line. Clear the **Dashed Lines** option to highlight selected items with a *solid* line.

Step 4. Set the **Line Thickness** (in pixels). The default setting is 2; if you want to increase the visibility of the line, make it thicker by increasing the number of pixels for **Line Thickness**.

Step 5. Click the color chip next to **Line Color** to open the Color Editor, choose a new line color, and then click **OK**. The **Line Color** preference updates with the new highlight line color.

Step 6. Click **OK** to save your changes, or click **Restore Defaults** to reset the line type and color to their original default setting. Close the *Creator Preferences* window.

Controlling Highlights in the Graphics View

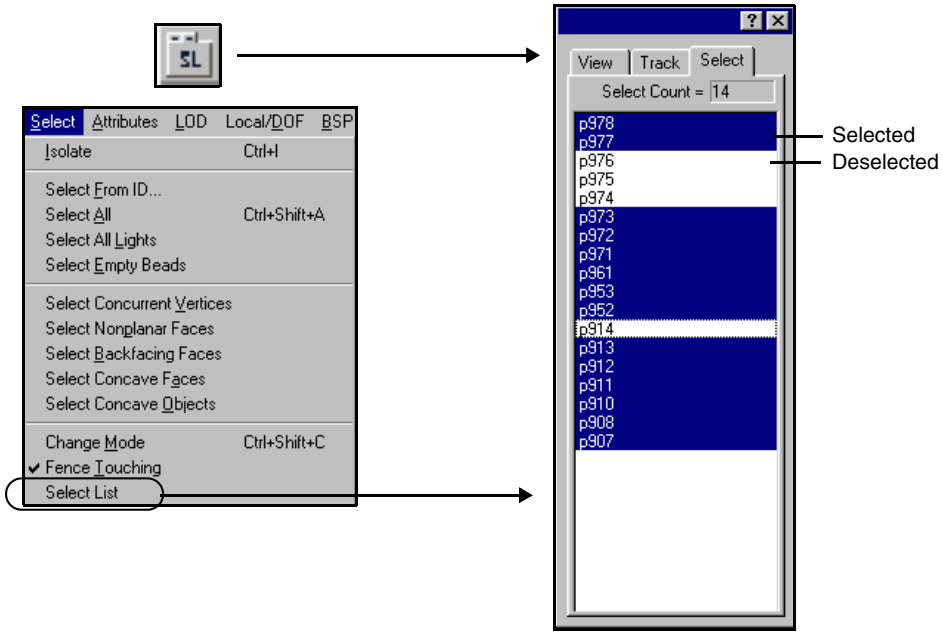


To toggle highlighting of selected items on and off, click **Draw Select Outline** in the View panel. When **Draw Select Outline** is turned off, selected items do not appear highlighted in the Graphics view; however, the nodes appear highlighted in the Hierarchy view and are displayed in the Select List.

Using the Select List



The Select List displays the identifier of each item in the database that currently is selected. To open the Select list, click the **Select List** button or choose **Select/Select List**. To deselect one or more items in this list (and thus deselect it in the database), click its identifier. Clicking a deselected identifier in the list selects it again.



To Add Items to the Select List

Hold down the Shift key while selecting additional items in the database. You cannot select two items in the same hierarchy subtree (such as an item and its descendant).

To Replace the Contents of the Select list

Select one or more items in the database. The previous list clears unless you hold down the Shift key.

To Remove Items From the Select List

Hold down the Shift key and click or fence items that are already selected in the database. The identifiers of the deselected items disappear from the Select list. You can also deselect an item by clicking its identifier in the Select list.

Displaying Attributes in the Select List

Attributes/Define ID Attributes lets you specify a set of attributes to appear in the Select List, based on the type of the selected node.

To display attributes in the Select List:

Step 1. Choose the modeling mode that corresponds to the node type for which you want to define ID attributes. For example, choose **Group** from the **Mode** list in the Toolbar to define ID attributes for a group. Each type of node has its own attributes.

Step 2. Choose **Attributes/Define ID Attributes**. The *Group ID Attributes* window opens, listing the attributes that correspond to the current modeling mode.

Step 3. Select each attribute you want to display by clicking its name in the *Group ID Attributes* window.

Step 4. Click **OK** to close the window.

Step 5. Select nodes in the Hierarchy view. The node's ID number and the attributes you selected are now display in the Select List.

Selecting With Arrow Keys

When an item is selected in either the Graphics view or the Hierarchy view, you can use arrow keys to deselect the current item and select its parent, child, or sibling. The *Up* arrow selects the parent of the original node; the *Down* arrow selects its first (leftmost) child; and the *Left* or *Right* arrow selects its left or right siblings.

Creator automatically performs a *Fit on Select* to adjust the eyepoint each time an item is selected with an arrow key. To prevent the eyepoint from changing, hold down Alt Ctrl while pressing an arrow key.

Using The Select Menu

Selection menu commands are useful tools for selecting items in the database that share certain characteristics. The following topics describe each menu command.

To search a database for other characteristics, including custom searches, see "Attribute Searching" on page 869.

Isolate

To work more efficiently, you can select items in your database and temporarily isolate them from the rest of the database. This allows you to more easily focus on a relevant part of your database and protects the parts of the database you do *not* want to change.

To open an isolated view, select one or more items in your database and then choose **Select/Isolate** (Ctrl+I). A new window opens, containing only the selected items. The new window looks and works like a Database window. It displays the same title as the original Database window, with "Isolate" and the node ID of the selected node appended.

Any changes you make to the isolated items are reflected in the original Database window. When you finish working in the isolated view, close its window or select other items in the original Database window.

The display in the isolated view is built from the highest-priority node that was selected when you chose **Select/Isolate**. If you perform an operation that changes the order of this node, all geometry will disappear from the isolated view. It will not disappear from your database, however. See "BSP Separation and Isolated Views" on page 930 for an example of this behavior.

The Live Button

Each isolated view has a **Live** button immediately to the right of the **Freeze Flags**. If this option is set, the isolated view automatically updates to reflect whatever is selected in the normal Database window. If this option is cleared, the isolated view contains the originally selected items, regardless of what you select in the Database window.

You can set whether the **Live** button will be the default setting by setting the **Default Isolate Window Live View** option in the Editing Preferences panel.

Select From ID

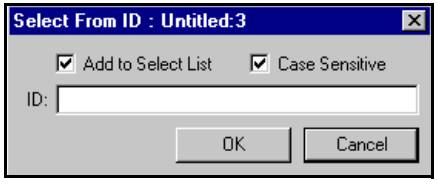
To select a node using its identifier, choose **Select/Select From ID** and enter the node ID in the **ID** field. The node is automatically added to the Select List.

To perform wild card selections, use the asterisk (*) key:

- p128* will select all nodes with p128 in the node ID (p128, p1280, p1281, and so on).
- p128* p129* will select all nodes with p128 or p129 in the node ID (the space character is the OR operator).
- p128* + *129 will select all nodes with a node ID that starts with p128 and ends with 129 (the plus character is the AND operator).
- p128* - *129 will select all nodes with a node ID that starts with p128 but does not end with 129 (the minus character is the NOT operator).

To add subsequent nodes to the Select list without removing the first node, set **Add to Select List**.

To center the eyepoint on the newly selected item, choose **View/Fit on Select** (V).



Select All

To select everything in the database, choose **Select/Select All** (Ctrl+A).

Deselect All

To make sure nothing is selected in the database, choose **Select/Deselect All** (Ctrl+D), click the database background, or hold down the Ctrl key and click anywhere in the database.

Select All Lights

To select all light points in the database, choose **Select/Select All Lights**.

Select Empty Nodes

To select all nodes that do not have children, choose **Select/Select All Empty Nodes**. Nodes at or below the face level will not be selected.

Select Concurrent Vertices

Select/Select Concurrent Vertices searches for adjacent vertices in the same face that share the same coordinate. When concurrent vertices are found, all but one of these vertices are selected. Press Delete or Backspace to remove the extra vertices. Vertices share a coordinate if the distance between them is less than the **Match Vertices Tolerance** Editing preference.

Select Faces with Coaligned Vertices

Finds and selects polygons containing three or more successive colinear vertices that fall below the **Coaligned Vertices Tolerance** Editing preference.

Select Nonplanar Faces

Selects faces that contain vertices that are not coplanar. To determine if all the points in a face are coplanar, Creator forms a temporary plane from the last three vertices of the face, and then analyzes the distance between the surface of this plane and the location of each remaining vertex in the face. If the distance is less than the **Select Non Planar Faces Tolerance** value in the **Info/Preferences/Editing** panel, the vertex is considered to be coplanar (see "Editing Preferences" on page 53). If any vertex falls outside this tolerance, the face is considered nonplanar, and is selected. Faces containing nonplanar vertices should be corrected using the **Triangulate** tool (see "Triangulate Tool" on page 188).

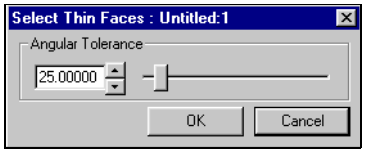
Select Backfacing Faces

Selects faces that are backfacing relative to the current eyepoint.

Select Thin Faces

Selects faces that have a corner less than a specified angle.This is useful for finding "sliver" polygons in your database.

When you choose **Select Thin Faces**, a dialog box opens. You set the tolerance angle to search for (the default value is 25).



If nothing is selected, any faces in the current database that meet or fall below the specified tolerance are selected. If faces are selected when **Select Thin Faces** is invoked, the thin faces are selected only from those faces.

Select Concave Faces

Selects faces that are not convex. Convex faces are more efficiently displayed than concave faces. To change the selected concave faces into convex faces, choose **Edit/ Triangulate**.

Select Concave Objects

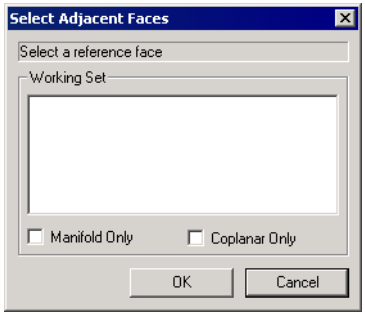
Selects objects that are not convex. Convex objects are more efficiently displayed than concave objects.

Select Adjacent Faces

Selects faces that are "topologically adjacent" to a reference face. Two faces are "adjacent" if they share a common edge. Two edges are common if their respective vertices are collocated. Two vertices are collocated if the distance between them is smaller than the "Match Vertices Tolerance" defined on the Editing Preference page.

When you choose **Select Adjacent Faces** in the **Select** menu, a dialog box opens and prompts you to select a reference face. **The Working Set** displayed in the dialog box specifies the scope under which adjacent faces will be searched in the database. The nodes listed here are hierarchical ascendants of the reference face, and each defines a different scope for searching. The scope of searching defined by a node includes all descendants of that node. Therefore, the node listed first has the largest number of nodes in its scope (it always includes all faces in the database). The node listed last has the smallest number of nodes. Since searching for adjacent faces in a large database is a processing-intensive function, you may find it useful to limit the number of nodes actually searched by selecting your working set node carefully.

After you select a reference face and specify a working set node, the tool searches all faces below that working set node for ones that are adjacent to the reference face, and selects each one it finds. In addition, the number of faces found is displayed at the top of the dialog box.



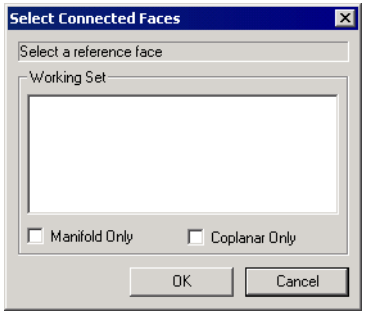
There are two options on the dialog that allow you modify the search for adjacent faces. Set **Manifold Only** to select only those adjacent faces that share a manifold edge; see “Select Non-Manifold Faces” on page 338 for more information about manifold faces and edges. Set **Coplanar Only** to select only those adjacent faces that are in the same plane as the reference face.

Select Connected Faces

Selects faces that are "topologically connected" to a reference face. When discussing "topological connectivity" with respect to faces, the notion of "adjacency" must be defined. Two faces are "adjacent" if they share a common edge. Two edges are common if their respective vertices are collocated. Two vertices are collocated if the distance between them is smaller than the "Match Vertices Tolerance" defined on the Editing Preference page. Using this definition of "adjacency", we can now define "connected" faces. Face A is "connected" to face B if A is "adjacent" to B or if there is one or more "adjacent" faces between A and B.

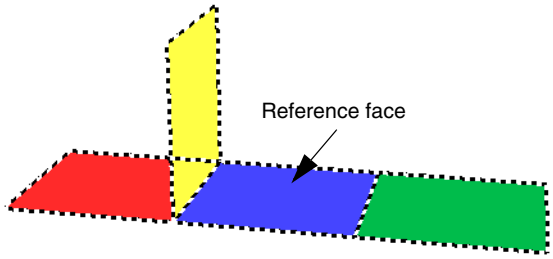
When you choose **Select Connected Faces** in the **Select** menu, a dialog box opens and prompts you to select a reference face. The **Working Set** displayed in the dialog box specifies the scope under which connected faces will be searched in the database. The nodes listed here are hierarchical ascendants of the reference face and each defines a different scope for searching. The scope of searching defined by a node includes all descendants of that node. The node listed first, therefore, has the largest number of nodes in its scope (it always includes all faces in the database). The node listed last has the smallest number of nodes. Since searching for connected faces in a large database is a processing intensive function, you may find it useful to limit the number of nodes actually searched by selecting your working set node carefully.

After you select a reference face and specify a working set node, the tool searches all faces below that working set node for ones that are connected to the reference face, and selects each one it finds. In addition, the number of faces found is displayed at the top of the dialog box.

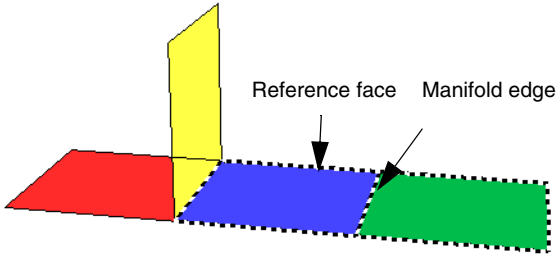


There are two options on the dialog that allow you modify the search for connected faces. Set **Manifold Only** to select only those connected faces that share a manifold

edge. See “Select Non-Manifold Faces” on page 338 for more information about manifold faces and edges. This example shows the connected faces selected, and then those connected faces that also share a manifold edge.



With **Select Connected** only, all of the faces connected with the reference face are selected.



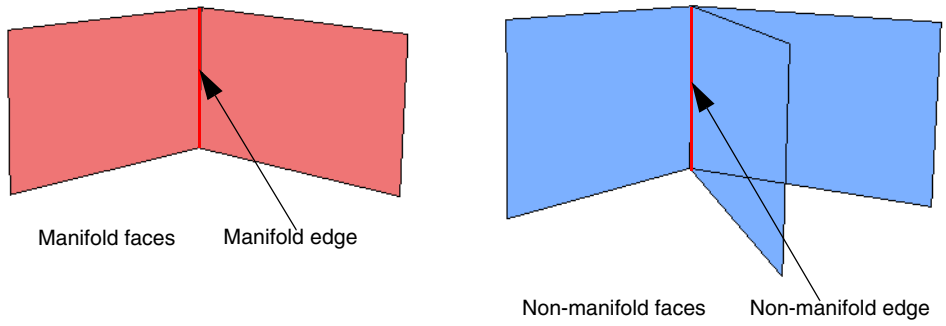
With **Select Connected** and **Manifold Only** set, only the faces that share a manifold edge are selected.

Set **Coplanar Only** to select only those connected faces that are in the same plane as the reference face.

Select Non-Manifold Faces

Selects all non-manifold faces within the database. The number of non-manifold faces that were selected displays at the bottom of the Creator window.

A manifold face contains only manifold edges. A manifold edge is an edge that is shared by at most two polygons. Conversely, a non-manifold edge is an edge that is shared by more than two polygons.



You might want to find non-manifold faces if you are looking for discontinuous surfaces. A continuous surface should not contain non-manifold faces. Non-manifold faces are faces that define where a surface changes from continuous to discontinuous.

Select Concurrent Faces

Selects all faces within the database that are on top of other faces. The concurrent faces must share all edges with each other. The number of concurrent faces that were selected displays at the bottom of the Creator window.

This is useful for finding concurrent faces to delete or to cut in as subfaces. See “Cut Subfaces Tool” on page 190 for more information about subfaces.

Change Mode

To change the mode in which an item is selected, select a new modeling mode in the **Mode** toolbox and choose **Select/Change Mode** (Ctrl+Shift+C). For example, if you select an object in Object mode, change to Vertex mode (Shift+V), and choose **Change Mode**, the vertices of the original object will be selected.

Fence Touching

Controls what is selected when you perform a fence selection. If **Fence Touching** is on, everything inside of or touched by a fence is selected. If the option is off, only items completely inside the fence are selected. You can also turn fence touching on or off by clicking **Fence Touch On** in the Toolbar.

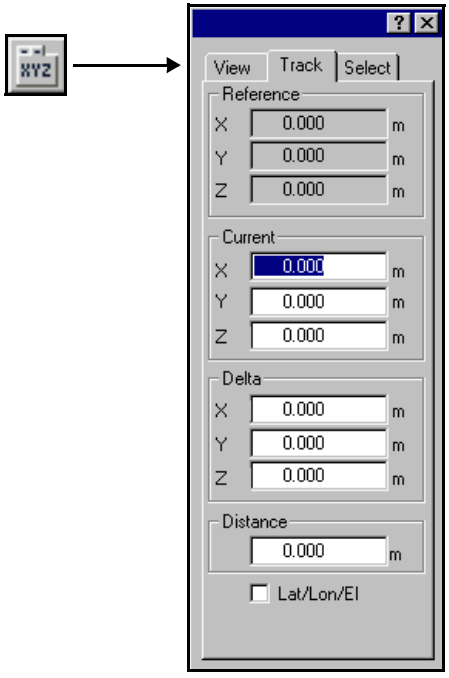


Select List

Opens the Select List. (See “Using the Select List” on page 329.)

11 The Coordinate Panel

The Coordinate panel shows information about the last two database coordinates that were entered or selected. To open the Coordinate panel, click the **Coordinate Panel** button in the Toolbar, or choose **Track Coordinate** in the **View** menu. You can also use this panel to place coordinates at exact positions on the tracking plane.



Reading the Coordinate Panel

The Coordinate panel controls have the following functions:

Reference

Shows the last coordinate entered or selected in the database.

Current

Shows the coordinate you are entering. Although it is more common to use the mouse, you can enter a coordinate by editing the value of each component (**X**, **Y**, and **Z**) and pressing Ctrl+Enter.

Delta

Shows the offset in *x*, *y*, and *z* from the **Reference** coordinate to the **Current** coordinate. To change the **Current** coordinate, edit the values of this field and press Ctrl+Enter. Creator calculates the new **Current** coordinate by adding **Delta** to the **Reference** coordinate.

Distance

Shows the length of the **Delta** vector (the distance between the **Current** and **Reference** coordinates). To change the **Current** coordinate, edit this value and press Ctrl+Enter. The new **Current** coordinate is calculated by adding the **Distance** to the **Reference** coordinate in the direction of the **Delta** vector.

Using the Coordinate Panel With Tools

You can place vertices on the tracking plane by entering coordinates in the Coordinate panel. The following example illustrates how to do this with the **Rectangle** tool. You can use the Coordinate Panel in a similar manner to place the vertices required by other tools.

Step 1. Set a group node or object node as parent.



Step 2. Choose **Rectangle** from the **Face Tools** toolbox.



Step 3. Click **Coordinate Panel** in the Toolbar.

Step 4. In the Coordinate panel, type *x* and *y* coordinates for the first point in the **Current** field. Press Enter. The point is displayed in the Graphics view and the coordinates move from **Current** to **Reference**.

Step 5. Select **Second Corner** in the *Rectangle* dialog box.

Step 6. Type *x* and *y* coordinates for the second point in the **Current** field and press Enter. The second point appears in the Graphics view.

Step 7. Click **OK** in the *Rectangle* dialog box.

12 Modeling Tips and Techniques

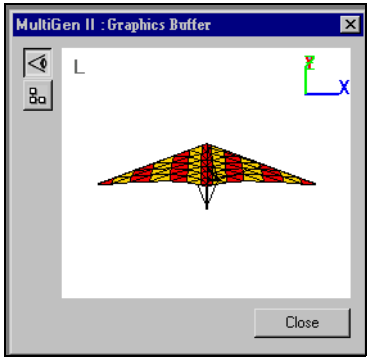
The following topics help save time and effort when modeling. These include using existing 2D drawings of objects to make 3D objects, using local coordinate systems when working far from the origin, and building databases with instancing, external references, and replication to maximize your system resources.

Working With the Graphics Buffer

The Creator graphics buffer is similar to the Windows Clipboard, but its contents are available only to the Creator application. You can copy or cut geometry to the graphics buffer, and paste the contents into a database, or use the contents of the graphics buffer to replace existing geometry. In addition, the **Populate** and **Scatter** tools use the contents of the graphics buffer to add geometry to terrain databases. To get consistent results when you paste from the graphics buffer into a database, you must make sure the geometry in the graphics buffer is centered at the origin.

Viewing the Contents of the Graphics Buffer

View Graphics Buffer in the **Edit** menu displays the Graphics buffer. The contents of the Graphics buffer cannot be edited.



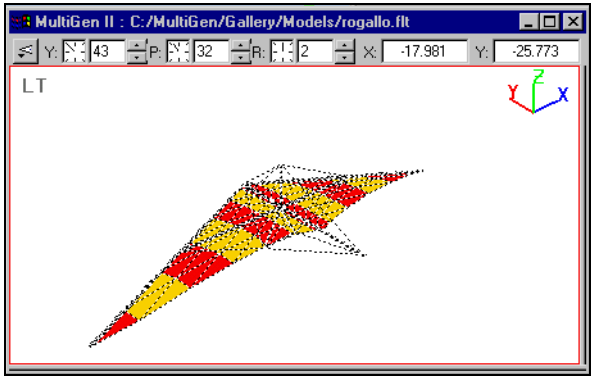
Copying and Pasting

Copy (Ctrl+C) and **Paste** (Ctrl+V) in the **Edit** menu let you copy geometry from one part of a database to another or copy geometry between different databases.

If the pointer is in a text field (for example, the **Comments** of an Attributes window), **Copy** and **Paste** can also be used to copy text strings.

Copy (Ctrl+C)

Copy in the **Edit** menu (and available from the Toolbar) copies the selected graphics into the Graphics buffer. If the pointer is in a text field, copies the selected text into the Text buffer.



Cut (Ctrl+X)

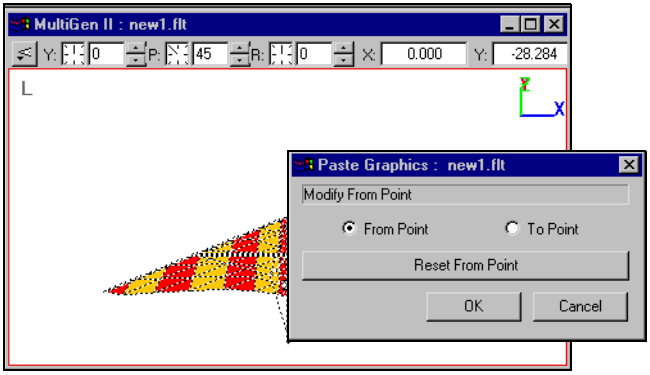
Cut in the **Edit** menu (and available from the Toolbar) copies the selected graphics into the Graphics buffer and deletes it from the database. If the pointer is in a text field, copies the selected text into the Text buffer and deletes it from the text field.

Delete

Delete in the **Edit** menu deletes the selected graphics or text without saving it in a buffer.

Paste (Ctrl+V)

Paste in the **Edit** menu (and available from the Toolbar) inserts the contents of the Graphics buffer under the current parent and starts **Translate** to position the new geometry. (See "Translate Tool" on page 144.) If the pointer is in a text field, **Paste** inserts the contents of the Text buffer.

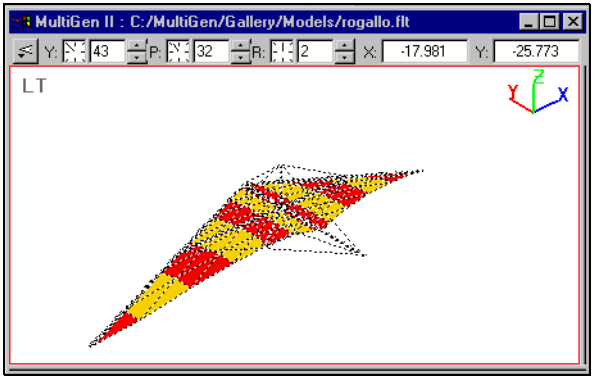


Replace Graphics

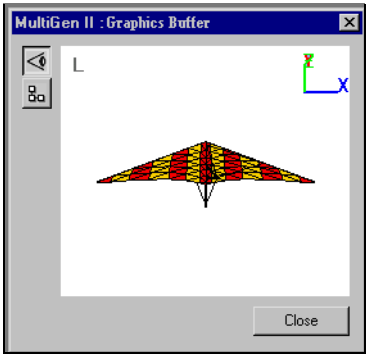
Discards each selected item in the database and replaces it with a copy of the geometry in the Graphics buffer. The parent of each selected item must be a legal point of attachment for the new geometry.

To Replace Graphics

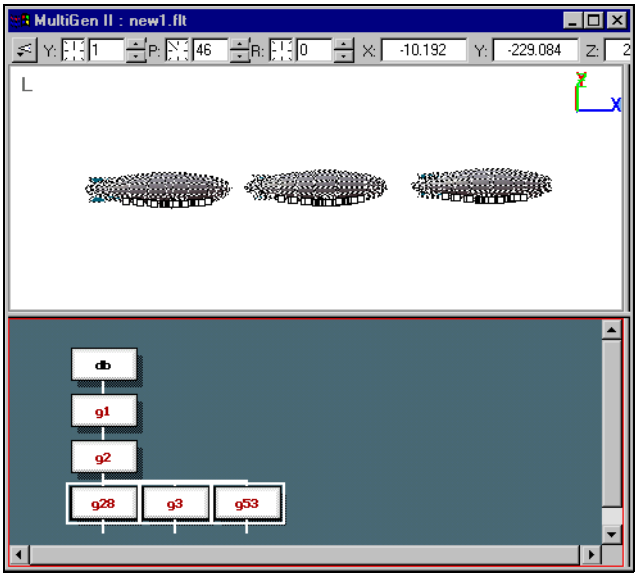
Step 1. Select the geometry to be substituted into the database.



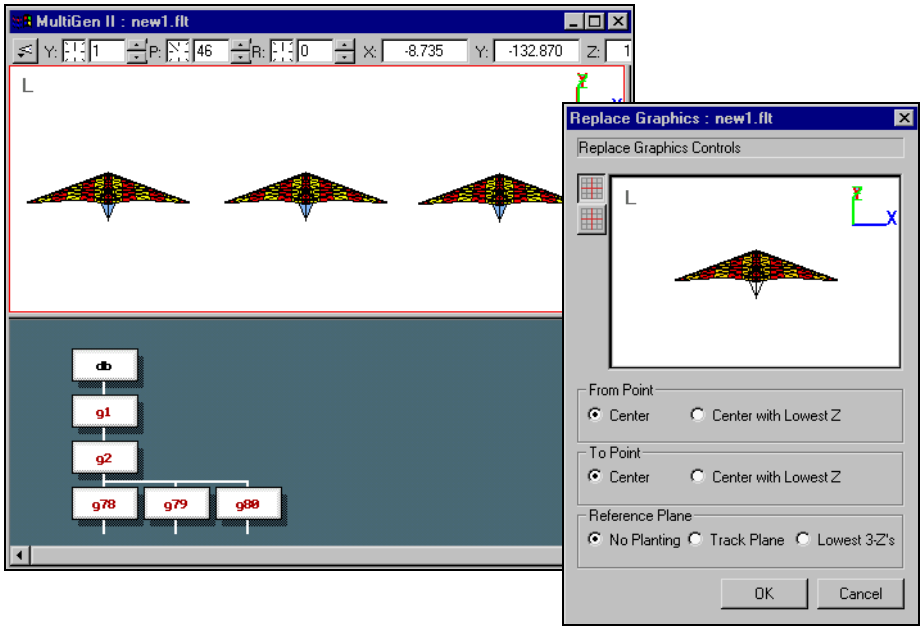
Step 2. Choose **Edit/Copy** (Ctrl+C) to copy the geometry into the Graphics buffer.



Step 3. Select the items you wish to replace. These items do not need to be in the same database as the original geometry. However, it is a good idea to select only one type of node. For example, select only groups or objects.



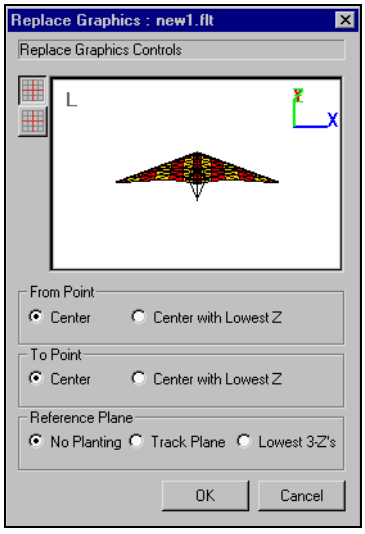
Step 4. Choose **Edit/Replace Graphics** and set the parameters in the *Replace Graphics* dialog box.



Replace Graphics Parameters

Replace Graphics parameters control the relative alignment of the selected items in the database and the items that will replace them in the Graphics buffer.

The *Replace Graphics* dialog box contains the following parameters:



From Point

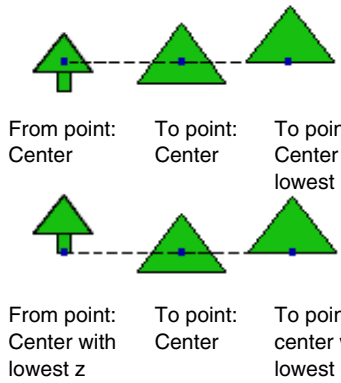
Defines the **From** point in the Graphics buffer. The geometry in the Graphics buffer is translated from the **From** point to the **To** point on each selected item.

- **Center** selects the geometric center of the contents of the Graphics buffer as the **From** point.
- **Center with Lowest Z** selects the average of the lowest *z* coordinates in the Graphics buffer as the **From** point.

To Point

Defines the **To** point on each selected item. As each item is replaced, the software calculates its **To** point and then maps the Graphics buffer **From** point to the **To** point.

- **Center** selects the geometric center of each selected item as the **To** point.
- **Center with Lowest Z** selects the average of the lowest *z* coordinates in the selected item as the **To** point.



From point of tree in buffer (with trunk) is translated to To point of each selected tree in the database

Setting From and To Points for Replace Graphics

Reference Plane

Controls whether and how a **Plant** is performed to place each replacement item on the tracking plane.

- **No Planting** inserts the replacement item without changing its geometry.
- **Trackplane** plants each replacement item on the tracking plane by moving its lowest vertices to intersect with the tracking plane. When **Trackplane** is selected, all replacement items are planted on the same plane.
- **Lowest 3-Zs** sets the lowest three vertices in each selected item to redefine the tracking plane before inserting and planting the replacement item. This allows each replacement item to be planted on a different tracking plane.

Centering a Model at the Origin

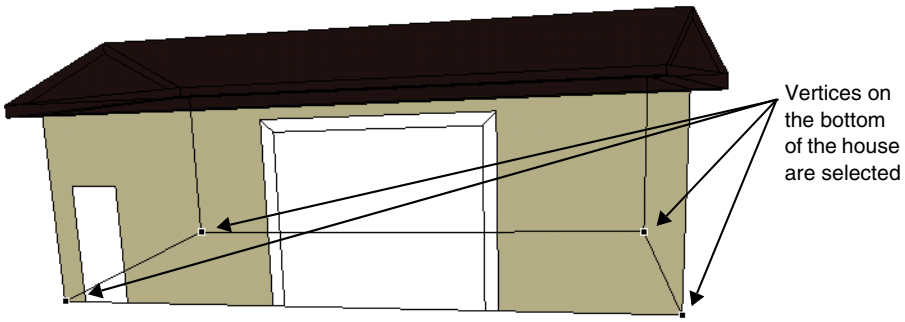
When you copy or paste all or part of a model, or use the **Populate** or **Scatter** tool to place models on a terrain, you will get unpredictable results if the part of the model you are trying to paste or place on terrain is not centered on the origin of the database from which you copied it. In addition, if you are using a model as an external reference, and the model is not centered at the origin, you will have to take its offset into consideration when you are calculating a translation; if you use the model many times (for example, placing 1000 external references of a tree into a terrain database), you might want to center the database at the origin first, to make your subsequent calculations easier.

The following example shows how to correct the position of a model if it is not centered at the origin. In this example, the center of the *bottom* of a house is transformed to the origin, because that is the part of the house that will rest on the terrain.

Step 1. Open the database of the model.

Step 2. Select Vertex mode from the Toolbar.

Step 3. Hold down the Shift key and select the vertices that will rest on the terrain. For example, in the following figure, you would select the four corners of the bottom of the house.

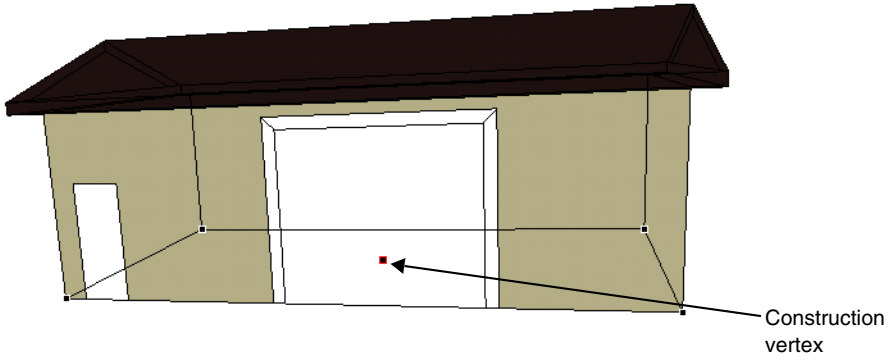




Step 4. Open the Vertex toolbox.

Step 5. Click the **Construct Average Vertex** tool. This creates a construction vertex that is centered on points you selected (in this example, on the bottom of the house).

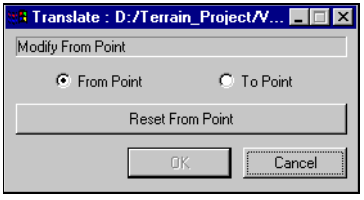




Step 6. Select Object or Group mode from the toolbar (whatever mode is applicable) and select the entire model.

Step 7. Open the **Maneuver** toolbox and click the **Translate** tool. The *Translate* dialog box appears.



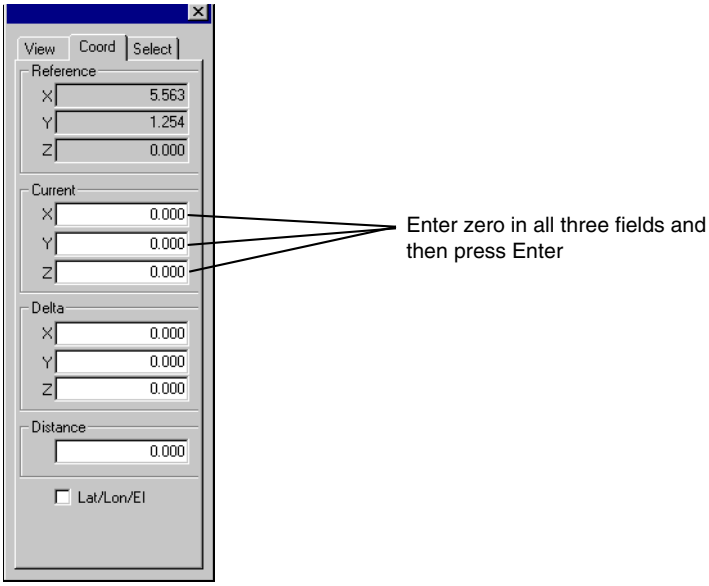


Step 8. Click the construction vertex to select it as the **From Point**. The selection in the Translate dialog box changes to the **To Point**.



Step 9. Open the **Coordinate Display** panel.

Step 10. In the **Current** area, enter 0 in the **X**, **Y**, and **Z** fields, and press Enter. This sets the **To Point** to the database origin.



Step 11. Click **OK** in the *Translate* dialog box.

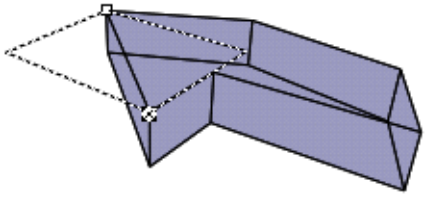
The bottom of the model is now centered at the database origin.

Referencing a Vertex or Edge

It is often necessary to place a coordinate at the same location as an existing vertex or along the line defined by an existing edge in the database. When you place a coordinate with the middle mouse button, Creator looks for a nearby vertex or edge and uses it to determine the position of the new coordinate, as follows:

To Reference a Vertex

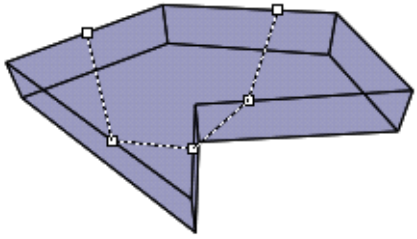
Point to a vertex and middle-click to place a new coordinate at the same location.



You can force a vertex reference by holding down Ctrl while clicking with the middle mouse button. This prevents you from accidentally referencing an edge.

To Reference an Edge

Point to an edge (not near a vertex) and middle-click to place a vertex on the edge, or drag with the middle mouse button along the edge and release the mouse button at the point where you want to place the vertex.



You can force an edge reference by holding down Alt while clicking with the middle mouse button. This prevents you from accidentally referencing a vertex.

To Snap a Vertex to the Tracking Plane

Hold down Ctrl+Alt and middle-click to enter a coordinate at the point of intersection between the line defined by the edge and the tracking plane.

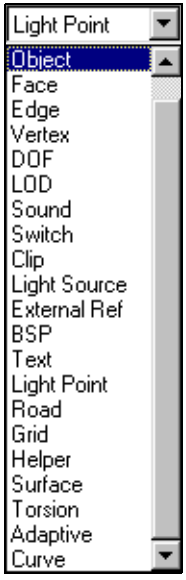
Changing the Modeling Mode

You can change the current modeling mode in several ways:



- Select the desired mode from the **Mode** toolbox.
- Use a keyboard short cut. For example, press Shift+F to change into Face mode.
- Select the desired mode from the **Mode** list in the Toolbar.

  The **Mode** list displays the current modeling mode. Click the arrow on the right side of the box to open the list of modes.



Correcting Non-coplanar Vertices

To draw correctly in the realtime system, the vertices of a face must all be in the same plane. You can use **Project** to correct nonplanar faces as follows:

Step 1. Choose **Select/Select Nonplanar Faces** to search the database for nonplanar faces. All nonplanar faces become selected.

Step 2. In the grid controls, click **VTX** and set the tracking plane by middle-clicking three vertices on one of the selected nonplanar faces. Pick vertices that define the plane you want to use for the whole face.



Step 3. In **Vertex** modeling mode, select all the vertices of the nonplanar face as well as all shared vertices in adjacent faces. If you do not get all shared vertices, the faces will not match after **Project**.



Step 4. Open the **Modify Geometry** toolbox.



Step 5. Click **Project** to force all selected vertices onto the tracking plane.

Using Quick Palettes

Quick palettes let you change the current color, material, or texture by right-clicking a palette button on the Toolbar. This is a faster way to change the current selection than opening and clicking in a palette. When you open a database, Creator collects all the colors, materials, and textures that have been used in the file and creates a set of Quick palettes containing only these items. Because Quick palettes are built when you open a database, new colors, materials, and textures that you apply while editing do not show up until you close and reopen the file.



To access a Quick palette, position the mouse pointer over the Color, Material, Light Source, or Texture button at the top of the screen, and hold down the right mouse button. The related quick palette appears, or an audible beep indicates that a Quick palette is not available.

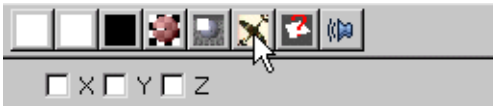
The following main palettes can have user-defined Quick palettes:

- Color (primary and alternate)
- Material
- Light Source
- Texture

You can open Quick palettes from the Database window, the *Terrain* window, or from the **From Points** box in the **Put Texture** and **4-Point Put** tools.

To Select an Item From a Quick Palette

Step 1. Position the pointer over the current display for the palette you want (**Current Material** in the Database window, for example) and hold down the right mouse button. If a Quick palette contains any items, a pop-up appears; if not, a beep sounds.

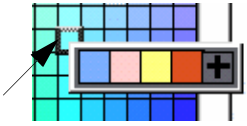


Step 2. Continue to press the right mouse button as you drag the pointer over an item you want, and then release. The pop-up disappears and the selected item becomes current.

To Add an Item to a Quick Palette

Step 1. Open one of the main palettes (the Color palette, for example).

Step 2. Hold the right mouse button down over the item you want to add. If the selected item is already in the Quick palette, a beep sounds. If the selected item is not currently in the Quick palette, the Quick palette pops up and allows you to add the item.



Step 3. Drag the pointer to the plus sign (+) on the Quick palette to add the item to the end (right side) of the list, or drag the pointer over an existing item to replace it.

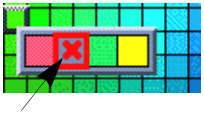
Note: As a shortcut, press Shift and click the item you want to add with the left mouse button. The item is automatically added to the end of the list and the pop-up does not appear.

To Delete an Item From a Quick Palette

Step 1. Open one of the main palettes.

Step 2. Press Shift as you hold down the right mouse button anywhere inside the palette. If a Quick palette contains any items, it pops up.

Step 3. Drag the pointer with the red "X" over the item you want to remove and release the mouse button. The pop-up disappears and item is removed from the list.



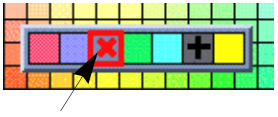
Note: To cancel the removal of an item, drag the pointer outside the pop-up and release the mouse button.

To Remove or Restore Quick Palette Items

Step 1. Open one of the main palettes.

Step 2. Press Shift as you hold down the right mouse button anywhere inside the palette. If a Quick palette contains any items, it pops up.

Step 3. While continuing to hold down the right mouse button, click the middle mouse button on an item to be removed.



Note: A red "X" follows the pointer over a new item to be marked for removal. After the item is selected, it is marked with a black "X".

Step 4. If the item has already been marked for removal, click it again with the middle mouse button to restore the selected item.

Step 5. Release both mouse buttons. The pop-up disappears and the selected items are removed from the list.

To Update Quick Palettes

New colors, materials, and textures that you use in the database do not automatically appear in Quick palettes. To update Quick palettes:

Step 1. Save the edited database file.

Step 2. Close the file.

Step 3. Reopen the database. The new Quick palettes contain all colors, materials, and textures that are in use in the database.

Using Background Images

Background images are images that you use as a backdrop while creating a model. These can be images that you scanned into a computer or an existing image in any of these file formats: RGB, RGBA, INT, INTA, TIF, GIF, JPG, PCX, LBM, TGA, PIX, or BMP. When this image is set as a background image, you can more accurately create a three-dimensional model to the same dimensions as the image.

Background images are especially useful for digitizing a drawing with the mouse; for example, when converting a blueprint of an object into a model that is true to scale. To make it easier to see both the database and image, switch the Graphics view into **Wireframe** drawing mode (see "Draw Order" on page 682).

To Load a Background Image

Step 1. To load a database image, choose **View/Background Image/Read Image**.

Step 2. Select a file in the *Open* dialog box, and then click **Open**.

The image appears in the lower-left corner of the Graphics view, and Creator displays the *Zoom/Move Image* dialog box.

Note: If the image does not appear, click **Background Image** in the View panel.



To make it easier to see both the database and image, switch the Graphics view into the **Wireframe** drawing mode (see "Draw Mode" on page 321).

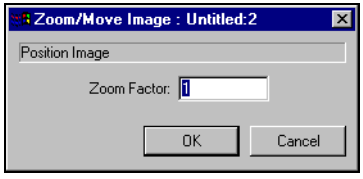
To Zoom or Move a Background Image

Step 1. Choose **View/Background Image/Zoom/Move Image**.

Step 2. Enter a number in the **Zoom Factor** field on the *Zoom/Move Image* dialog box.

You can also zoom in on the image by clicking the image with the middle mouse button. Each time you click the image, the zoom factor increases by a factor of one (for example, the zoom factor creases from two to three).

To zoom out on the image, enter a number in the **Zoom Factor** field or hold down Alt and middle-click on the image. This decreases the zoom factor by one. You cannot zoom out beyond a zoom factor of one. This is the point at which each texel in the background image maps to one pixel on the screen.

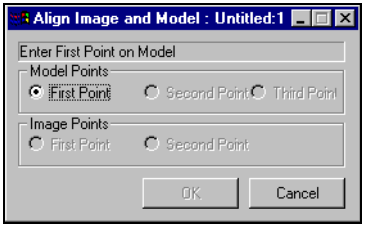


Step 3. To move or scale the image relative to the eyepoint, drag the image into position in the Graphics view with the left mouse button. You do not need to point to the image; the amount and direction of the drag define the amount and direction of the move.

Note: You can reposition the image at any time by choosing **View/Background Image/Zoom/Move Image** to display the *Zoom/Move Image* dialog box.

To Align a Model to the Background Image

Step 1. To align a selected portion of the database with the image, choose **View/Background Image/Align Image and Model.**



Step 2. Select three points in the database and two points on the model. Creator adjusts the eyepoint to match the first two points on the model with the two image points. The third point on the model defines the plane the eyepoint looks toward. Although it is useful to think of the database being scaled and rotated to match an image, alignment is actually performed by adjusting the eyepoint, not changing the database.

Step 3. Click **Background Image** in the View panel to show or hide the display of the background image.

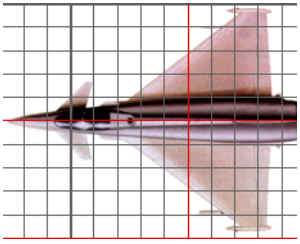


To Model with a Background Image

Step 1. Load, scale, and position the background image.

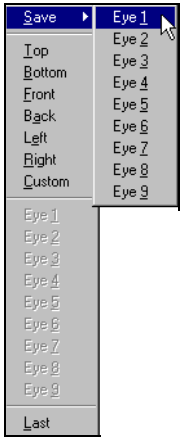
Step 2. Set the grid spacing to an even increment of the model's key dimension (length or wingspan). For example, if the wingspan of an airplane is 15 meters, set the grid spacing to one, three, or five.

Step 3. Use the grid controls to align the tracking plane grid along the center line of the background image.



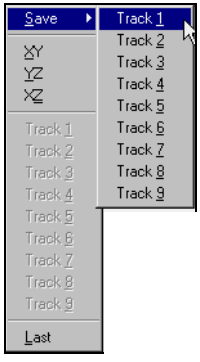


Step 4. Save the eyepoint position.





Step 5. Save the tracking plane position.



Step 6. Create the faces for the model using the background image as a reference.

Step 7. Make the faces into a three-dimensional model using standard tools such as **Wall** and **Loft**.

Step 8. Click **Background Image** in the View panel to turn off display of the background image when you are finished modeling.

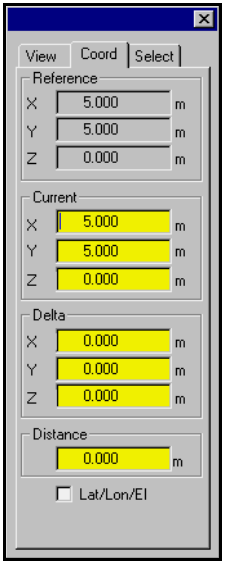


Working With Local Coordinate Systems

The Creator modeling environment is based on a *global database system*. The database origin is centered at 0, 0, 0 in the global database system and most models use the origin as their center. However, at times it may be inconvenient to use this system. When you are working at distances far from the database origin or in a different orientation from the global axis, you may want to define a *local coordinate system*.

A local coordinate system has its origin wherever you want it. You can place it at the center of a rotating planet in a solar system model or at a designated spot on the tracking plane so you can apply transformations using simple numbers.

The **Current**, **Delta**, and **Distance** values in the Coordinate panel are shown in yellow to indicate they are showing values in the local coordinate system. However, the **Reference** numbers are still shown in black; they do not change.



You can also define a local coordinate system using a node that contains a transformation in its path (see "Insert Transformation Matrix Tool" on page 157). Select a node in the hierarchy *below* the node with the transformation and choose **Local-DOF/ Set Local From Path**.

To Define a Local Coordinate System



Step 1. Click **Grid On/Off** in the View panel to make the tracking plane visible. It is easier to do the next step when the tracking plane is visible.

Step 2. Choose **Local-DOF/Set Local** to set a local coordinate system.



Step 3. In the Graphics view, click to select an origin, a point on the *x*-axis, and a point on the *y*-axis.

Step 4. Click **OK**.

To Define a Local Coordinate System With a Transformation

Step 1. Select a node in the hierarchy below the node with the transformation.

Step 2. Choose **Local-DOF/Set Local From Path**. The local coordinate system is centered on the object.

To Remove a Local Coordinate System

Step 1. In the Hierarchy view, select the node with the local coordinate system attached.

Step 2. Choose **Local-DOF/Set Global**. The local origin is cleared and the Coordinate panel is reset to show normal database coordinates.

Note: Transformations to a node are not affected by removing the local coordinate system.

To Add Geometry to a Local Coordinate System

Step 1. Define a local coordinate system.

Step 2. Select the geometry's node.

Step 3. Choose **Local-DOF/Make Local**. This redefines the node's coordinate values by placing a Put transformation matrix on the node (this does not alter its position). You must have a local coordinate system defined before you use this function.

To Remove Geometry from a Local Coordinate System

Step 1. Select the node.

Step 2. Choose **Local-DOF/Make Geometry**. This removes the transformation from the node without altering its position.

Instances

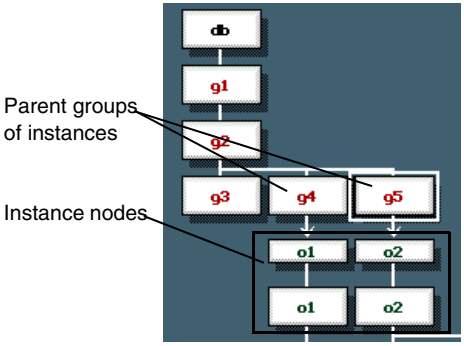
An *instance* is a reference to an existing geometry in a database. An instance looks the same as a copy of a geometry, but unlike copies, they are not geometries; they are simply pointers to an existing geometry.

Since only one copy of each geometry is stored in memory, instancing improves performance by saving computer memory, disk space, and processing time. Instancing can also increase the performance of the database in the realtime system, resulting in faster frame rates. Instancing is an efficient way to design a database, especially with simple, repetitive models, such as trees in a forest.

Clicking an instance node (the node with the arrow pointing to it) expands and contracts the display of its hierarchy. Because all instances are references to a single original item, selecting or editing one instance in the Hierarchy view simultaneously selects and edits all instances. In the Graphics view, selected instances are outlined in blue.

Creator translates, orients, and scales instances by automatically placing a transformation matrix on (or above) the parent group in the hierarchy when you use a **Maneuver** tool.

Instancing is indicated in the Hierarchy view by an arrow leading from the parent to a small copy of the instanced node.



In the Graphics view, the instance is created at the same coordinates as the original geometry, so it is not immediately visible. After you create the first instance, you must relocate it using one of the tools in the **Maneuver** toolbox. You can also use the **Duplicate** tool to make many copies of the instance once you create the first one.

To Create an Instance

Step 1. Create a new group node. The parent group of an instance cannot have any other children.

Step 2. Click **Parent** to make the new group node the parent for the instance.

Step 3. Select the geometry to be instanced.



Step 4. Open the **Create Tools** toolbox.



Step 5. Click **Create Instance**.

The instance is now on top of the original geometry. Use the **Translate** tool to move it to a different position, so it is visible:

Step 1. Select the group node that is above the instance node (do not select the instance node).



Step 2. Open the **Maneuver Tools** toolbox.



Step 3. Select the **Translate** (Shift+T) tool.

Step 4. Click **To Point** and then select a point in the Graphics view.

Step 5. Click **OK**. The instance is now located at a different position from the original geometry.

To Make Copies of an Instance

Step 1. Select the group node that is above the instance node (do not select the instance node).



Step 2. Open the **Duplicate Tools** toolbox and click the **Duplicate** (Shift+T) tool.

Step 3. Click **To Point** in the *Duplicate* window and then select a point in the Graphics view.

Step 4. Click **OK**. A new instance appears.

Step 5. To create additional instances, select the parent group node of the original node that was instanced (the one that does not have a translation matrix applied), and repeat steps 2 through 4. (If you duplicate a translated node, the new instance node will be translated from the **To Point** you specify.)

The additional instances will be placed in the hierarchy under their own group nodes, which share a parent group with the original geometry that was instanced. To make copies of all these instances at once, select the *parent group of all the instances*, and repeat steps 2 through 4.

To Change an Instance into Geometry

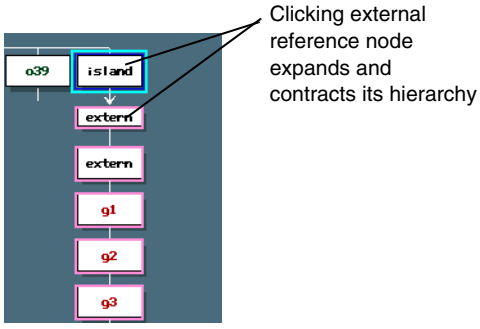
To convert an instance to normal geometry, select it and choose **Local-DOF/Make Geometry** (see "To Remove Geometry from a Local Coordinate System" on page 367).

External References

An *external reference* is a method that places geometry in one database by referencing the geometry of another database. External referencing allows you to include the contents of one database in another without pasting in the geometry. This saves disk space and streamlines the assembly of complicated databases. Unlike instances,

external references are *read only*; you can position, orient, and scale an external reference, but you cannot edit its contents. To edit the contents of an external reference, you must go to the database that contains the geometry (the *referenced* database) and make your changes. All external references to the edited database will change accordingly.

In the Graphics view, selected external references are outlined in red. In the Hierarchy view, they are outlined in blue. Clicking an external reference node (the node with the arrow pointing to it) expands and contracts the display of its hierarchy.



By default, external references are automatically loaded into memory and displayed when you open a database. You can automatically display or hide external references by setting the Flight preference **Read External References**.

Working with External References

External references allow you to include the entire contents of one database file in another file without cutting and pasting. To do this, place an external reference node in the hierarchy, then assign the directory path and file name of another database to the node. External references can be attached to any node that functions as a group (group, switch, DOF, etc.).

The full path and filename is displayed in the **Select Panel** (when you select the external reference node) and in the **ID** field of the *X-ref Attributes* window.

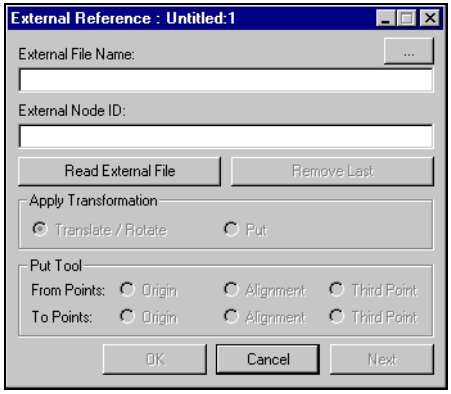
Creator uses the directory path and file name specified in the **ID** attribute of the *X-Ref Attributes* window to load each external reference. If no directory path is specified, the software searches for external references first in the local directory, then in the resource path defined by the environment variable MPI_CREATOR_FLTEXTERNPATH (see "Environment Variables" on page 43).

To Create an External Reference

Step 1. Select a group.

Step 2. Click **Parent** to set the database parent node for the external reference.

Step 3. Choose **File/External Reference**.



Step 4. In the *External Reference* window, type the name of the file to be referenced, or browse the file system and select the file.

Step 5. Click **Read External File**.

Step 6. In the Graphics view, use your mouse to select an origin point.

Step 7. Select an alignment point.

Step 8. Select a third point. The externally referenced model appears in the Graphics view.

Step 9. Click **Done** to finish or **Next** to create another external reference.

Note: If you click **Next**, then click **Read External File** again, another external reference is added to the database.

If you prefer, you can position the external reference by selecting **From Points** on the external reference and **To Points** in the database. This is analogous to using **Put** (see "Put Tool" on page 154), except you cannot change the scale of the external reference during positioning. (To change the scale, select the external reference node and turn on the **Scale** tool. See "Scale Tool" on page 149.)



You can also create an external reference by opening the **Create** toolbox and clicking **Create External Reference**. This creates an external reference node.To assign a file to the node, double-click it to display the *X-Ref Attributes* window. Click the **External** tab and enter the filename in the **ID** field.

External References in the Graphics View



You must be in **External Reference** modeling mode to select an external reference in the Graphics view. Selected external references are displayed with red vertices instead of white. When an external reference is selected, you can transform it using standard functions such as **Translate/Rotate** in the *External Reference* window or **Scale**.

Reading and Re-Reading External References

Setting the Flight preference **Read External References** automatically displays any external references when you open a database. External references may be *indirect*, that is, referenced in files that are, themselves, external references. When **Read External References** is set, all indirect references are automatically loaded and displayed.

You can also disable **Read External References** and load and display external references manually with **File/Reread Externals**.

To Automatically Display External References

Step 1. Choose **Info/Preferences**.

Step 2. Click the **Flight** tab.

Step 3. Set the **Read External References** checkbox. Now external references will automatically be loaded into memory and displayed when you open a database. If this checkbox is *not* set, external references are not displayed. This saves computer memory, especially for databases with large numbers of external references.

To Manually Display an External Reference



Step 1. Open the **Mode** toolbox.

Step 2. Click **External Reference Mode**.

Step 3. In the Hierarchy view, select the external reference nodes that you want to display.

Step 4. Choose **File/Reread Externals** to display the selected nodes in the Graphics view.

Note: Choosing **File/Reread Externals** with nothing selected loads and displays all external references.

Converting External References to Instances

Converting external references to instances copies the external database and saves the geometry with the current database as instances. When you convert an external reference to an instance, you have an instance of a geometry in the database, not the geometry itself. Instances use memory more efficiently than copies in a database (see "Instances" on page 368).

To Convert External References to Instances

Step 1. In the Hierarchy view, select one or more external reference nodes.

Step 2. Choose **File/Convert Externals**.

Listing and Editing External References

You can list all external references in your current database by choosing **Edit/List External References**. You can use the *List External References* window to search for specific references, including wild card (*) searches. You can also change the directory paths for one or more files rather than re-importing each reference model or replace one external reference with another.

To Search for an External Reference

Step 1. Choose **Edit/List External References**.

Step 2. In the *List External References* window, enter a search query string in the text field.

Creator allows the following in the query string:

A space ( ) indicates a second search condition and is interpreted as "or". To search for multiple conditions, insert a space between each entry. For example, apples.flt oranges.flt bananas.flt searches for the OpenFlight files apples, *or* oranges, *or* bananas.



***** indicates a wild card search. For example, c:\mydir*.flt will search for, and list, all the files having a .flt extension that are located the directory c:\mydir.

+ indicates additional text to search for and is interpreted as "and". For example, bldg+ofc will search for all files whose names contain both bldg *and* ofc.

– indicates that the search should exclude the condition and is interpreted as "not". For example, -red -orange will search for, and list, all files whose names *do not* contain the words red or orange.

Step 3. Click **Search Whole Path**, or **Search Filenames**

Search Whole Path searches for all external references in the directory path specified in the text field.

Search Filenames searches for external references whose file names satisfy the search conditions specified in the text field.

To Replace External References

Step 1. Choose **Edit/List External References**.

Step 2. In the *List External References* window, select the external reference to be replaced, and click **Replace Files**.

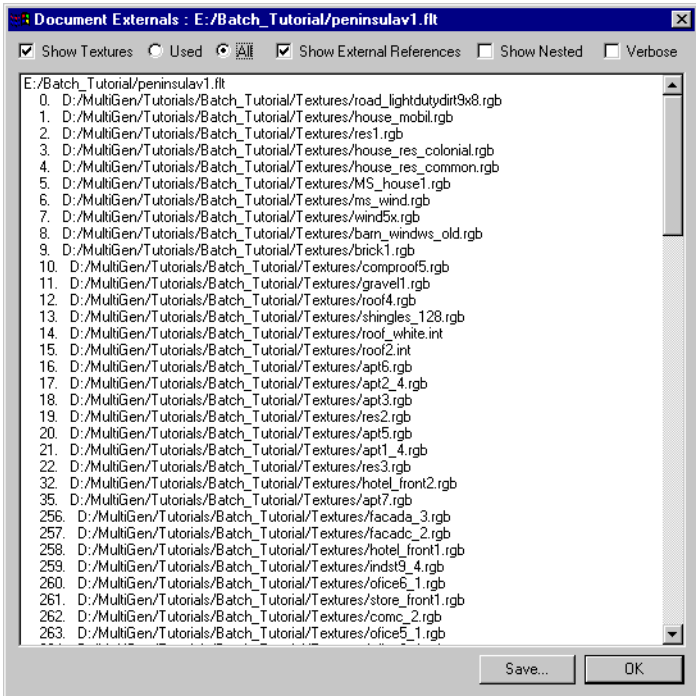
Step 3. In the *Select File to Replace* dialog box, browse and select the file to replace the current external reference and click **Save**.

Documenting External References

You can display a list of external references in your current database and write the contents of the list out to an ASCII text file for your project records. You can choose to display all external references, only models, or textures, or only the external references that are used. You cannot modify the list or any of the directory paths from the *Document Externals* window – the list is strictly for reference purposes. To modify the list of files or change the directory path, you must choose **Edit/List External References** and use the *List External References* window.

To document external references:

Step 1. Choose **Info/Document Externals**. The *Document Externals* window opens.



Step 2. Set the checkboxes to display either textures, external references, or both. The list updates immediately when you change a setting.

Step 3. To write the list to a text (*.txt) file, click **Save...**

Step 4. In the *Select File to Write* dialog box, choose the directory path, enter a file name in the **File name** field, and then click **Save**.

Step 5. Click **OK** to close the *Document Externals* window.

Document Externals Preferences

Preferences in the *Document Externals* window gives you several ways to display external references. When you have displayed the desired external references you can print the list to an ASCII text file.

Show Textures

When this checkbox is set, lists textures in the current database according to one of the following options:

- **Used** displays all the textures that have been assigned to polygons.
- **All** lists all the textures loaded into the Texture palette, whether or not they have been assigned to polygons.

Show External References

When this checkbox is set, lists every external reference in the current database. If the database contains multiple copies of the same external reference, each one will be listed.

Show Nested

When used with **Show Textures** and the **Used** option, lists the directory path and file name of all textures loaded in the Texture palette and assigned, as well as textures applied to the external references.

When used with **Show External References**, lists the directory path and file name of all external references in the database with an X next to it identifying it as an external reference.

Verbose

Adds headings to the texture and external reference information. When set with **Show Textures**, the headings are "Textures in Palette" (**All** option) and "Textures in Use" (**Used** option). When set with **Show External References**, each external reference and its directory path is listed under an "External Reference" heading.

Save

Saves the contents of the *Document Externals* window to an ASCII text file.

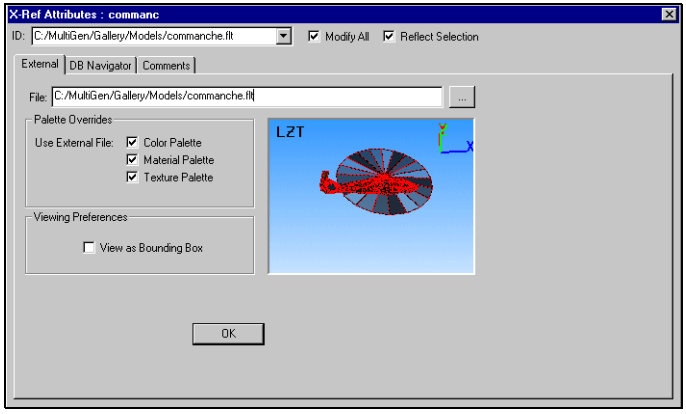
OK

Closes the *Document Externals* window.

External Reference Attributes

An *external reference* is a way to place geometry in one database by referencing the geometry of another database. The characteristics of an external reference, including the path to the externally-referenced model, are stored in the External Reference attributes.

To open the *X-Ref Attributes* window, double-click an external reference node or select an external reference and choose **Attributes/Modify Attributes**.



The *X-Ref Attributes* window has three panels: **External**, **DB Navigator**, and **Comments**.

External Panel (X-Ref Attributes Window)

Double-click an external reference node to display the *X-Ref Attributes* window. Select the **External Reference** tab to set the attributes in the **External Panel**.

Name is the path and filename of the external reference. This is set automatically when you create the external reference.

The **Use External File** checkboxes force the external reference to use its own Color, Material, and Texture palettes. If any of these checkboxes are cleared, the external reference is displayed using the corresponding palette in the current database.

View as Bounding Box displays external references as bounding boxes in the Graphics view, rather than as geometry. This reduces the amount of memory needed to display the external reference.

Billboards

Billboards are polygons that always face the eyepoint. Billboards are commonly used to represent symmetrical objects like poles or trees in the database. They can be used to simulate very detailed objects and still keep a very low polygon count.

You can make a two-dimensional billboard object look three-dimensional by mapping a texture to it. This is useful when creating an object that is difficult to model in 3D, like a tree, which can take many polygons to accurately represent. By mapping a texture of the tree to a single billboard polygon, the viewer sees a very detailed tree, which is always facing the eyepoint.

A billboard must be a single polygon, modeled around the *z* axis, within the *x*-*z* or *y*-*z* coordinate plane. It is usually a three-point or four-point polygon that rotates around the *z* axis to continually face the eyepoint.

Note: Billboard rotation is a realtime system effect and is not displayed in Creator.

"To Create a Billboard" on page 380

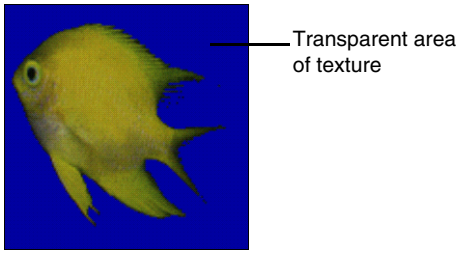
"Positioning a Billboard" on page 381

To Create a Billboard

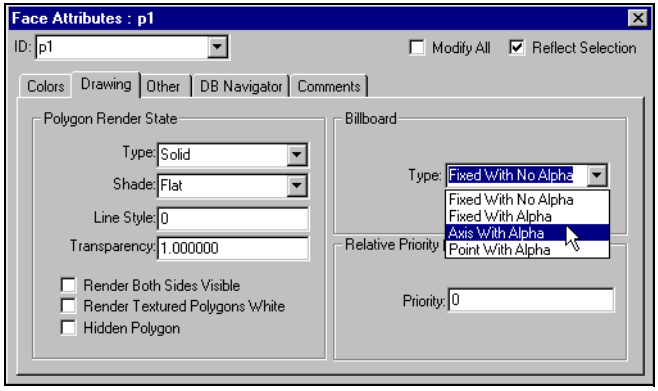
Step 1. Select a group or object node to be the parent of the billboard and click **Parent**. The parent node should not have any children other than the billboard.

Step 2. Construct a three-point or four-point polygon, in the *x*-*z* or *y*-*z* coordinate plane, around the *z* axis.

Step 3. If you want to apply a texture to the billboard, use the **Put Texture** tool from the **Map Texture** toolbox to apply a single repetition of a texture pattern to the billboard.



Step 4. Double-click the polygon to open its *Face Attributes* window.



Step 5. Click the **Drawing** tab. Make sure the **Render Both Sides Visible** face attribute is cleared.

Step 6. Select **Axis with Alpha** from the **Type** list to indicate the polygon rotates around the *z* axis to face the eyepoint.

Positioning a Billboard

A billboard is positioned by placing a transformation matrix on its parent. You can also instance a billboard to make it easy to use more than one copy of the billboard in the database.

To Position a Billboard Polygon Without Instancing

Step 1. Make sure the billboard polygon is attached to a group with no other children.

Step 2. Select the parent of the billboard polygon.



Step 3. Open the **Maneuver Tools** toolbox.



Step 4. Click **Insert Transformation Matrix**.

Step 5. Use the *Insert Matrix* dialog box to add a transformation to the selected node (see "Insert Transformation Matrix Tool" on page 157).

To Position a Billboard With Instancing

Step 1. Make sure the billboard polygon is attached to a group with no other children.

Step 2. Select the parent of the billboard group.

Step 3. Select the child of the parent group.

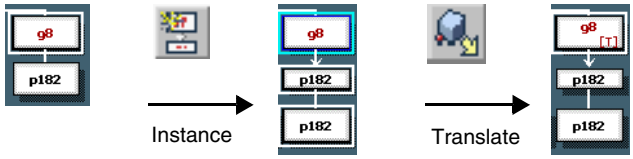


Step 4. Open the **Create Tools** toolbox.



Step 5. Click **Create Instance**.

Step 6. Click a **Maneuver Geometry** tool (such as **Translate**) to position the instanced billboard in the database. Creator automatically places a transformation matrix on the parent group (see "Instances" on page 368).



Instancing and Positioning a Billboard

Meshes

Meshes define geometric primitives that share attributes and vertices. Unlike individual polygons, which each have a unique set of attributes and vertices, meshes represent "sets" of related polygons, each sharing common attributes and vertices. Using a mesh, related polygons can be represented in a much more compact format. When you generate a mesh from selected faces, the faces are triangulated with shared edges and vertices in a t-strip. The runtime system efficiently processes this type of formation.

> **Note:** Many runtime systems automatically create meshes prior to running a database. If you are using one of these systems, creating meshes in Creator does not improve the runtime performance, but it does improve on-screen display during modeling.

There are two ways to create meshes in Creator: manually, by selecting a Group or Object node and using **Edit/Generate Mesh** to create a mesh from all the child faces; or automatically, during Polymesh terrain generation, by checking the **Make Mesh Nodes** option.

Some of Creator's features do not work on mesh. These features are:

- Subfaces
- Feature projection
- Trackplane setting
- Vertex or face mode selection

If you used **Edit/Generate Mesh** on polygons that are part of a model, you must decompose the Mesh node, discussed in "Decomposing a Mesh" on page 385, to enable any of the features in this list. Using either **Edit/Generate Mesh** on an existing terrain skin or using the **Make Mesh Nodes** option when creating a Polymesh terrain skin has special considerations.

Using Meshes With Terrain

If you convert terrain with Polymesh triangulation, you can choose to combine the Face nodes into Mesh nodes by setting the **Make Mesh Nodes** option in the Triangulate tab when Polymesh is selected as the algorithm (see "Polymesh Terrain Conversion" on page 1011). You cannot project features onto meshes, and all the resulting polygons

must have the same color. This limitation eliminates the ability to project individual features, to create batch terrain with features, to do batch post-projection of features onto the meshed terrain, and to use contour colors to apply different colors to different altitude ranges. For this reason, using **Make Mesh Nodes** to generate a mesh terrain is not recommended for any situations but high flyovers, where you are going to apply a texture with terrain and feature elements, such as a satellite photo, and you do not need underlying geometry.

If you want to use terrain meshes with features, you can project the features onto a polygonal terrain first, and then select terrain groups and use the **Generate Mesh** command to convert the polygons in the individual groups into meshes. (*Polygonal terrain* means a terrain that has been converted using a different algorithm such as Delaunay or TCT, or using Polymesh without setting the **Make Mesh Nodes** option.) Features that were projected flat will be eliminated, however, because meshes cannot have subfaces. Features that were not projected flat, and features that were cut in through TCT triangulation, will be unaffected because the vertices in the underlying terrain will not move when the mesh is created. No new features can be added after the mesh is formed, however.

Using Generate Mesh

When you generate a mesh, you select a parent node, such as a Group or Object node. All Face nodes with the same face attributes below the selected Group or Object node combine into a single mesh. You create separate meshes if you select multiple Group or Object nodes,or if at least one attribute, such as the color and texture, of Face nodes are different from each other.

> **Note:** Any DOF, LOD, Light Source, or Subface nodes within a group that you select will be deleted when you generate a Mesh node, since they cannot be represented by the mesh structure.

To generate a mesh:

Step 1. Select either a Group node or Object node. You cannot select individual Face nodes to combine into a Mesh node.

Step 2. Select **Generate Mesh** from the **Edit** menu. The individual Face nodes with the same attributes are combined into a Mesh node. All other nodes below the parent Group or Object node are discarded.

Editing a Mesh

You can edit Mesh nodes and modify their attributes, such as the face color or texture, which are the same as face attributes. When you modify attributes in the Mesh Attributes window, the changes are applied to all of the faces in the mesh. To modify attributes for a specific face in the mesh, you must decompose the Mesh node into individual Face nodes, as discussed in "Decomposing a Mesh" on page 385. See "Mesh Attributes" on page 852.

Step 1. To select faces in the Graphics view and edit them, select **Mesh** mode from either the **Mode** toolbox or from the Toolbar.

Step 2. Select the faces in the mesh that you want to modify.

Step 3. Select **Attributes/Modify Attributes** to modify attributes in the Mesh Attributes window. You can also double-click on the mesh in the Graphics view to display this window.

Note: You can double-click the Mesh node in the database hierarchy to edit the attributes in the Mesh Attributes window.

Decomposing a Mesh

You can decompose a Mesh node to separate the faces into Face nodes. The faces remain triangulated. This is useful when you want to perform functions such as adding or deleting faces, resetting the tracking plane, adding subfaces, or changing attributes for a specific face in the mesh, for example.

When you decompose a Mesh node, the Face nodes are not in their original configuration in the database hierarchy. Face nodes are reorganized for the t-strip formation in the Mesh node. Object nodes are also not saved in the Mesh node. To see all of the nodes in their original configuration, you must select **Undo: Generate Mesh** from the **Edit** menu.

Step 1. Select either the parent node or the Mesh node. You can also select the mesh in the Graphics view in Mesh mode.

Step 2. Select **Decompose Mesh** from the **Edit** menu to separate the mesh node into individual Face nodes.

13 Materials

Materials provide special effects that simulate the light reflecting characteristics of substances like wood, plastic, and metal when a database is in **Gouraud**, **Lit**, or **Lit Gouraud** lighting mode. Material properties are defined in a Material palette and can be selectively applied to faces in the database.

The following sections describe how to create material properties and apply them to faces in your database. They also describe how to use the Material palette menus to modify existing properties, create new ones, and manage the properties in your palette to create a custom palette that can be loaded whenever those properties are needed.

Material Preferences

You can set Creator preferences to automatically apply the current material property to newly-created geometry. To do this, set the Editing preference **Auto Apply Current Material**.

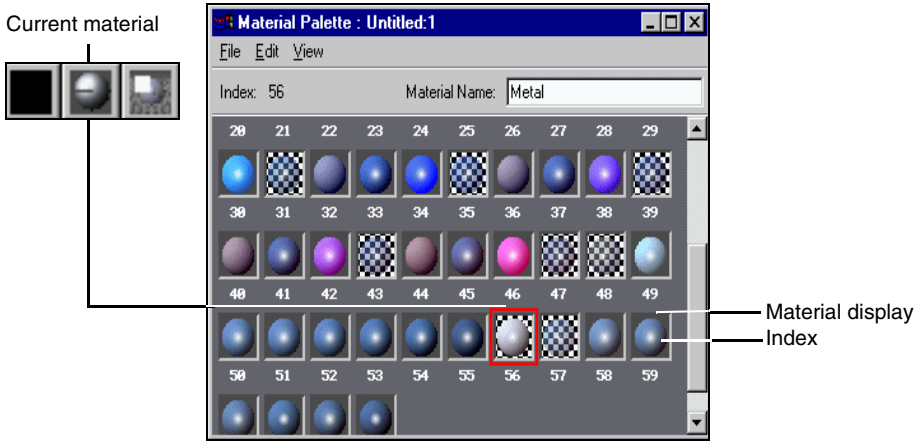
Flight preferences also contains the setting **Modulate Current Color with Material in Material Palette**. When this checkbox is set, the color of the current material is blended with the face color.

The Material Palette

Each OpenFlight database has its own Material palette containing a set of material definitions that can be applied to faces.

To open the Material palette for the active database (the default palette if no database is open), click **Current Material** in the Toolbar or choose **Palettes/Material Palette**. (Double-clicking **Current Material** opens the *Modify Material* window.) Creator's default Material palette contains 64 of the most commonly-used material property definitions.

The Material palette displays a sample of each of the material types defined in the palette, called a **Material Display**. If a material property has been named, the **Material Name** of the current material displays in the field at the top of the palette. Underneath each Material Display is the palette **Index** number of the material.



The *current* material is the material selected in the Material palette at any given time. To select a material, open the Material palette and click one of the material display spheres.

To close the Material palette. click **File/Close** or click the close window button.

Material Palette Menus

The Material palette menus are used to load and save material palettes to disk, work with material definitions, and specify how you want the material definitions to appear in the Material palette.

The **File** menu loads saved material palette files, saves both new and existing material palettes to files, and closes the Material palette.

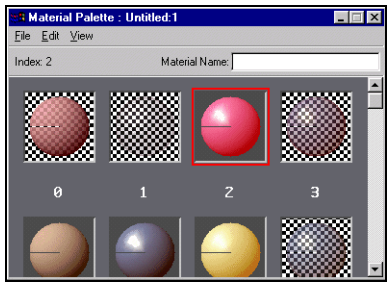
Using the **Edit** menu, you can create new material definitions, cut, copy, paste, delete and modify existing material definitions, and purge the current material palette of any definitions that are not assigned in the database. The **Cut**, **Copy**, and **Paste** functions can be used between different material palettes.

The **View** menu contains two selections that control palette display and current material retrieval.

- **Thumbnail** displays a thumbnail view of the current *Material* palette. This is the default setting. To restore the palette to its original size, choose **Thumbnail** again.



Thumbnail On



Thumbnail Off

- When Thumbnail view is not set, only eight materials are visible at any time in a Material palette. Choosing **Fit on Select** immediately displays the current material.

Creating New Material Definitions

Although the default Material palette contains a number of material definitions that you can modify to suit your needs, you can create as many new definitions as you wish.

To Create a New Material Definition

From the Material palette **Edit** menu, choose **New**. A new definition appears in the Material palette as the Current material.

To Copy a Material Definition From Another Palette

Step 1. From the Material palette **File** menu, choose **Load Palette**.

Step 2. Select the Material palette that contains the material definition to be copied, and then click **Open**.

Step 3. Select the material definition to be copied, and choose **Edit/Copy**.

Step 4. In the same Material palette, choose **File/Load Palette**, and load the target Material palette.

Step 5. Choose **Edit/New** to create a new material definition in the target Material palette.

Step 6. Choose **Edit/Paste** to copy the material properties into the newly created material definition.

To Delete a Material Definition From the Material Palette

Step 1. Highlight the material definition to be deleted.

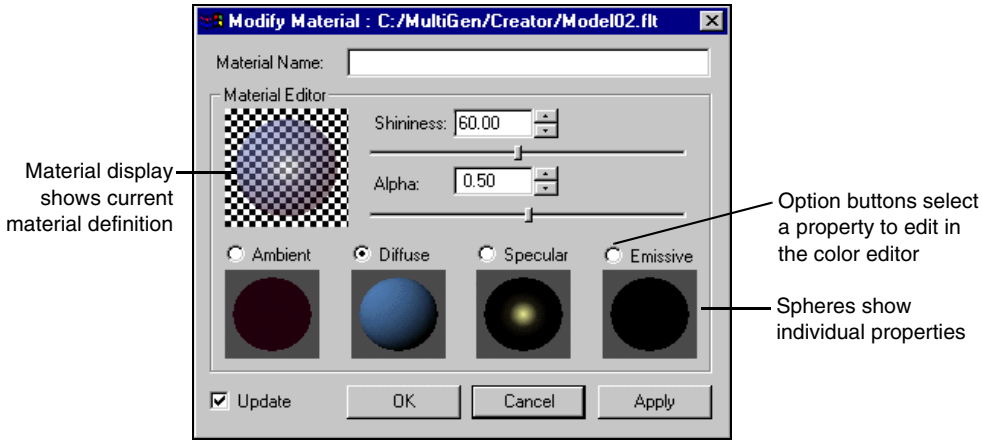
Step 2. From the palette's **Edit** menu, choose **Delete**.

To Delete All Unused Material Definitions From a Palette

From the Material palette's **Edit** menu, choose **Delete Unused**. Any material definitions that have not been assigned will be deleted.

Modifying Material Properties

Edit/Edit Material opens the *Modify Material* window (shown below). Each material definition is made up of an optional **Material Name** plus a set of properties that define its color, shininess, and transparency. You can also load a material into the *Modify Material* window by double-clicking it in the Material palette or by double-clicking **Current Material** in the Toolbar. The *Modify Material* window always contains the current material.



Ambient, **Diffuse**, **Specular,** and **Emissive** material properties are defined in terms of color. In the *Modify Material* window, each of these properties is displayed, individually, on the sphere below its name. The option buttons above these names load properties, one at a time, into the color editor. **Shininess** and **Alpha** material properties are scalar values and are set with slider controls.

If the **Update** checkbox is set, the material properties applied to geometry in the database instantly update whenever they are modified.

To edit a material definition's properties:

Step 1. Select the material in the Material palette.

Step 2. Choose **Edit/Edit Material**, or double-click the **Material Display**.

Step 3. Type a name in the **Material Name** field, if desired. The field is limited to 11 characters.

Step 4. Change the **Shininess** property by moving the slider, using the arrow buttons to increase or decrease the value, or entering a value in the field. Note how the material changes as these values change. **Shininess** defines the material's glossiness.

Step 5. Change the **Alpha**, or transparency, property by moving the slider, using the arrow buttons to increase or decrease the value, or entering a value in the field.

Step 6. Click the **Ambient**, property to open the *Color palette* window. Select a color and click **OK** to close the window. **Ambient** is the light the material reflects from other objects.

Step 7. Repeat Step 6. for the **Diffuse**, **Specular** and **Emissive** properties.

Step 8. Click **OK** to accept the material definition.

Modify Material Parameters

Modify Material parameters control the material definition's visual characteristics.

Shininess

The mirror-like quality of the material's surface. Shininess is indicated by the size of the highlight on the material ball. A high shininess value produces a small highlight, giving the material a metallic appearance.

Alpha

The amount of transparency in the material. A value of 0 defines a transparent material; a value of 1 defines an opaque material. On computers that display alpha, the checkerboard background fades in or out of view behind the sphere as you change this value.

Note: Pay careful attention to database traversal order when using material transparency. A transparent face blends with the face that is lower than it in the traversal order. (See "Database Traversal Order" on page 688.)

Ambient

The amount and color of light that the material reflects from other objects in the scene. This affects surfaces that are not illuminated directly. Face color interacts with the ambient lighting effect.

Diffuse

The amount and color of light that the material reflects in all directions, regardless of the eyepoint position. This results in a flat reflection that is brightest where light strikes the surface at a 90° angle. Face color interacts with the diffuse lighting effect.

Specular

The brightness and color of the highlight, or glare. This is brightest when the eyepoint is located at the light's angle of reflection. Metal often imparts a reflected color to the specular highlight. Specular light is directly affected by the **Shininess** property.

Emissive

Defines the amount and color of non-reflective light the material produces. In other words, the color of the material. This parameter is used for objects that glow.

Saving and Loading Material Palettes

Saving a custom set of materials simplifies the use of similar material palettes in similar databases. Custom Material palettes can be saved and loaded into other databases where the material characteristics are similar. All previous material definitions are discarded when you load a new Material palette, and the new materials automatically replace old materials everywhere they have been applied in the database.

You can specify a Material palette that loads automatically whenever Creator launches. This Material palette must be saved as `flt1.materials`. Creator looks for this file first in the folder defined by the MPI_CREATOR_RESOURCEPATH variable, and then in the working directory. The working directory for Windows is the directory where Creator is installed, and for IRIX it is the directory where Creator was launched. If `flt1.materials` is not found, then Creator loads a standard default Material palette of 64 material definitions.

To Save a Material Palette

Step 1. Click **File/Save Palette**. The *Write Material Palette File* dialog box opens.

Step 2. Enter a name in the **File Name** field of the and click **Save**.

To Create a Default Material Palette

The following steps create a default material palette that will be loaded each time Creator launches.

Step 1. Edit the definitions in a Material palette, using the options in the **Edit** menu, until you are satisfied with its contents.

Step 2. Click **File/Save Palette**. The *Write Material Palette* dialog box opens.

Step 3. Set **Save In** to the folder specified by the MPI_CREATOR_RESOURCEPATH environment variable, if it was set. Otherwise, set **Save In** to the working directory.

Note: For Windows the working directory is the directory where Creator is installed, and for IRIX, it is the directory where Creator was launched.

Step 4. In the **File Name** field type `flt1,` and click **Save**. The Material palette will be saved with the suffix `.materials`.

Step 5. Restart Creator to activate the new default palette.

To Load a Material Palette

Step 1. Choose **File/Load Palette** to open the *Open Material Palette File* dialog box.

Step 2. Select a Material palette to load and click **Open**.

Applying Material Properties

Material properties are applied to faces using the **Insert Material** tool in the **Properties** toolbox, or by choosing **Edit/Insert Material**. The properties of the current material are applied. Any changes made to the material definition's properties are automatically updated in the faces to which it was applied.

When the Editing preference **Auto Apply Current Material** is set, the current material is automatically applied to each newly created face in the database. If this preference is clear, a material index of –1 (no material) is assigned to each new face.If you have set the Flight preference **Modulate Current Color with Material in Material Palette**, the color of the current material is blended with the face color.

You can only view material properties in **Lit**, **Gouraud**, or **Lit Gouraud** shading mode because these properties absorb or reflect light to varying degrees. The default shading mode is **Flat** because it requires the least system resources. With **Flat** shading, faces

appear in their assigned colors and the effects of light sources and material properties are ignored.

Note: You must **Calculate Shading** (Ctrl+G) and choose **Lit**, **Gouraud**, or **Lit Gouraud** shading mode (see "Database Shading" on page 620) to see the effects of the material properties applied to your models. (Likewise, in wireframe mode, the edges will display the colors of materials, only if your shading is set to a mode which displays material properties.)

Insert Material Tool



Insert Material applies the current material to all selected faces.

To Insert a Material



Step 1. In **Face** mode, select the target face or faces.

Step 2. Choose **Insert Material** in the **Properties** toolbox, or choose **Edit/Insert Material**.



To Remove a Material From a Set of Faces

Step 1. Select the faces.

Step 2. Choose **Edit/Remove Material**. This sets the Material index value attribute of each selected face to -1.

Get Material From Face Tool



Changes the current material to one that is assigned to the selected face.

To Change the Current Material Setting



Step 1. Set the modeling mode to **Face**.

Step 2. Select the face where the material has been applied.



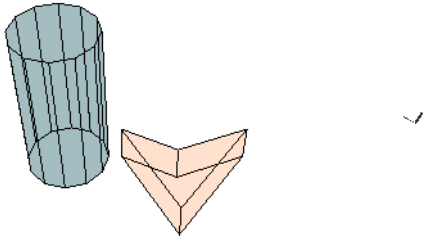
Step 3. Click **Get Material From Face** in the **Properties** toolbox. The Current Material button in the Toolbar changes to the new material.

Alternatively, you can open the Material palette and select a new material.

Exercise: Apply Material Properties

Material characteristics add interest and lifelike qualities to inanimate shapes. They make things appear opaque, transparent, shiny, or dull. In the following steps you will select and apply a material definition to one of the shapes in your database. You will then modify the material definition and observe the changes on your database.

Step 1. Open the file `Model02.flt` in the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory.



Note: If you did not install the exercise files when you installed Creator, you can open this file directly from the Creator installation CD.

Step 2. In Object mode, select the cylinder.



Step 3. Click **Current Material** in the Toolbar to open the Material palette.



Step 4. Select a material that looks shiny (29 for example) from the palette.

Step 5. Open the **Properties** Toolbox and click **Insert Material** to apply the material to the cylinder.



Step 6. Select the triangular shaped model.

Step 7. Select a semi-transparent model (such as 47) from the Material palette. Close the palette.

Step 8. Click **Insert Material** to apply the material property to the model.

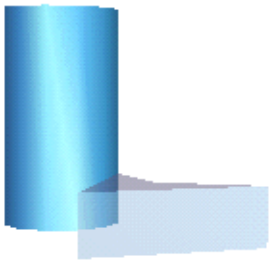


Step 9. In the View Panel set the **Drawing Mode** to **Solid**.

Note: At this point, you won't be able to see the material properties you applied to your models. You must Calculate Shading first.

Step 10. Choose **Attributes/Calculate Shading** (Ctrl+G).

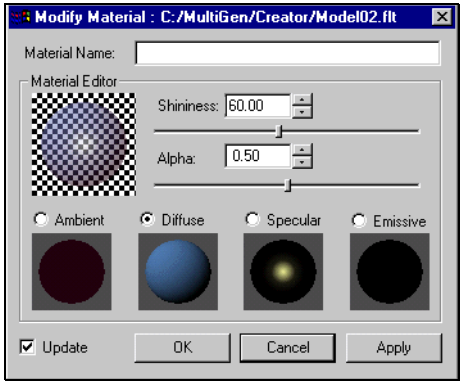
Step 11. In the *Calculate Shading* dialog box, set **Shading Model** to **Lit**, and click **OK**. You should now see the material properties you applied to your models.



Step 12. Rotate the eyepoint to view the material effects from different sides. Note the transparency of the triangle.

Step 13. Double-click **Current Material** in the Toolbar and drag the *Modify Material* dialog box to the side so that you can change the material's characteristics and view the effect on your model simultaneously.





Step 14. Drag the sliders for **Shininess** and **Alpha** values and view the results on your model in the Graphics view.

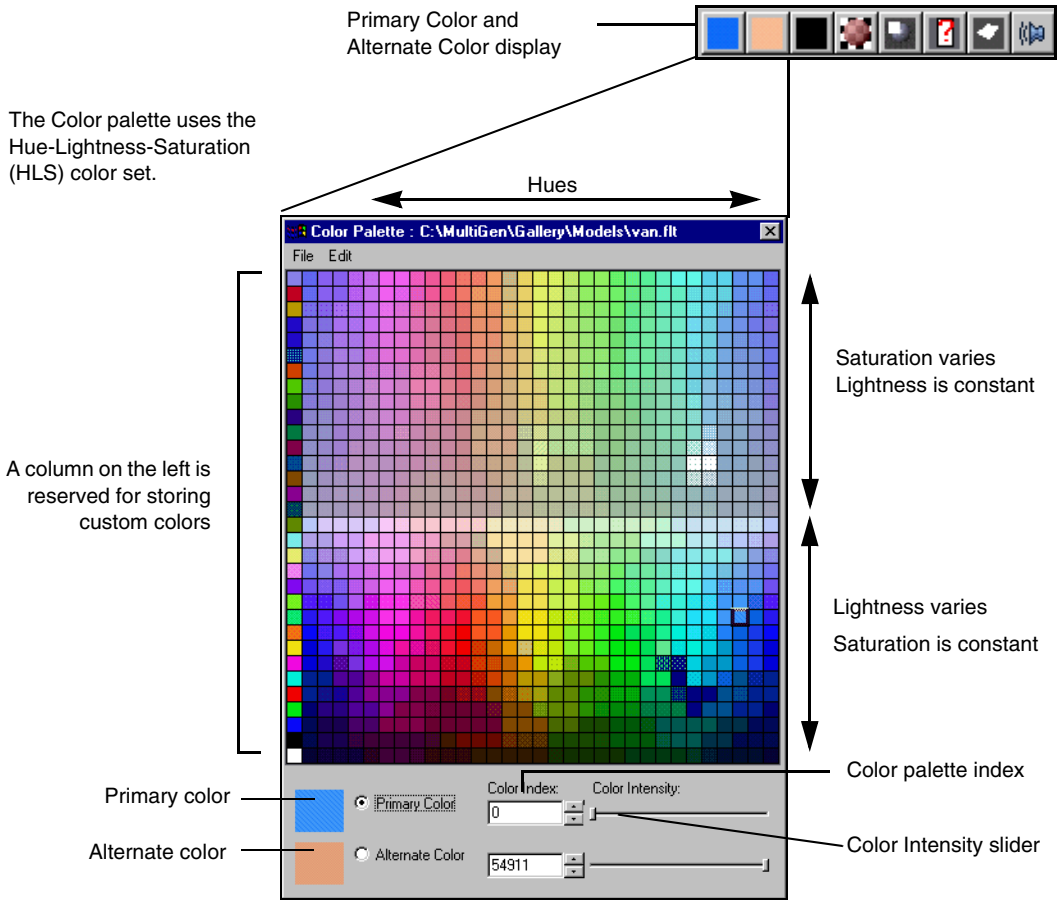
Step 15. Save your database, if you want. Your material definitions will be saved along with it.

14 Color

Color is a face attribute in the database. Each database uses a *Color palette* containing colors that can be applied to faces. In a flat-shaded Graphics view, each face is colored (or *filled*) when viewed from the *front* (the side from which vertex numbering appears to increase in counterclockwise order).

Color Palette Window

To open the *Color Palette* window, choose **Palette/Color** (Ctrl+P).



Each color in the Color palette has a hue, lightness, and saturation (HLS) component. The hues are aligned horizontally across the palette. The upper half of the palette shows variations in each hue's saturation in graduated steps, and the lightness component is held constant for all steps at mid range (about .70). The lower half of the palette shows saturation held constant at maximum richness, but with lightness varied at each step from maximum (white) to minimum (black).

Primary Color and Alternate Color

The *Color* palette controls the **Primary Color** and **Alternate Color** of the top database. You select the primary color and alternate color in the *Color Palette* window.

The *primary color* determines the color of each new face in the database; *alternate color* displays intensity texture patterns. (See "Color Palette Window" on page 399.)

Note: The *Color Palette* window displays the Color palette, primary color, and alternate color of the top Database window. If you switch to another Database window while the *Color Palette* window is open, its contents update accordingly.

To change the primary color or alternate color:

Step 1. Open the *Color Palette* window by clicking either **Primary Color** or **Alternate Color** on the Toolbar.

Note: A single click opens the Color palette. A double click opens the color editor.

Step 2. Select either the **Primary Color** or the **Alternate Color** option button.

Step 3. Select a new color by clicking it in the Color palette.

Step 4. To adjust color intensity, drag the **Color Intensity** slider left or right until the shade you want appears in the display. (See "Selecting Color Intensities" on page 407.)

Current Color Buttons

The Current color buttons on the Toolbar show the currently select primary and alternate color. When you change a primary or alternate color in the Color palette the respective Current color button updates automatically.

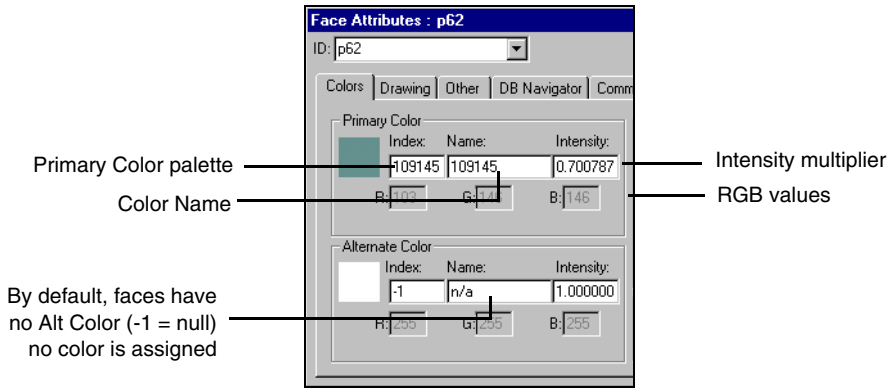
Color Attributes

New face color attributes are set automatically, based on the primary color. Color attributes can later be changed in the *Face Attributes* window or by assigning a different

color to the face with the **Insert Color** tool. (See "Inserting and Getting Color from Face" on page 409.)

To edit face color attributes, select a face in the Graphics view (or a face node in the Hierarchy view) and choose **Attributes/Modify Attributes** (Ctrl+=).

See "Face Attributes: Colors Panel" on page 841 for more information.



Color Names

Color names can be assigned to individual color entries, enabling you to specify different colors from multiple Color palettes for use in a single database. For example, you can create two palettes that reflect different seasons, and then assign the same color name (wheat, for example) to green in the spring palette and to brown in the fall palette. Color names can be associated with Color palette entries but not with intensity values.

> **Note:** Make sure your realtime system supports the use of color names before creating them in a database.

When you assign a color to a face, its color **Name** is listed along with the color **Index** as a face attribute. If the color **Index** and color **Name** don't match (as a result of a change in one Color palette, for example), the color **Name** takes precedence.

Also note the following:

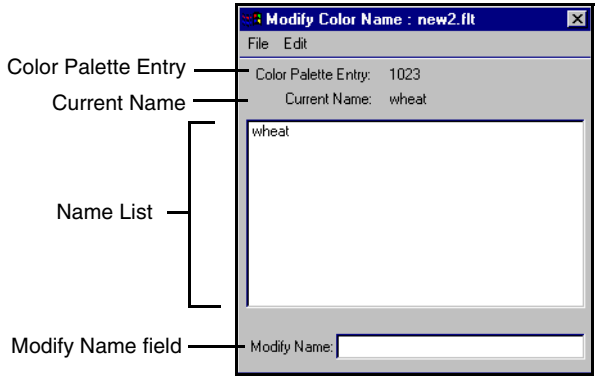
- Each color name is tied to a fixed position in the Color palette.
- All color names in a given Color palette must be unique.
- An entry in the Color palette can have more than one name.
- Only the full-intensity colors displayed in the Color palette can be named. Other individual shades of intensity cannot be named.
- Editing a color in the palette may affect the usefulness of its name. For example, changing a color named “wheat” to the color blue turns every face assigned that name and that Color palette index blue.
- Only the current color name (highlighted) is assigned to faces whenever the corresponding color is assigned.

Creating Color Names

To create a color name:

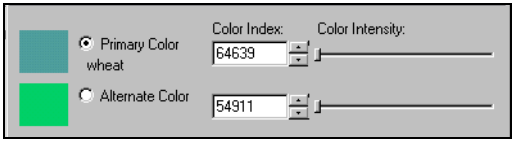
Step 1. Select the color in the Color palette.

Step 2. Choose **Edit/Edit Color Name** in the *Color Palette* window to open the *Modify Color Name* window.



Step 3. Enter the color name in the **Modify Name** field.

Step 4. Choose **Edit/Add Color Name**. This name is transferred to the **Name List** and becomes the **Current Name**. It also appears next to the **Primary Color** box in the *Color Palette* window.



Note: As long as the *Modify Color Name* window remains open, the **Color Palette Entry** and **Current Name** automatically reflect the Current color in the Color palette.

Modify Color Name Menu

The **Edit** menu in the *Modify Color Name* window has the following commands:

Set Current Color

Changes the **Current Name** to the selected name in the **Name List**.

Add Color Name

Adds the name in the **Modify Name** field to the **Name List**.

Delete Color Name

Removes the selected name from the **Name List**.

Replace Color Name

Replaces the selected name in the **Name List** with the contents of the **Modify Name** field.

Search

Searches the Color palette for the name in the **Modify Name** field, and loads that color into the *Modify Color Name* window.

Modify Color Name Controls

Color Palette Entry

Shows the entry number (0 to 1023) of the selected color in the Color palette. This refers to each entry (which represents 128 intensities) of the palette, not individual Color palette indices.

Current Name

Shows the name that appears when you select this color in the *Color Palette* window. The Current color name is assigned to faces whenever the Current color is assigned.

Name List

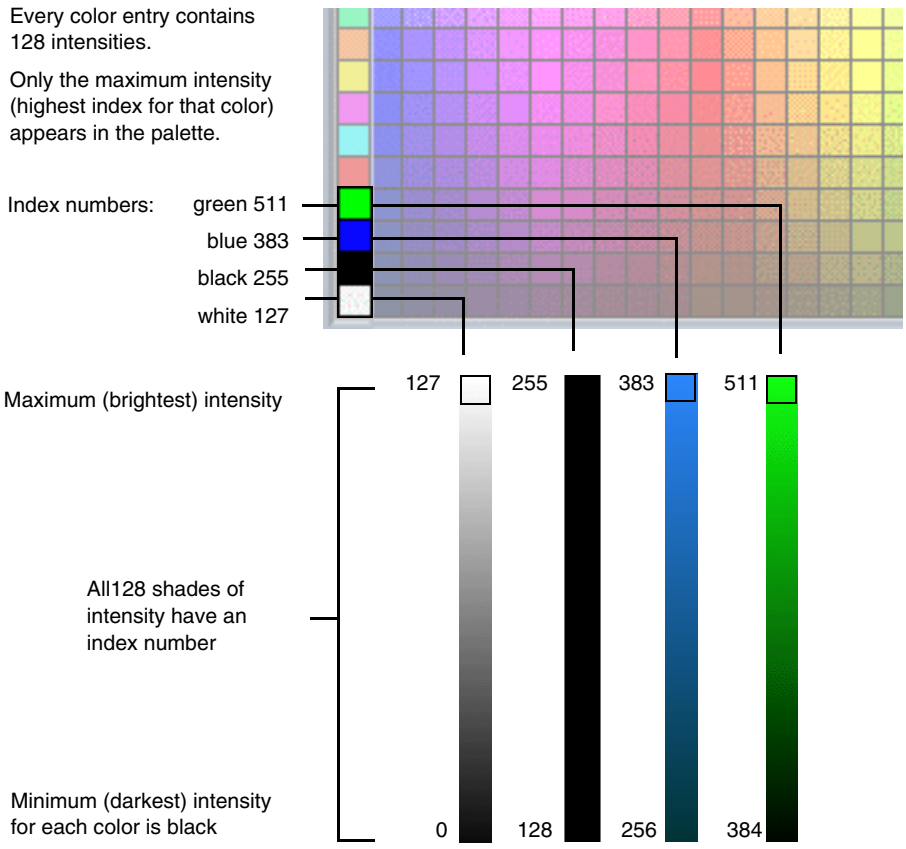
Shows all names assigned to the Current color in the Color palette. Each face with this color attribute **Name** references this entry in the palette.

Modify Name

Specifies a string to add to the **Name List** or use in a search.

Color Palette Index

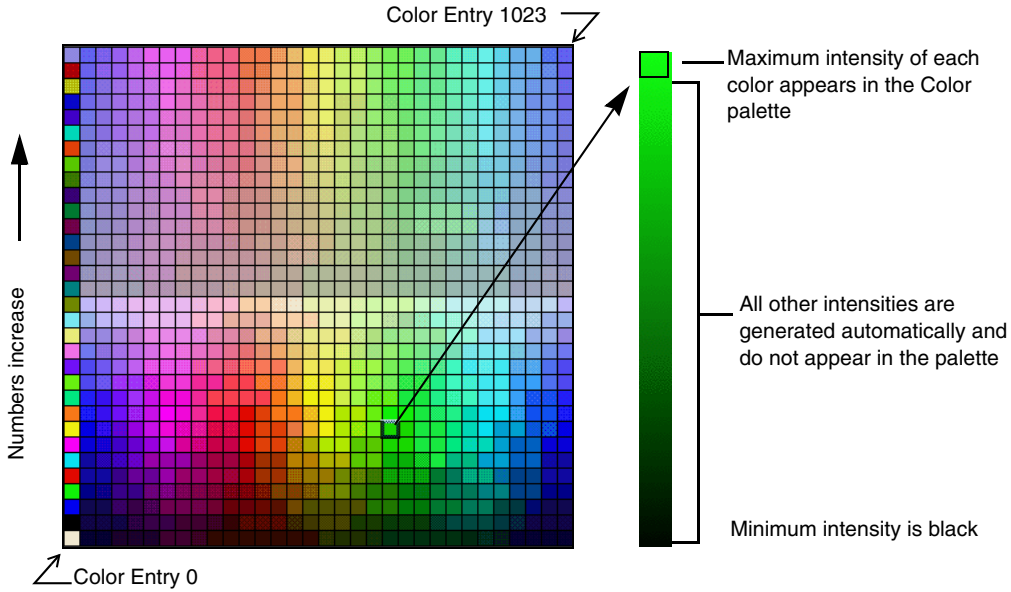
Color palette index is a number (0 to 131071) that uniquely identifies each color *and* each intensity in the database. Index numbers of color entries increase as you move up from the lower-left corner of the Color palette. Although each entry also contains 128 intensities, only the maximum intensity colors are displayed in the palette. Color entries are therefor 128 indices apart; the darkest intensity of each color has the lowest index, and the brightest intensity has the highest index.



Color Intensity

The Color palette contains 1024 color entries (or color chips) that can be edited independently. Each editable color entry is further divided into a band of 128 shades (*intensities*), with only the brightest shade (*maximum intensity*) appearing in the palette. A total of 131,072 colors (1024 entries x 128 intensities) are available.

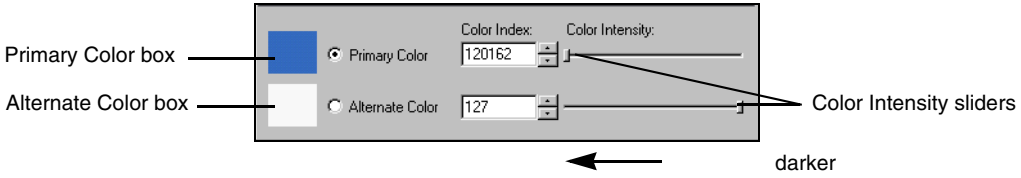
Note: Gouraud shading is dependent on the 128 shades associated with each color applied to a face. (See “Shading Preferences” on page 621.)



Selecting Color Intensities

The intensity of the primary color or alternate color is controlled by dragging the corresponding **Color Intensity** slider left or right in the *Color Palette* window.

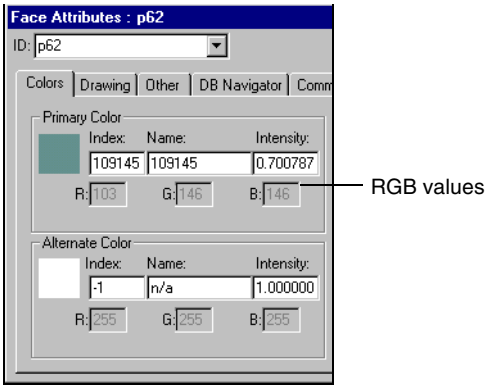
As you drag the slider to the left, the color displayed in the **Primary Color** box gets darker. This darker shade has a lower color index number than the full intensity shade of the color. The original full intensity shade still appears in the Color palette.



Note: The color intensity applied to each individual face can be further controlled by editing the **Intensity** face attribute. This value, with a default of 1.0, has a range of 0.0 to 1.0 that lets you reduce the maximum intensity of the color **Index** face attribute. (See "Color Attributes" on page 401.)

RGB Mode

There are two color modes available in Creator: Index color and RGB color. In Index mode, the color of each face is determined by its **HLS** (Hue, Lightness and Saturation) attribute values. In RGB mode, the color of each face is determined by its **RGB** attribute values.



The color mode you choose depends on your realtime system. Typically, Index color is used on systems that are limited in the number of colors they can display, while RGB color is used on systems that tolerate an arbitrary number of colors.

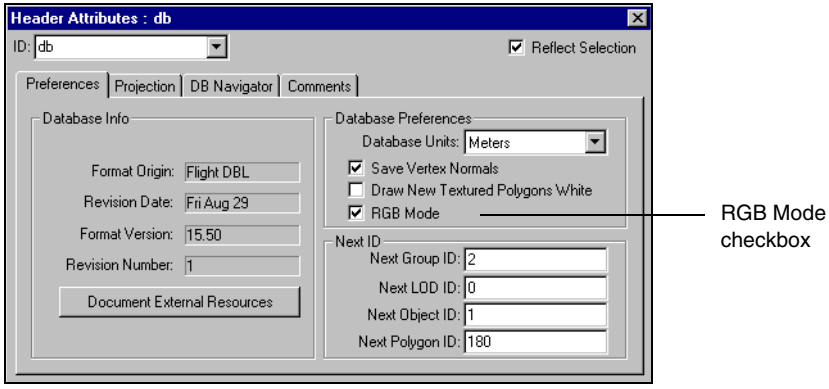
The number of possible colors in each mode is:

- Index color (HLS): 1024 (indices) x 128 (intensities) = 131,072 possible colors.
- RGB color: 256 (red) x 256 (green) x 256 (blue) = 16,777,216 possible colors.

Changing Color Mode

Index mode is the default color mode in Creator.

To switch from Index mode to RGB mode, open the Database Header attributes and set the **RGB Mode** checkbox. When you do this, existing indexed-color faces are automatically converted to their RGB equivalents.



To switch from RGB mode to Index mode, open the Database Header attributes and clear the **RGB Mode** checkbox. When you do this, Creator searches the Color palette for the closest match to the RGB color assigned to the face, and assigns the index of the matching color to the face.

Note: Switching back and forth between RGB mode and Index mode is not recommended because the two color models do not match one another exactly.

Assigning a Color in RGB Mode

Use the primary color, just as in Index mode. Note that the indices are irrelevant to RGB mode - the palette is only significant in that it gives you a convenient selection of RGB values.

Inserting and Getting Color from Face

The **Insert Color** and **Get Color from Face** tools can change the color and sample the color of existing faces in the database.

Insert Color Tool



The current Primary color is automatically assigned to each new face that is created. You use **Insert Color** (Ctrl+R) to apply the current Primary color to existing faces.

To Use Insert Color

Step 1. Select the face(s).

Step 2. Choose **Edit/Insert Color** (Ctrl+R), or click **Insert Color** in the **Properties** toolbox.



Press Alt while clicking **Insert Color** to assign *only* the alternate color into the selected faces. Press Ctrl+Alt while clicking **Insert Color** to assign *both* the current primary *and* alternate colors to the selected faces.

If shading has been calculated for a face, you will not see the color change until you calculate the shading again (see "Database Shading" on page 620.

Get Color From Face Tool



Get Color From Face sets the primary color to match the color of the face you select with this tool.

To Set the Primary Color From a Face

Step 1. Select the face(s). If more than one face is selected, **Get Color From Face** uses the color of the first face in the Select list.



Step 2. Click **Get Color From Face** in the **Properties** toolbox.

Press Alt while clicking **Get Color From Face** to set *only* the alternate color in the Color palette from the **Alt Color** attribute of the selected face.

Press Ctrl+Alt while clicking **Get Color From Face** to set both the primary and alternate colors in the Color palette from the corresponding attributes of the selected face.

Note: You can also select a face and then choose **Edit/Color From Face**.

Editing Colors

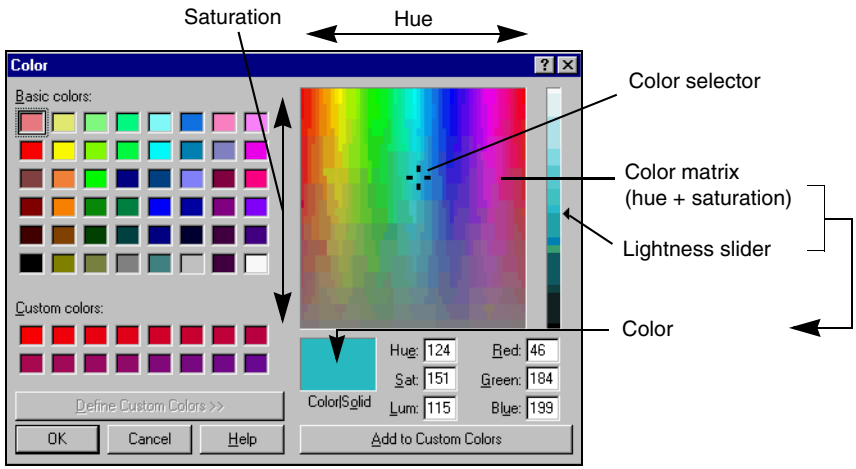
Color palette entries can also be edited or replaced. Changing the definition of one of the entries in the palette changes all 128 shades of that color. Colors displayed in the

color matrix can be defined as Hue, Lightness, Saturation (HSL) and Red, Green, Blue (RGB) colors. For Windows, Creator uses the standard Windows *Color* dialog box to do this. For IRIX, Creator uses a color picker.

Editing Colors in Windows

Step 1. Double-click an existing color entry in the Color Palette, or choose **Edit/Edit** in the *Color Palette* window. The Windows Color dialog box opens and the selected color appears in the **Color|Solid** box.

When a selected color appears in the Windows Color dialog box, it is displayed as a combination of the selected color and of the lightness slider setting.



Note: The **Lum** field in this case represents "lightness". The **Lightness** slider values (0 - 240) define one of three components of the HLS color map. It is not related to the **Color Intensity** slider or the index values (0 - 127) in the Color palette.

Step 2. Drag the **Color Selector** on the **Color Matrix** or click a specific color with the left mouse button. The selected color appears in the **Color|Solid** box.

Note: You can also enter specific HLS and RGB values to define a color.

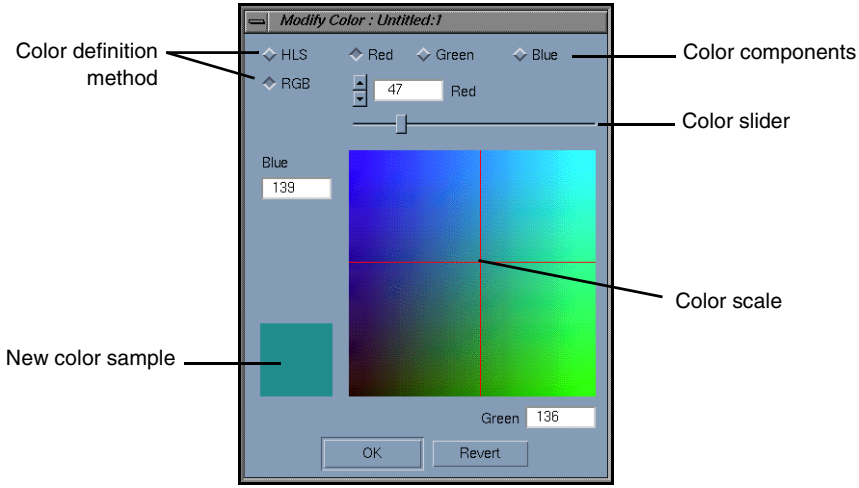
Step 3. Drag the lightness slider arrow to adjust the lightness setting of the color, if desired.

Step 4. Click **OK**. The *Color* dialog box closes and the newly defined color replaces the color entry previously selected in the Color palette.

Note: **Basic Colors** and **Custom Colors** are default system colors and are not directly mapped to default Creator Color palette entries. Only the current color in the **Color|Solid** box will be transferred to the selected color entry in the *Color Palette* window.

Editing Colors in IRIX

Step 1. Double-click an existing color entry in the Color palette, or choose **Edit/Edit** in the *Color Palette* window. The *Modify Color* dialog box opens and the selected color appears in a color sample on the left side of the dialog box.



Step 2. Set the color definition method by setting either the **HLS** (hue, lightness, and saturation) or **RGB** (red, green, blue) option button.

Step 3. Select a color component (**Hue**/**Lightness**/**Saturation** or **Red**/**Green**/**Blue**) and use the slider to edit it, or enter a value in the color component field. The color sample updates automatically.

Step 4. Drag the crosshairs of the Color scale to edit the other two color components, or enter values in their respective fields.

Step 5. When the color sample looks the way you want it, click **OK**.

Resetting the Color Palette

After editing or replacing color entries using the color editor, you may want to reset the Creator Color palette to its original HLS color state.

To reset the Color palette to the default color map, choose **Edit/Make HLS Map** in the *Color Palette* window.

Saving and Loading Color Palettes

New databases load a default HLS color palette. However, you can develop a custom Color palette, and save it in a file (*.color) that can be loaded by other similar databases. For example, a group of terrain databases might all use a Color palette with a large selection of earth tones.

You can also create a custom Color palette that loads automatically whenever Creator launches. This Color palette must be saved as flt1.color. Creator looks for this file first in the folder defined by the MPI_CREATOR_RESOURCEPATH variable, and then in the working directory. The working directory for Windows is the directory where Creator is installed, and for IRIX it is the directory where Creator was launched. If flt1.color is not found, then Creator loads the standard HLS color palette.

Saving a Color Palette

The following steps save the Color palette of the top database in a file.

Step 1. Choose **File/Save Palette** from the *Color Palette* window. The *Write Color Palette File* dialog box opens.

Step 2. Type a file name in the **File Name** field, and click **Save**.

Loading a Color Palette

The following steps load a Color palette file and assign it to the top database.

Step 1. Choose **File/Load Palette** in the *Color Palette* window. The *Open Color Palette File* dialog box opens and lists all files in the current directory having a .color suffix.

Step 2. Select the Color palette file to be loaded or type the file name in the **File Name** field, and click **Open**.

Creating a Default Color Palette

The following steps create a default Color palette that will be loaded each time Creator launches.

Step 1. Create a custom Color palette by editing color intensities and names, adding colors to, and deleting colors from the palette.

Step 2. Click **File/Save Palette**. The *Write Color Palette File* dialog box opens.

Step 3. Set **Save In** to the folder specified by the MPI_CREATOR_RESOURCEPATH environment variable, if it was set. Otherwise, set **Save In** to the working directory.

Step 4. In the **File Name** field type flt1, and click **Save**. The Color palette will be saved with a .color suffix.

Step 5. Restart Creator to activate the new default palette.

Color Matching

Several modeling functions search the Color palette to select colors for use in the database. For example, the Gouraud shading process assigns vertex color by selecting the closest match between a calculated vertex color and the colors available in the palette. The Color palette is also searched when a file is imported from another database format or from a previous version of OpenFlight if the Flight preference **Use Attribute Index Matching** is set.

The **Intensities for RGB Matching** Flight preference (range 1 to 128) sets the number of levels of intensity that are searched in each entry of the Color palette. If this value is one, the color search is restricted to the 1024 maximum-intensity colors displayed in the palette. Each additional increment permits the next darker intensity level in the palette to be searched. Restricting this value makes searching faster, but results in a less precise matching of colors.

If you set **Use Face Color Intensities** (in the *Calculate Shading* dialog box) when performing a Gouraud shading, the color table is not searched and, therefore, **Intensities for RGB Matching** has no effect. (See "Shading Preferences" on page 621.)

Infrared Color

An infrared (IR) view of a database is one in which the appearance of each face is determined by a combination of its **Infrared** and **IR Material** attributes. The **Infrared** face attribute contains an integer code (range 0 to 4095), the meaning of which is defined by the realtime system. The **IR Material** face attribute contains the Material palette index of a material to be used in Infrared viewing mode.

Infrared Codes

To make it easier to build a database containing infrared codes, Creator contains several functions that let you preview infrared. Keep in mind, these functions are designed to help you model and do not affect the way infrared is displayed on the realtime system.

In general, Creator interprets each IR code linearly as a grayscale intensity value, with 0 representing black and 4095 representing white. You can override this default with controls in the IR palette.

Infrared View



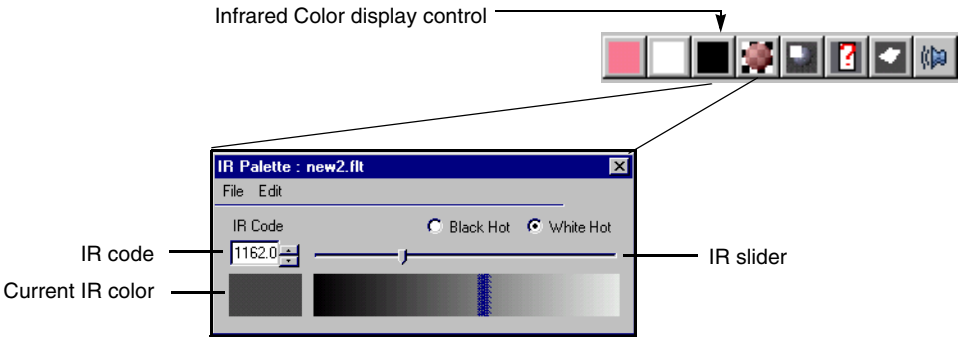
To preview the infrared assignments in a database, turn on **Infrared View** in the *View* panel.

Note: Because the default value of the **Infrared** face attribute is "0," faces that have not been assigned an explicit infrared value initially will appear black.

Infrared Palette

The infrared palette lets you set up an IR code for assignment to faces. It also controls the way Creator interprets each **Infrared** face attribute in Infrared viewing mode. Remember, the final display of the database in realtime is controlled by the interpretation the realtime system places on the **Infrared** and **IR Material** face attributes.

To open the IR palette, choose **IR** from the **Palettes** menu or double-click **Infrared Color** in the Toolbar.



IR Palette Controls

IR Code

Determines the current (selected) IR code in the IR palette. To change the current IR code, edit the code number or drag the slider in the *IR Palette* window.

Current IR Color

Displays the color associated with the current IR code. Unless a debug color has been assigned, this is a combination of the IR color and the intensity value associated with the current IR code.

Black Hot/White Hot

Controls whether the maximum IR code is associated with white or black.

IR Palette Edit Menu Functions

Set IR Color

Assigns the primary color to the IR palette. The intensity values defined by each IR code are applied to this color for display in Creator. This color defaults to white each time you start Creator.

Default IR Color

Resets the IR color to white. This causes all IR codes that are not assigned debug colors to display in grayscale.

Set Debug IR Color

Assigns the primary color to the current IR code. Assigning different debug colors to IR codes makes it easy to identify faces with each code in the database. Debug colors are saved with the preferences file.

Clear Debug Colors

Clears all debug colors, so faces display using a combination of the **IR Color** and their intensity values.

Infrared Modeling Tools

The infrared modeling tools enable you to insert infrared color into faces and to get infrared color from faces already containing infrared color.

Insert IR Color Tool



Assigns the current IR code to all selected faces.

To Insert IR Color

To insert the current IR code into the selected face(s):



Step 1. Open the **Properties** toolbox in the toolbox caddy.

Step 2. Click **Insert IR Color** to apply infrared color to the selected face(s).

Get IR Color From Face Tool



Sets the current IR code from the Infrared attribute of a selected face.

To Get the IR Color From a Face

To get the IR code from the **Infrared** attribute of a selected face:



Step 1. Open the **Properties** toolbox.

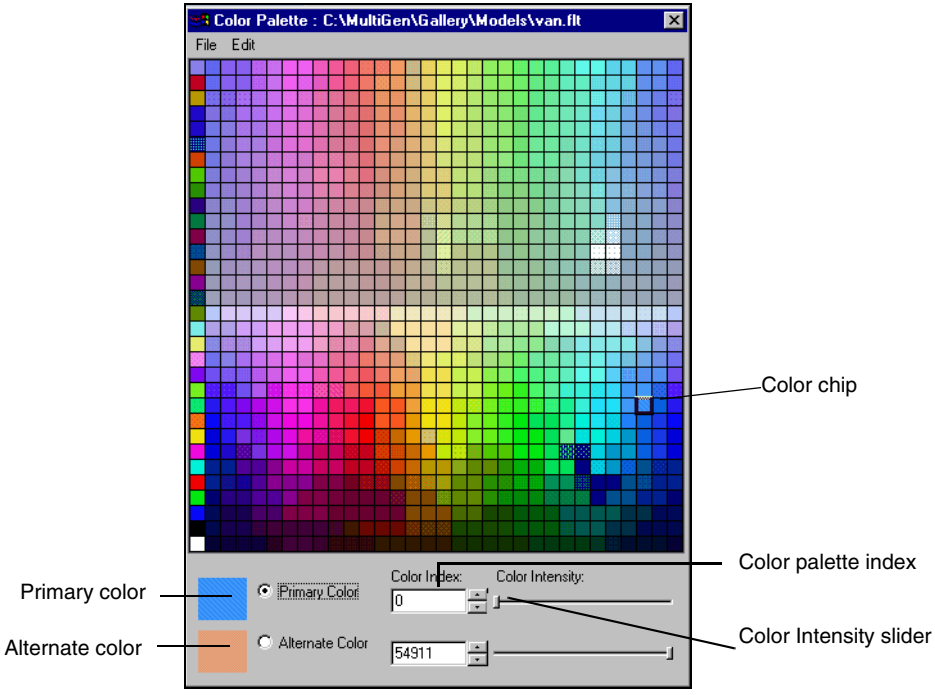
Step 2. Click **Get IR Color From Face**.

Note: If more than one face is selected, the IR code is set using the first face on the Select list.

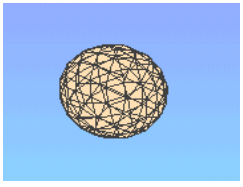
Exercise: Adding Color to a Model

In the following exercise you will select a color, name it, and insert it into a model. Before you begin, you may want to print this topic to make it easier to follow the instructions as you work with Creator.

Step 1. Choose **Palettes/Color** (Ctrl+P) or click **Primary Color** in the Toolbar. The *Color Palette* window opens.



Step 2. In the Graphics view, create a simple object, such as a sphere. The primary color is automatically applied to the object.



Step 3. Click a different color chip in the Color palette.

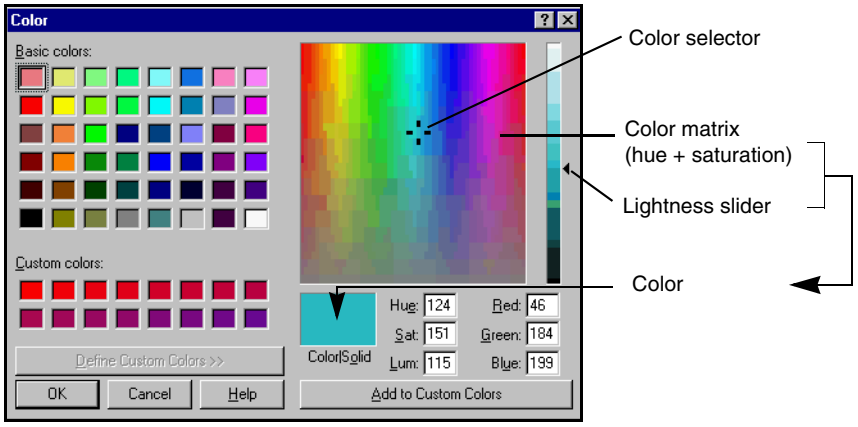
Step 4. Move the **Color Intensity** slider to create a darker shade.



Step 5. Open the **Properties** toolbox in the toolbox caddy.

Step 6. Click **Insert Color** to apply the color to the object.

Step 7. Double-click a different color chip in the Color palette to open the color editor.



Step 8. Move the lightness slider to lighten or darken the color.

Step 9. Click **OK**. The edited color replaces the selected color in the Color palette.



Step 10. Click **Insert Color** in the **Properties** toolbox to apply the color to the object.

15 Texture

Textures are bitmapped images that are mapped onto polygons to give your database a photo-realistic appearance without increasing the polygon count. Textures can be produced, for example, by scanning and editing photographs.

Textures are composed of color data elements, called *texels*. (Contrast this to a *pixel*, which is one unit of the on-screen display.) A texel's size depends on the resolution of the texture and the texture's scaled size. When the 2D texture is applied to a 3D model, the u and v coordinates of the texture image are mapped to the x, y, and z coordinates of the model.

Texels are made up of color channels. Typically, a channel is 8 bits. The color channels are grayscale, red, green, blue, and alpha. Creator supports the following image formats:

- INT - Intensity (one channel: 8-bit grayscale)
- INT-A - Intensity-alpha (two channels: 8-bit grayscale, plus 8 bits of alpha)
- RGB (three channels: 8 bits each of red, green, and blue)
- RGB-A - RGB-alpha (four channels: 8 bits each of RGB plus 8 bits of alpha).
- 16-bit Intensity
- PIX (Alias)
- BMP
- CT (clip texture)
- GIF
- IFF/ILBM
- JPEG
- PCX
- PNG
- PPM
- RAS
- SUN
- TIFF
- TGA
- RLA (Wavefront)

For PIX (Alias) through RLA (Wavefront) in this list, Creator evaluates the number of color channels the image file contains and expresses the file type attribute as Intensity, Intensity-alpha, RGB, or RGB-alpha. It does not, however, convert the file format.

To apply a texture pattern to a model in Creator, you must first load the texture file into a Texture palette and select it as the current texture. Then you select the face or faces to which the pattern will be applied, and apply the pattern using one of the Creator **Property** or **Texture** tools.

You can modify a texture's attributes to use filters that control any blockiness, or *pixellation*, in pixel- to-texel appearance when the texture is applied and displayed.

To organize textures, Creator provides a Texture palette, which stores references to all the textures in your database and displays a thumbnail view of each texture for easy reference. To apply textures to faces in the database, Creator includes a set of texture mapping tools and a Texture Mapping palette.

The following sections describe how to use the Texture palette and Texture Mapping palette, how to map texture patterns onto faces in the database, and how to modify a texture pattern and its mapping.

Texture Palettes

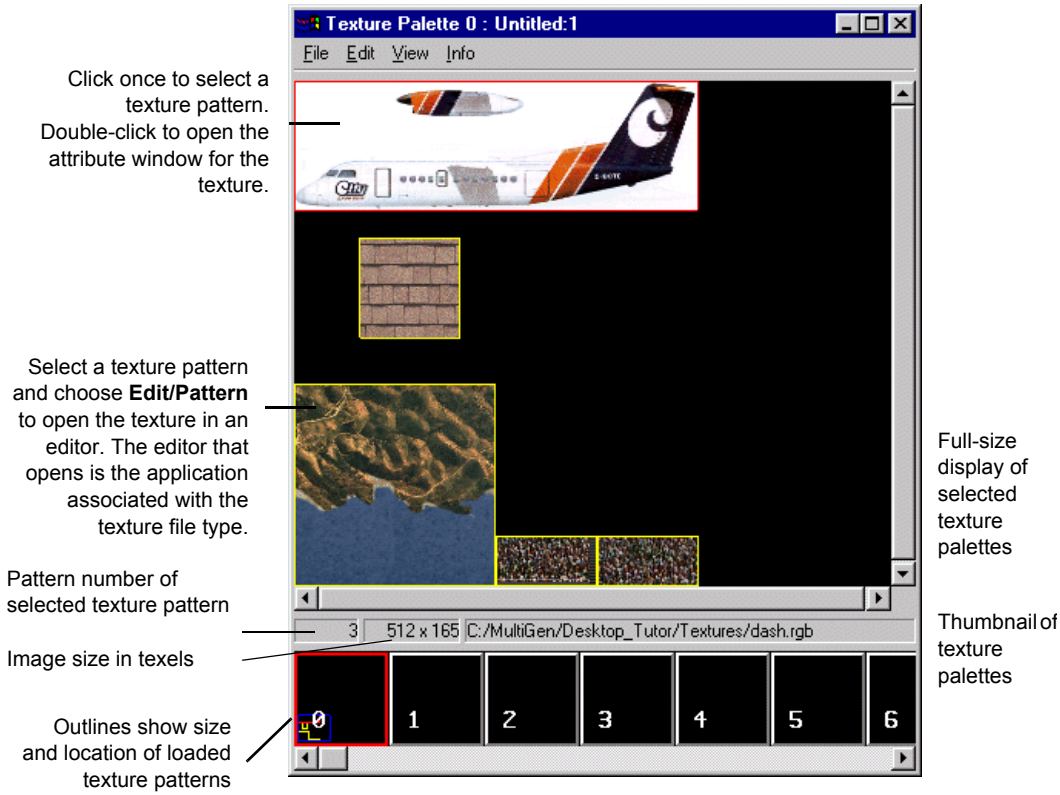
Before texture patterns can be applied, they must be loaded into one of the database Texture palettes. Each database has a set of 128 Texture palettes, each holding up to 256 individual texture patterns, with a maximum of 2048 x 2048 texels per palette. These palettes are stored in the file `default.txt`. When you open the *Texture Palette* window, a thumbnail of the individual Texture palettes are displayed in the scrollable area at the bottom of the window. The miniature of the currently-selected palette is outlined in red and appears at its full size in the upper portion of the window.

To open the *Texture Palette* window, choose **Palettes/Texture** or click the **Current Texture** button on the Toolbar.

Each texture in the Texture palette is assigned a unique **Pattern** number, which identifies it for use in the database. The pattern number is assigned to the **Texture** attribute of each face to which the texture pattern is applied.

When multiple databases are open, the *Texture Palette* window displays the palettes belonging to the current database. If no database is open, the window contains an empty set of palettes you can load and save for future use.

Note: You can also assign frequently-used textures to Quick palettes during a modeling session (see "Using Quick Palettes" on page 358).



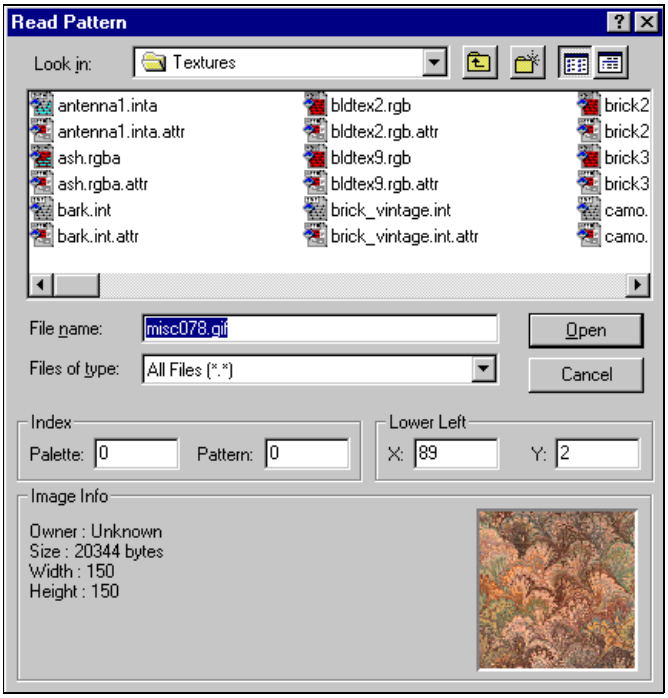
Texture tools work with the selected texture pattern in the Texture palette, also referred to as the *current* texture pattern. To select a texture pattern, click it. The current texture pattern is outlined in red.

The pattern number, size in texels, and full directory path of the current texture pattern are displayed in the Texture palette.

To Add Textures to the Palette

To load a texture pattern into the selected Texture palette:

Step 1. Double-click an empty area of the Texture palette to open the *Read Pattern* dialog box.



Note: You can also choose **Read Pattern** from the Texture palette **File** menu, and specify an offset, in texels, from the lower-left corner of the palette to prevent the new texture from overlapping previously-loaded textures. (Overlapping has no effect on textures; it just makes them less visible in the palette.)

Step 2. Select a texture pattern from the list in the *Read Pattern* dialog box. The lower-right corner displays a thumbnail image of the selected texture.

You can select more than one texture pattern from the list using Shift+click (or click and drag), and load them into the Texture palette at the same time.

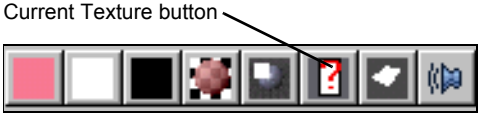
Step 3. You can choose the palette and pattern numbers that will be assigned to an individual texture pattern at the time you open it. To do this, enter a number in the **Palette** and/or **Pattern** fields.

Step 4. Click **Open** to place the selected texture pattern in the Texture palette. If you specified a palette and pattern number in the *Read Pattern* dialog box, the texture pattern will be placed in the appropriate palette and the designated pattern number will display in the Texture palette.

Note: If no attribute file exists for the selected pattern, Creator creates one using default texture attributes and assigns the original image file name with the additional extension `.attr` (as in `mytexture.rgb.attr`). For more information on default texture attributes, see "Texture Variables and Preferences" on page 541.

Current Texture

The **Current Texture** button on the Toolbar displays a thumbnail of the texture pattern that is currently selected in the Texture palette. When no texture is selected in the Texture palette, the **Current Texture** button displays a question mark.



Clicking the **Current Texture** button opens the Texture palette. Double-clicking the **Current Texture** button launches the editing application that is associated with the current texture's file type.

The Texture Palette Menus

The Texture palette menu has options to load textures, load and save texture palette files, manage texture patterns, control the appearance of the Texture palette, and get information about textures loaded in the Texture palette.

The Texture Palette Edit Menu

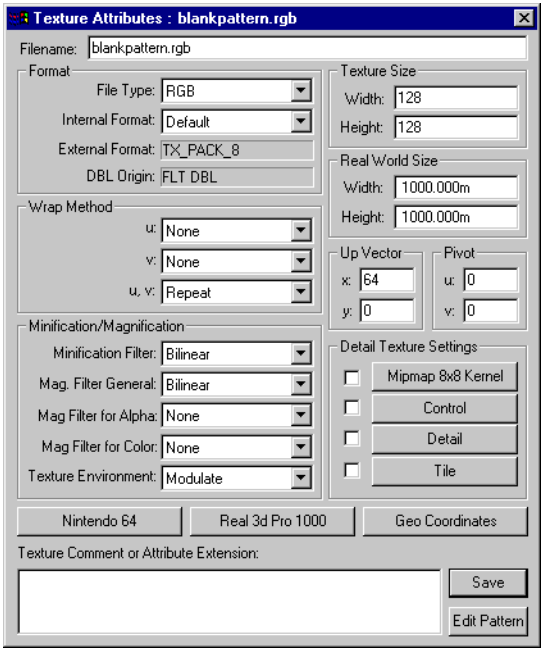
The Texture palette **Edit** menu has the following commands, which you use to create, edit, move, and delete texture patterns.

Edit Pattern

Edit Pattern in the Texture palette **Edit** menu launches an image editor, such as Adobe Photoshop, GIMP,or the Creator Texture Editor, that is associated with the current texture pattern and loads the current texture pattern into the image editor. (See "Editing Texture Patterns" on page 545). You can also launch the image editor by double-clicking the current texture on the Toolbar.

Blank Pattern

Blank Pattern in the Texture palette **Edit** menu creates a new (empty) RGB, RGB-Alpha, Intensity, or Intensity-Alpha texture pattern, and opens its *Texture Attributes* window. In this window, you can edit the name, size, and other attributes of the new pattern, click the **Edit Pattern** button to launch the appropriate editor, and click the **Save** button to save the new pattern. For more information on texture attributes, see "Texture Attributes" on page 523.

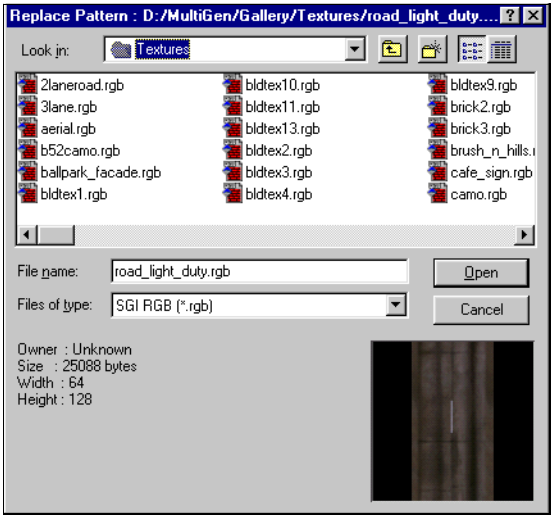


Reload

Reloads the current texture pattern, current palette, or all the palettes from the original location. If changes are made outside of Creator to a texture pattern or Texture palette, you can reload the latest version.

Replace Pattern

Replace Pattern in the Texture palette **Edit** menu replaces the current texture pattern with the texture pattern you choose in the *Replace Pattern* dialog box.



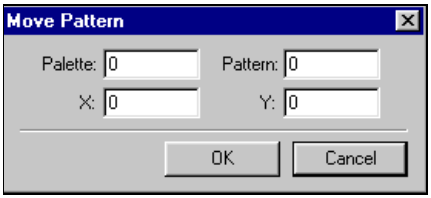
The new pattern automatically replaces the old pattern everywhere it is applied in the database. The mapping ***u*** and ***v*** settings do not change. All indices remain the same.

This command is identical to the **Replace File** option in the *List Textures* dialog box. (See "Replace File" on page 432.)

Move Pattern

Move Pattern in the Texture palette **Edit** menu moves the current texture pattern to the Texture palette location you specify in the *Move Pattern* dialog box. (You can also move a pattern to a new location by dragging it.)

You can move the texture to one of the other 126 palettes within the Texture palette file, give the texture a different pattern number within the same or a new palette, and/or adjust the X and Y offset values to change the position in the palette where the texture is displayed.



The fields in the *Move Pattern* dialog box correspond to similar entries in a default Texture palette file. See "Texture Palette Files" on page 441.

Move Pattern does not change the way the selected texture is mapped in the database, but it can change the pattern number assigned to the texture. If this happens, Creator searches the database and updates the pattern number in the Face attributes of all polygons to which the texture is assigned. If the texture is assigned to a large number of polygons (in a terrain database, for example), this operation might take some time. In addition, the operation is stored in memory for **Undo**.

Modify Attributes

Opens the *Texture Attributes* window of the current texture file for editing (see "Texture Attributes" on page 523).

Delete Pattern

Deletes the current texture pattern from the Texture palette and removes all references to the texture from the database. The actual texture pattern is stored in a separate file and is not affected by this function.

If the deleted pattern was mapped to faces in the database, the texture pattern in the Face attributes of each of these faces is set to -1 (no texture) when you use **Delete Pattern**. If you want to delete the current texture pattern from these faces and replace it with a new pattern, use **Replace Pattern** instead (see "Replace Pattern" on page 427).

Remove Unused

Deletes all texture patterns from the Texture palette that have not been applied to polygons in the database. This does not affect the texture files; only their references in the Texture palette are deleted.

Clear Palette

Deletes all texture patterns from the current Texture palette and removes all references to the corresponding textures from the database. This does not affect the texture files; only their database references are deleted.

Clear All Palettes

Deletes all texture patterns from all 127 palettes in the Texture palette file, and removes all texture references from the database.

The Texture Palette View Menu

The Texture palette **View** menu has the following commands, which you use to view texture patterns in the Texture palette.

Fit on Select

Moves the selected texture pattern and adjusts its resolution to fit in view.

Center Select

Moves the selected texture pattern in view at the current resolution. **Center Select** is the same as adjusting the view with the scroll bar.

View All

Displays all of the textures at the highest resolution in which all textures fit in the Texture palette.

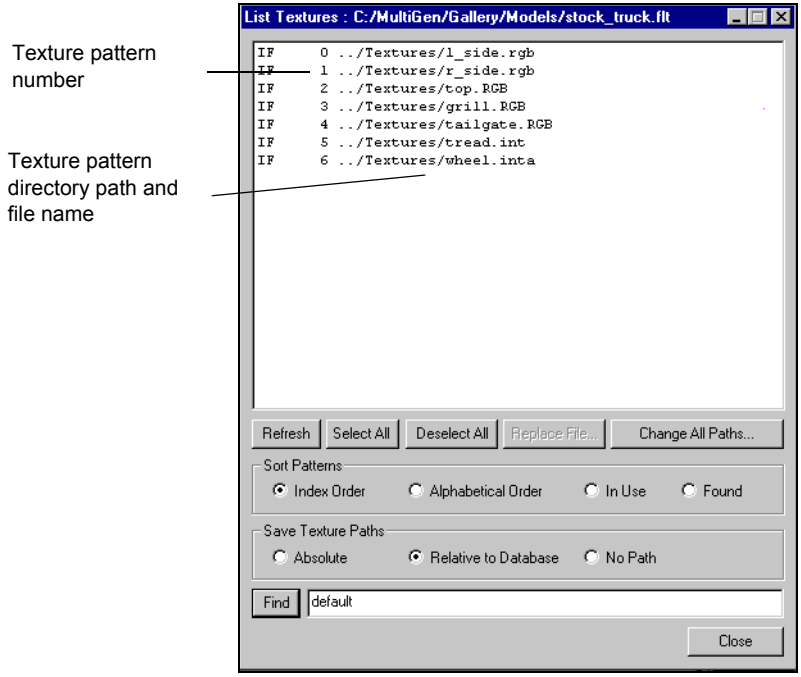
> **Note:** You can press the Shift key and right-click to manually zoom in and out of the Texture palette view.

The Texture Palette Info Menu

Functions in the Texture palette **Info** menu help manage texture patterns in the Texture palettes. The **List Textures** and **Statistics** dialog boxes update dynamically as you add or delete texture patterns from the palette; you may find it convenient to keep these dialog boxes open on the desktop while changing the Texture palette.

List Textures

List Textures in the Texture palette **Info** menu opens the *List Textures* dialog box, which displays the directory path and file name of every texture pattern belonging to the current database, along with controls that you can use to change texture pattern directory paths.



Refresh

Restores the original texture list.

Select All

Selects all the textures in the list.

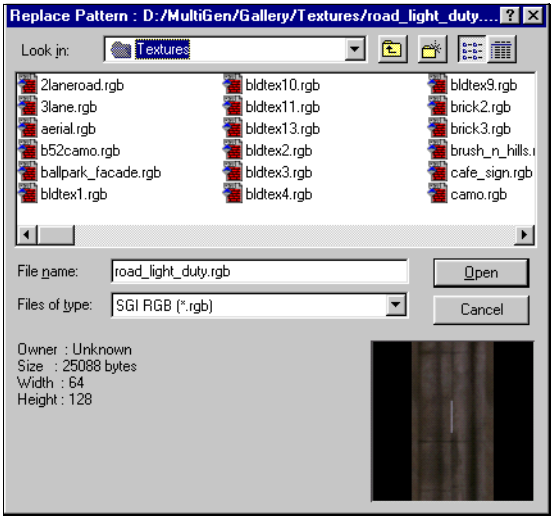
Deselect All

Deselects all the textures in the list.

Replace File

Replaces the selected texture pattern with a new texture. This function is identical to the **Edit/Replace Pattern** menu item.

Click **Replace File** in the *List Textures* dialog box to open the *Replace Pattern* dialog box. The texture pattern you choose in this dialog box replaces the selected texture pattern in the *List Textures* dialog box. The new pattern automatically replaces the old pattern everywhere it is applied in the database.

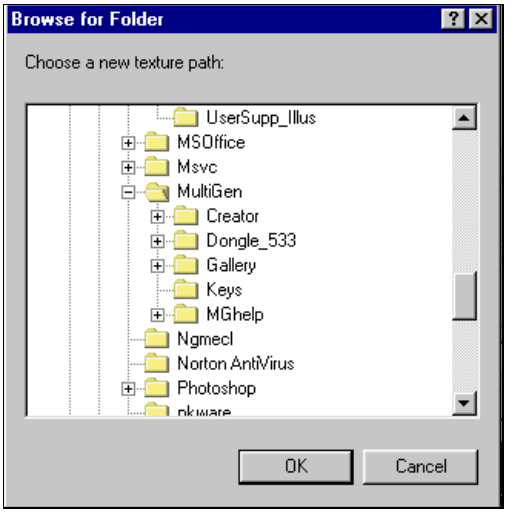


Change All Paths

When clicked, opens a browse window to select a new directory path for all the texture patterns listed.

Change All Paths

Click **Change All Paths** in the *List Textures* dialog box to open the *Browse for Folder* window. Select a new directory path for the texture files, and then click **OK**. Every texture pattern listed will be updated with the new directory path and folder.



Sort Patterns

Displays the list of texture patterns in either alphabetical order, in order of increasing texture pattern indices, or according to whether or not the textures patterns have been mapped. Any unused textures display the letters NU, for “not used”, before the directory path.

Find

Among the listed texture files, searches for the string you enter in the **Find** field.

Save Texture Paths

Selects the form of the directory path for each file in the database.

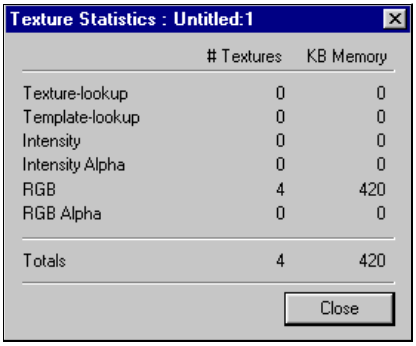
- **Absolute** specifies a fixed directory path to each texture file.
- **No Path** specifies only the texture file. Select **No Path** if you want to use the path that is set for `MPI_CREATOR_TXTPATH`, which is the Creator texture path environment variable.

- **Relative to Database Directory** specifies each directory path relative to the directory of the current database.
- **Relative to Working Directory** specifies each directory path relative to the directory from where Creator is launched.

Statistics

Your realtime system is likely to have limits in the number of textures it can support, their type, and how much memory is made available for them. Creator keeps track of this information whether or not the texture is currently displayed in the database. You can view it by choosing **Info/Statistics** from the Texture palette.

Note: Statistics does not compute mipmapping. Assigning a mipmap texture attribute will result in increased texture memory usage by a factor of 1.3.

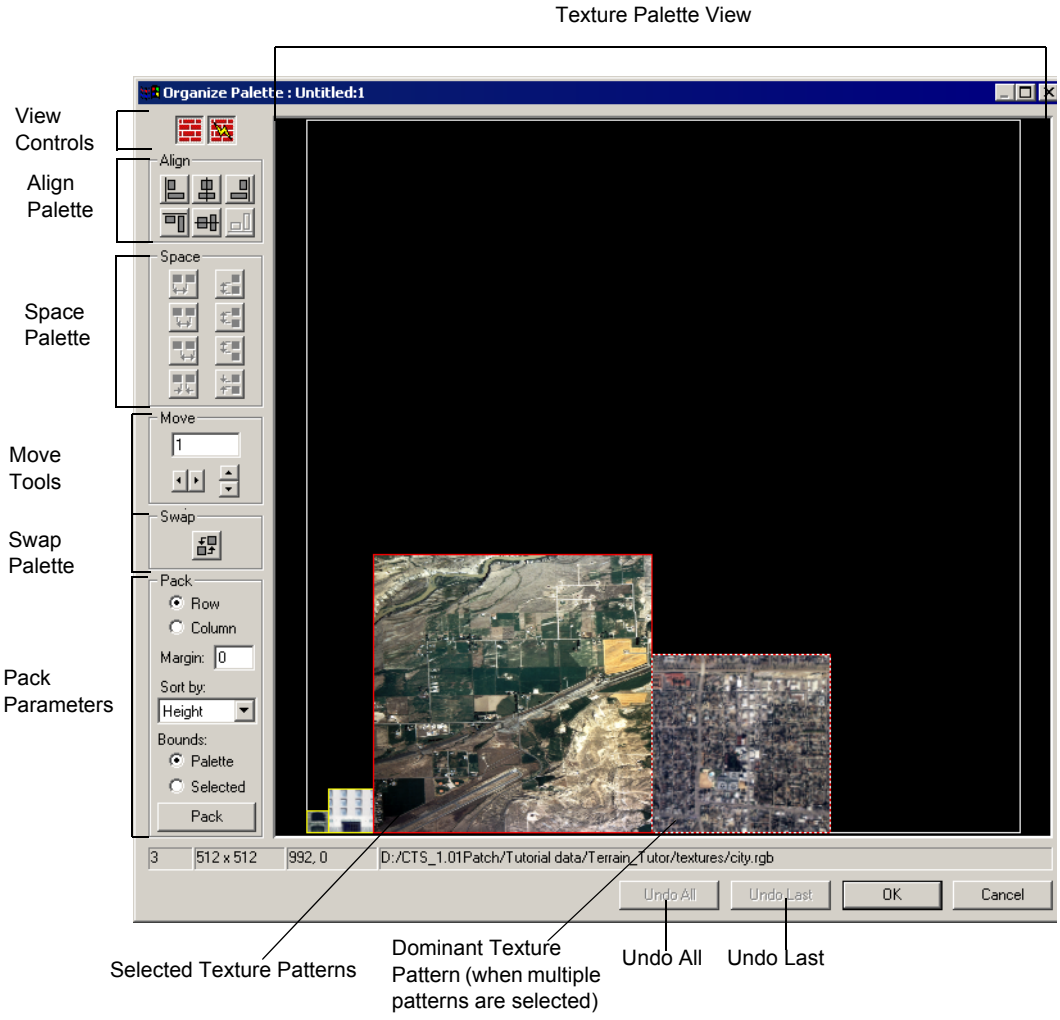


Organizing Texture Palettes

The **Organize Palette** tool, located in the Texture palette **Edit** menu, enables you to control how you want to arrange texture patterns within a texture palette. You can use the mouse to position patterns, or you can use tools within **Organize Palette** to align, space, or sort patterns within the palette.

The Palette view in the *Organize Palette* window displays the patterns as they are positioned in the texture palette. They are scaled down so that they all fit within the view. You can arrange the texture patterns in a specific order, or pack them closely together with a specified margin between them. When you are done organizing your

patterns within the palette, click **OK** to save the changes, **Undo Last** or **Undo All** to undo changes, or **Cancel** to not save changes and close the window.

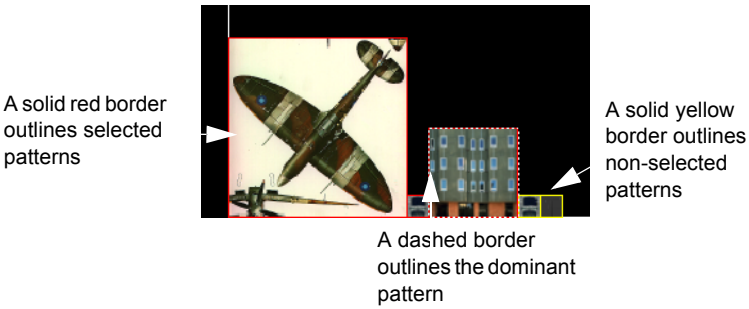


For a quicker method, you can select **Pack Palette** in the Texture palette **Edit** menu to pack patterns in the current palette. **Pack Palette** uses the last settings used in the **Organize Palette** tool. If **Organize Palette** was not previously used, then **Pack Palette** uses the default settings. Select **Pack All Palettes** to pack all palettes instead of only the current palette.

Selecting Patterns in the Palette View

To select patterns in the Palette view, you can either click on an individual pattern using the left mouse button, or drag a fence around multiple patterns. Press the Shift key while dragging to select more than one pattern. If a pattern is already selected while you press Shift, you will deselect that pattern without affecting any other selected pattern. The selected pattern is outlined with a solid red border.

When multiple patterns are selected, one of the patterns becomes the dominant pattern (outlined with a dashed line). When you are aligning multiple patterns, **Organize Palette** uses the dominant pattern to determine how the other patterns are aligned. If you want to set a new dominant pattern, and the pattern you want to be dominant is not selected, you should hold the Shift key and left-click on the pattern; if it is already selected, you should hold the Shift key and left-click on the pattern twice (the first click will deselect it; the second will select it again, and make it dominant).



Moving Patterns in the Palette View

There are several ways to move patterns in the Palette view. You can drag them using the mouse, or use one of the many tools in the *Texture Organizer* window, such as the **Align**, **Space**, **Move**, **Swap**, and **Pack** tools.

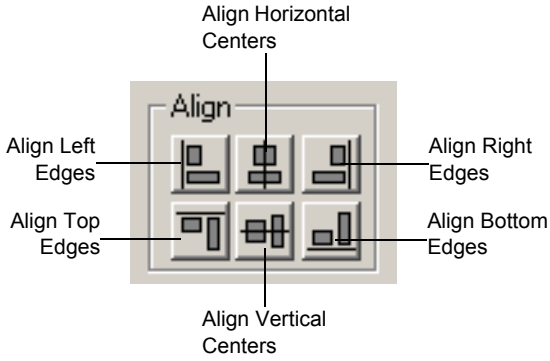
To move a single pattern with the mouse, press the left mouse button over the pattern that you want to move, and drag it to the new position in the Palette view. To move multiple patterns, first select all the patterns that you want to move. When all the patterns are selected, press the left mouse button over any of the selected patterns, and drag the mouse to the new position.

> **Note:** If you press Ctrl while dragging, you can constrain the movement to a single axis.

Align Tools

The **Align** tools in the Align palette enable you to align patterns within the Palette view. Each **Align** tool uses the position of the dominant pattern to determine the position of the other selected patterns. The dominant pattern remains in its current position, and the remaining selected patterns are moved to align with the dominant pattern.

You must select at least two patterns to activate the **Align** tools. Only the tools that can be used for the current configuration of the selected patterns will be active.



Align Left Edges

Aligns the left edges of the selected patterns to the left edge of the dominant pattern.

Align Horizontal Centers

Aligns the horizontal centers of the selected patterns to the horizontal center of the dominant pattern.

Align Right Edges

Aligns the right edges of the selected patterns to the right edge of the dominant pattern.

Align Top Edges

Aligns the top edges of the selected patterns to the top edge of the dominant pattern.

Align Vertical Centers

Aligns the vertical centers of the selected patterns to the vertical centers of the dominant pattern.

Align Bottom Edges

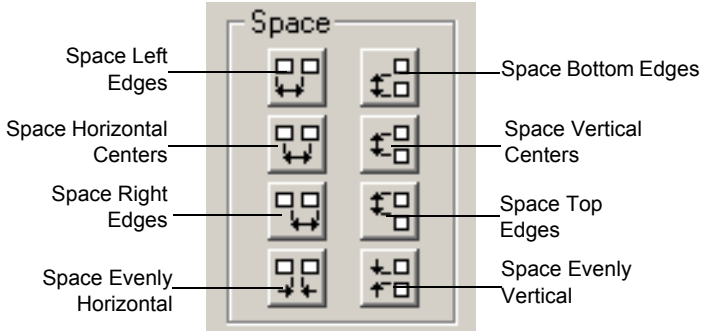
Aligns the bottom edges of the selected patterns to the bottom edge of the dominant pattern.

Space Tools

The **Space** tools in the Space palette enable you to space patterns within the Palette view. When you use a horizontal spacing tool, the left-most and right-most selected patterns remain in their current positions, and the remaining selected patterns are spaced evenly between them. Similarly, when you use a vertical spacing tool, the top-most and bottom-most selected patterns remain in their current positions, and the remaining selected patterns are spaced evenly between them.

You must select at least three patterns to activate the **Space** tools. Only the tools that can be used for the current configuration of the selected patterns will be active.

> **Note:** The position of the dominant pattern does not affect how the spacing tools work.



Space Left Edges

Spaces the selected patterns so that the horizontal distance between the left edges of each adjacent pattern is an equal distance.

Space Horizontal Centers

Spaces the selected patterns so that the horizontal distance between the horizontal centers of each adjacent pattern is an equal distance.

Space Right Edges

Spaces the selected patterns so that the horizontal distance between the right edges of each adjacent pattern is an equal distance.

Space Evenly Horizontal

Spaces the selected patterns so that the horizontal distance between each adjacent pattern is an equal distance.

Space Top Edges

Spaces the selected patterns so that the vertical distance between the top edges of each adjacent pattern is an equal distance.

Space Vertical Centers

Spaces the selected patterns so that the vertical distance between the vertical centers of each adjacent pattern is an equal distance.

Space Bottom Edges

Spaces the selected patterns so that the vertical distance between the bottom edges of each adjacent pattern is an equal distance.

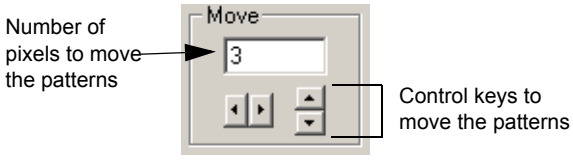
Space Evenly Vertical

Spaces the selected patterns so that the vertical distance between each adjacent pattern is an equal distance.

Move and Swap Tools

The **Move** and **Swap** tools enable you to easily move patterns a specified distance in pixels, or to swap the positions of two patterns.

Move Patterns



Moves the selected patterns the specified number of pixels. You can enter a value from 1 to 100. Click the right and left arrow keys to move the patterns horizontally. Click the up and down arrows to move the patterns vertically.



Swap Positions

Swaps the positions of two patterns.

Pack Parameters

You can pack patterns within the palette according to parameters that you specify. You first select the patterns that you want to pack. You specify how you want those patterns arranged, such as in a row, column, or a sort order, and then press **Pack**.

Row or Column

Packs patterns together in either a row or column.

Margin

Specifies the spacing between patterns in pixels.

Sort By

Specifies the attribute used to pack the selected patterns. Patterns can be packed by **Height**, **Width**, **Position**, **Index**, or **Name**. The **Name** attribute sorts patterns in alphabetical order; the **Index** attribute sorts patterns by index number in ascending order (0 first); **Width** and **Height** sorts in descending order (bigger first); and the **Position** attribute sorts patterns in ascending order by the x position (left-most first).

Bounds

Specifies the bounds in which the patterns are packed. If **Palette** is selected, sorted patterns will be positioned to fill the entire palette. If **Selected** is selected, sorted patterns will be confined to the bounding box around the selected patterns.

View Controls

These **View** buttons control how the Palette view is drawn.



Draw Patterns

Draws patterns in the Palette view. When **Draw Patterns** is deselected, draws only the pattern outlines.



Draw Dynamic

Draws patterns while you drag them in the Palette view. When **Draw Dynamic** is deselected, draws only the pattern outlines while you drag patterns.

Texture Palette Files

A *Texture palette file* is an ASCII file (such as `myfile.txt`) that specifies the contents of the Texture palette. Texture palette files can be saved and used in other databases. They are useful for saving and loading related sets of texture patterns. A Texture palette file contains a list of entries, in the format: `palnum patnum x y filename`.

You can create a default Texture palette that loads automatically whenever a new database is opened. This Texture palette must be saved with the name `default.txt`. Creator looks for this file first in the folder defined by the MPI_CREATOR_TXTPATH variable, and then in the working directory. The working directory for Windows is the directory where Creator is installed, and for IRIX it is the directory where Creator was launched. If `default.txt` is not found, then Creator loads an empty Texture palette.

Note: In the *Terrain* window's **Project** panel, you select a texture palette for the terrain you are about to create. This palette automatically becomes the new default palette for the session; when you exit from Creator (Windows) or start Creator in a different shell window (IRIX), Creator reverts to the usual search pattern for a default palette.

Texture Palette File Parameters

Each Texture palette entry is defined with the following parameters:

palnum

Specifies the number (range 0 to 127) of the palette in which the pattern is loaded.

patnum

Specifies the local pattern ID (range 0 to 255) on the palette specified by `<palnum>`.

x

Specifies the horizontal offset (range 0 to 2047) from the lower-left corner of the palette to the lower-left corner of the pattern.

y

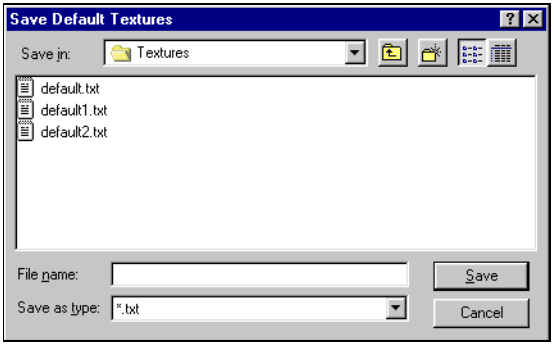
Specifies the vertical offset (range 0 to 2047) from the lower-left corner of the palette to the lower-left corner of the pattern.

Filename

Specifies the directory path and file name of the texture pattern.

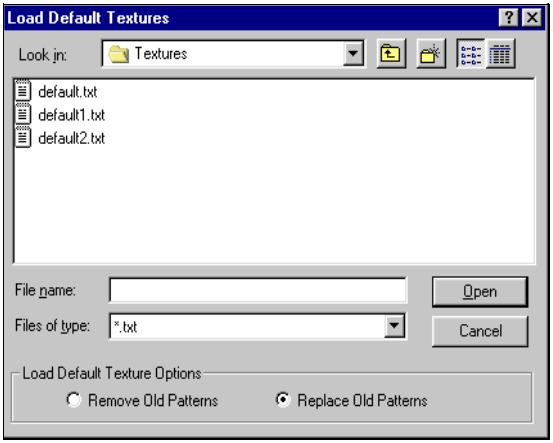
To Save Texture Palette Files

To save a Texture palette file, choose **File/Save Palette** from the Texture palette and enter a name in the *Save Default Textures* dialog box. Click **Save**.



To Load Texture Palette Files

To load a Texture palette file, choose **File/Load Palette** from the Texture palette and choose a file in the *Load Default Textures* dialog box. Click **Open**.



Load Default Textures Parameters

[Under Construction]

Remove Old Patterns

Specifies that all other Texture palettes are cleared before the new set of patterns is loaded.

Replace Old Patterns

Specifies that the new set of patterns is *added* to the Texture palettes. New patterns replace existing patterns with the same patter number numbers, but do not affect the other texture patterns in the palette.

To Create a Default Texture Palette

The following steps create a default Texture palette that will be loaded each time a new database is creaed.

Step 1. Load the desired texture patterns into the Texture palette.

Step 2. Click **File/Save Palette**. The *Save Default Textures* dialog box opens.

Step 3. Set **Save In** to the folder specified by the MPI_CREATOR_TXTPATH environment variable, if it was set. Otherwise, set **Save In** to the working directory.

Step 4. The working directory for Windows is the directory where Creator is installed, and for IRIX it is the directory where Creator was launched

Step 5. In the **File Name** field, type `default`, and click **Save**. The Texture palette will be saved with a `.txt` suffix.

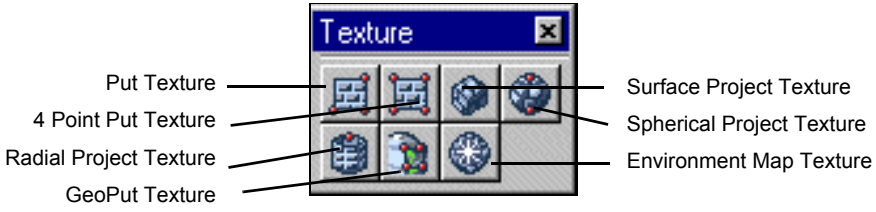
Step 6. Restart Creator to activate the default palette.

Applying Texture

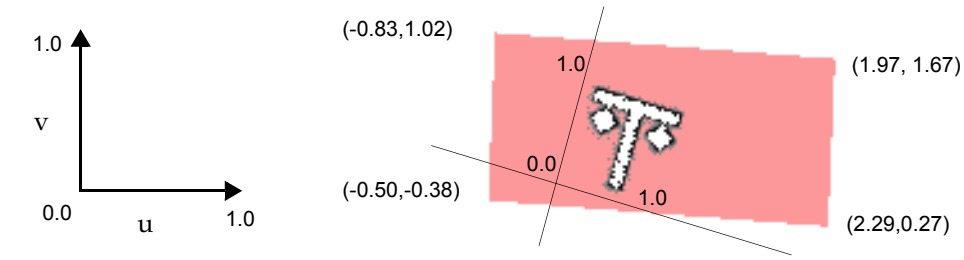
When a texture pattern is loaded into the Texture palette, it can be applied to, or removed from, a face or group of faces using tools in the **Properties** or **Texture** toolboxes and in the **Edit** menu. Tools in the **Properties** toolbox can also be used to change the **Current Texture** to one that has been previously applied. The following sections describe each of the tools and menu options used to apply texture patterns.

Texture Tools

The **Texture** tools apply texture patterns to faces, and modify the mapping of applied texture patterns.



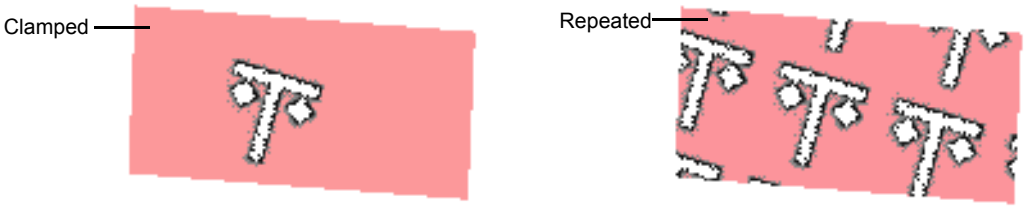
Creator applies texture patterns by mapping the texels in the original image onto faces in the database. To do this, the software defines a normalized u,v coordinate system for the original texture pattern, then maps the pattern into the database by saving u,v values as vertex attributes of each textured face.



u,v coordinate system of original texture pattern

u,v values of textured vertices of face

When a textured surface is larger than a single instance of the applied texture pattern, the pattern is either *clamped* or *repeated*, depending on the **Wrap Method** texture preference (see "Texture Preferences" on page 541.)



Put Texture (3-Point Put) Tool



Put Texture maps a texture onto one or more faces using three points: **Origin**, **Alignment**, and **Shear**. Three **From** points are mapped from the texture pattern to three **To** points on the object. You can position, orient, scale, and shear the current texture pattern while applying it. You can also specify the number of times to repeat the texture pattern within a texture.

Note: One grid space in the texture application tool is equal to one *uv* unit.

To Apply a Texture Using the Put Texture Tool

See "Put Texture Parameters" on page 447 for descriptions of all the parameters in the *Put Texture* dialog box.

Step 1. Open the **Mode** toolbox and select **Face** mode (Shift+F).

Step 2. Select the face(s) to be textured.



Step 3. Open the **Texture** toolbox.



Step 4. Click the **Put Texture** tool.

Step 5. Set the Put Texture Parameters.

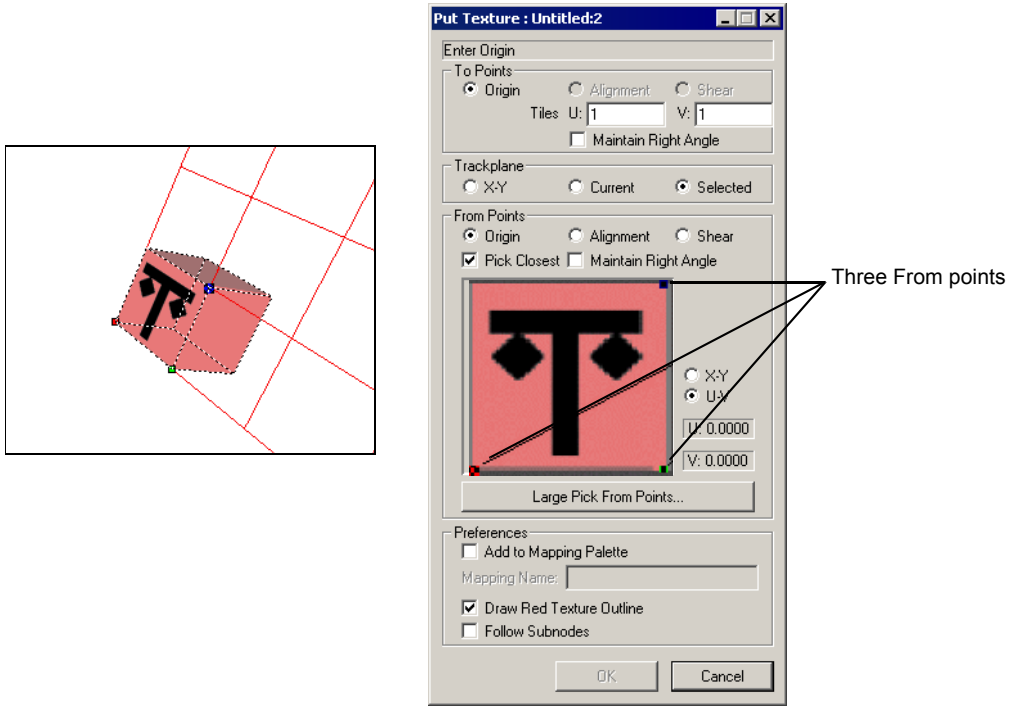
Step 6. Click the positions for the **To** points in the database, either on vertices, along edges, or on the tracking plane.

Note: One texture grid space is equal to one u,v unit.

Entering the **Alignment** or **Shear** point with the middle mouse button constrains the point either to reference an existing vertex in the database or to lie along one of the edges of the texture grid.

Step 7. Modify the positions of the **To** and **From** Points if necessary. Use Ctrl-left mouse click to select the desired point for editing.

Step 8. Click **OK**.



Put Texture Parameters

The *Put Texture* dialog box contains the following parameters:

To Points (Origin, Alignment, and Shear)

Determine texture mapping. Each **From** point on the texture pattern maps to the corresponding **To** point in the database, as follows:

- **Origin** specifies the location of the From **Origin** point (which corresponds to the lower right corner of the texture pattern) on the face.
- **Alignment** defines the alignment and scale of the texture pattern along the *u* axis. If you hold down the Alt key while entering this point, the texture pattern is not scaled. Instead, the length of the *u* axis is defined using the texture's **Real World Size** attribute.

- **Shear** defines the alignment and scale of the texture pattern along the *v* axis. If you hold down the Alt key while entering this point, the texture pattern is not scaled. Instead, the length of the *v* axis is defined using the texture's **Real World Size** attribute. (See "Texture Attributes" on page 523.)

Tiles U, V

The number of times the texture pattern repeats in both the *u* and *v* directions. You can repeat a texture pattern before or after the texture is applied to the face in the *Put Texture* dialog box.

> **Note:** The values can be either whole numbers or fractional amounts, such as *1.5* for **Tiles U** and *2.4* for **Tiles V**.

Trackplane

Controls the definition of the tracking plane while you are positioning the texture. When you close the *Put Texture* window, the tracking plane returns to its previous position. The following controls are available:

- **X-Y** sets the tracking plane to the xy plane. This is especially useful in applying a large terrain texture, such as an airport landing field.
- **Current** uses the tracking plane definition in effect when you start **Put Texture**.
- **Selected** sets the tracking plane to the first face in the Select list.

From Points

Defines the **From** points, **Origin, Alignment,** and **Shear**. The coordinate of the selected point appears to the right of the image in the *Put Texture* dialog box. If From Points **X-Y** is set, the coordinate is displayed as an offset, in texels, from the lower-left corner of the texture pattern. If **U-V** is set, the point is displayed in the *u*,*v* coordinate system.

Pick Closest

Overrides the **From Points** option buttons (**Origin, Alignment**, and **Shear**). If **Pick Closest** is set, Creator selects the **From** point closest to the place you click in the texture image.

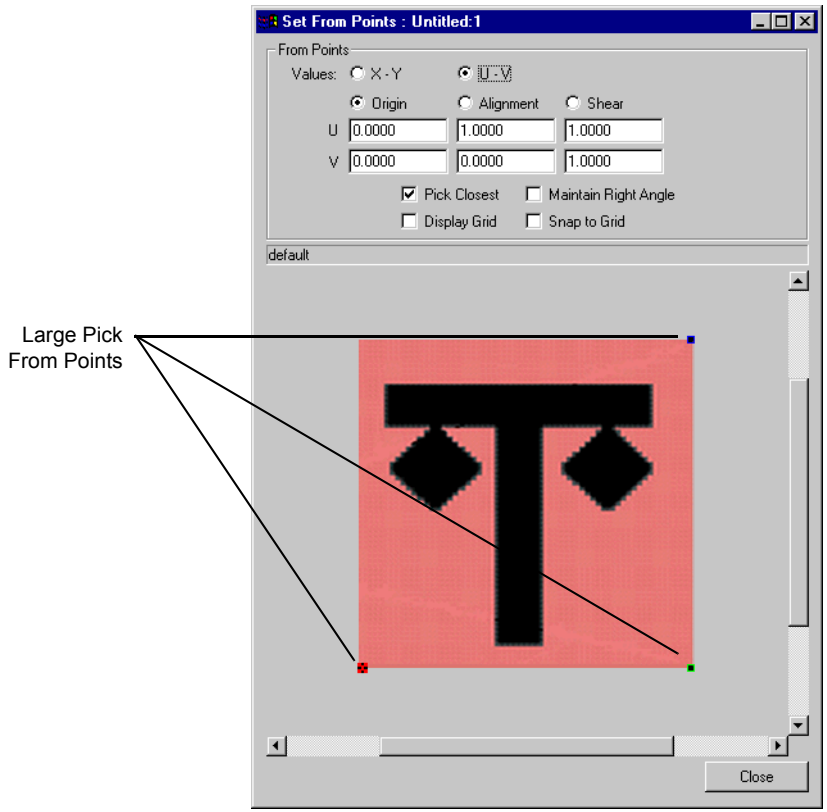
Maintain Right Angle

Maintains a right angle with the **From/To** points that you choose (**Origin, Alignment**, and **Shear**). The third point that you choose automatically becomes perpendicular with the second point to form a right angle. Any point that you move maintains a right angle with the other points.

Large Pick From Points

Opens the *Set From Points* window in which you can place points more accurately on a larger image of the current texture. The coordinates of the selected point also appear in editable text fields. The **Tiles U** and **V** values in the *Put Texture* dialog box are used as the *u*,*v* values.

If **Display Grid** is set, you can clearly identify individual pixels. If **Snap to Grid** is set, points snap to the nearest pixel. If **Pick Closest** is set, Creator selects the **From** point closest to the place you click in the texture image.



Add to Mapping Palette

Controls whether the texture mapping defined during this function is saved in the Texture Mapping palette. (See “The Texture Mapping Palette” on page 516.)

Mapping Name

Associates a text name with the new texture mapping. “The Texture Mapping Palette” on page 516 This is useful so you can reuse this texture mapping.

Draw Red Texture Outline

Controls the display of a red grid showing the position and orientation of the applied texture. The red outline appears automatically on computers that do not display dynamic texture.

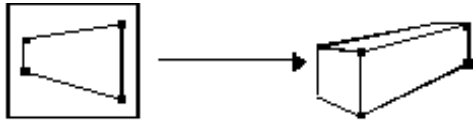
Follow Subnodes

Controls whether subfaces are affected by **Put Texture**.

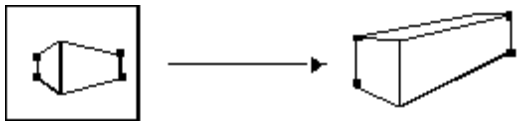
4 Point Put Texture Tool



4 Point Put Texture (also known as Put Texture Perspective) lets you perform perspective correction through the use of a fourth control point, **Perspective**, and an image scaling tool, **Depth Scale**. Perspective correction is used for oblique images (images that are not face-on), without the need to correct the image with one of the image warp tools.



4 Point Put Texture can also be used to wrap a texture pattern around a corner. For example, two sides of a building are mapped from a single texture pattern using the points shown and setting **Depth Scale** to add depth to the texture pattern. This permits the texture pattern to wrap around a corner and match at the vertices you select. To perform this type of texture application, the dimensions of the building in the database model must be scaled proportionally to match the dimensions of the building in the image.



To Apply a Texture Using the 4 Point Put Tool

See "4 Point Put Texture Parameters" on page 451 for descriptions of all the parameters in the *4 Point Put Texture* dialog box.

Step 1. In Face mode, select the faces to be textured.



Step 2. Open the **Texture** toolbox.

Step 3. Click the **4 Point Put Texture** tool.

Step 4. Set the controls in the *4 Point Put Texture* dialog box.

Step 5. Click the positions for the **To** points in the database.

Note: One texture grid space is equal to one *uv* unit.

Entering the **Alignment**, **Shear,** or **Perspective** point with the middle mouse button constrains the point either to reference an existing vertex in the database or to lie along one of the edges of the texture grid.

Step 6. Modify the positions of the **To** and **From** Points if necessary. Use Ctrl-left click to select the desired point for editing.

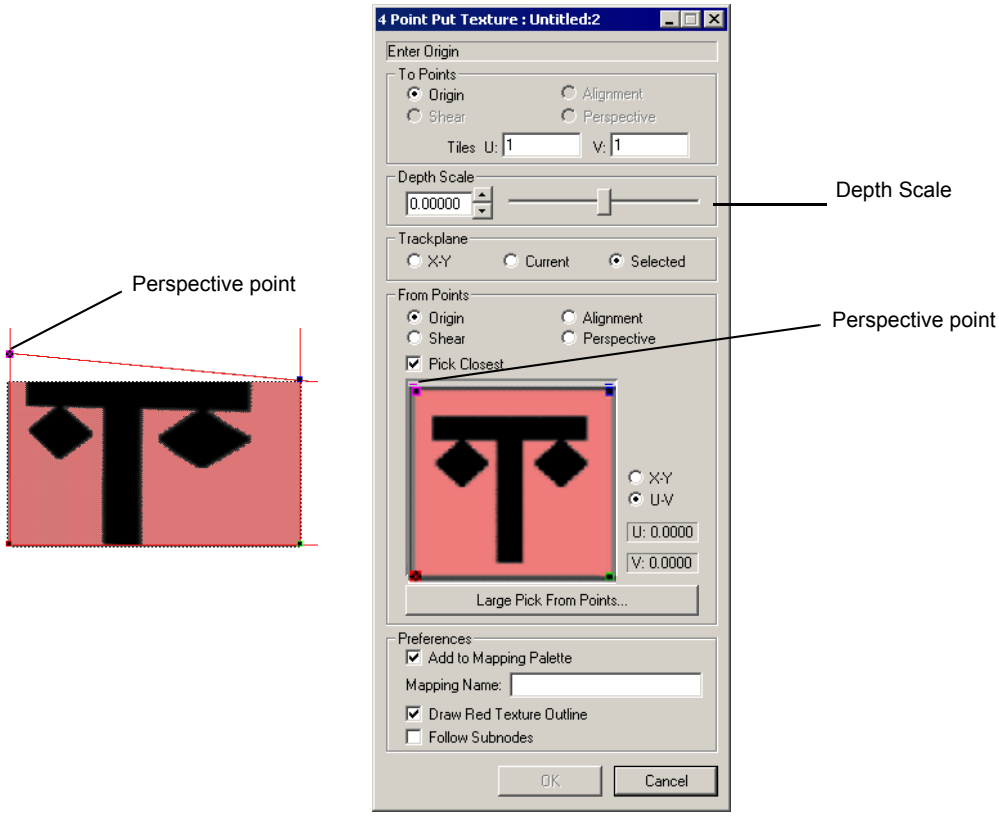
Step 7. Click **OK**.

4 Point Put Texture Parameters

The *4 Point Put Texture* dialog box contains the following parameters.

To Points (Origin, Alignment, Shear, and Perspective)

Determine texture mapping. Each **From** point on the texture pattern maps to the corresponding **To** point on an object, as follows:



- **Origin** places the texture pattern **Origin** (**From** point) on an object.
- **Alignment** defines the alignment and scale of the texture pattern *u* axis. If you hold down the Alt key while entering this point, the texture pattern is not scaled. Instead, the length of the *u* axis is defined using its **Real World Size** attribute.
- **Shear** defines the alignment and scale of the texture pattern along the *v* axis. If you hold down the Alt key while entering this point, the texture pattern is not scaled. Instead, the length of the *v* axis is defined using its **Real World Size** attribute. (See "Texture Attributes" on page 523.)
- **Perspective** defines a fourth point for changing the perspective of the texture pattern.

Tiles U, V

The number of times the texture pattern repeats in both the *u* and *v* directions. You can repeat a texture pattern before or after the texture is applied in the *4 Point Put Texture* dialog box.

> **Note:** The values can be either whole numbers or fractional amounts, such as *1.5* for **Tiles U** and *2.4* for **Tiles V**.

Depth Scale

Warps the image out from the original plane (defined by the **Origin**, **Alignment**, **Shear**, and **Perspective** points) in order to wrap around a corner, countering the apparent "bulge" at the juncture of the two polygons. The values are within an arbitrary range of -1.0 to +1.0.

Trackplane

Controls the definition of the tracking plane while using **4 Point Put**. When you close the *4 Point Put Texture* dialog box, the tracking plane returns to its previous position. The following controls are available:

- **X-Y** sets the tracking plane to the xy plane. This is especially useful in applying a large terrain texture, such as an airport landing field.
- **Current** uses the tracking plane definition in effect when you start **4 Point Put Texture**.
- **Selected** sets the tracking plane to the first face in the Select list.

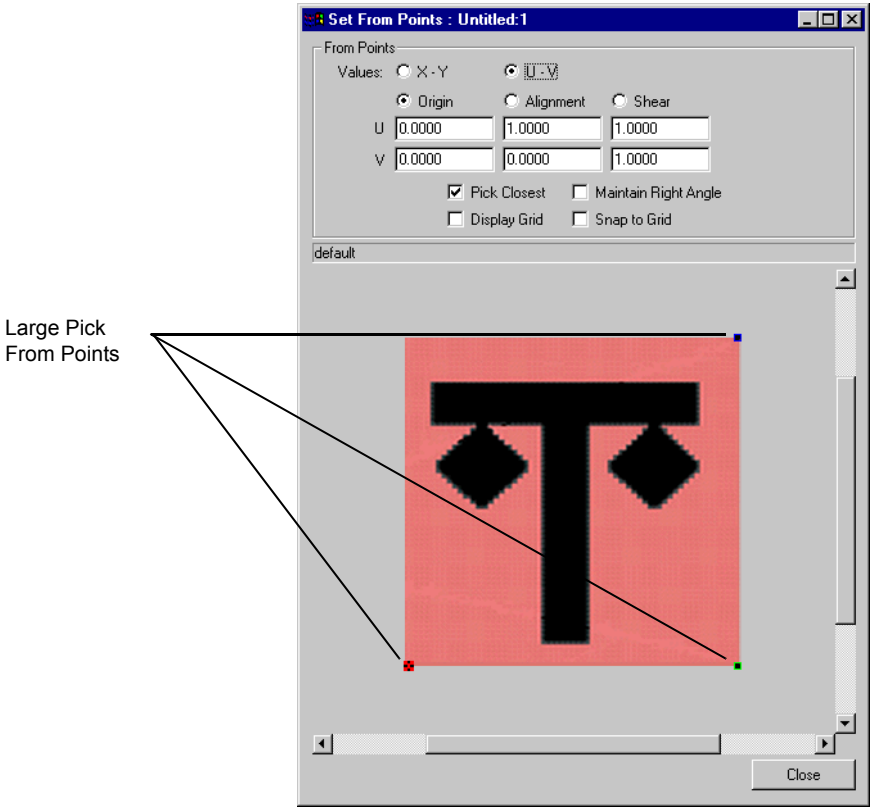
From Points

Defines the **From** points, **Origin, Alignment, Shear** and **Perspective**. The coordinate of the selected point appears to the right of the image in the *Put Texture* dialog box. If **From** Points **X-Y** is set, the coordinate is displayed as an offset, in texels, from the lower-left corner of the texture pattern. If **U-V** is set, the point is displayed in the *u*,*v* coordinate system.

Large Pick From Points

Opens the *Set From Points* window in which you can more accurately place points on a larger image of the current texture. The coordinates of the selected point appear in editable text fields. The **Tiles U** and **V** values in the *4 Point Put Texture* dialog box are used as the *u*,*v* values.

If **Display Grid** is set, you can clearly identify individual pixels. If **Snap to Grid** is set, points snap to the nearest pixel. If **Pick Closest** is set, Creator selects the **From Point** closest to the point you click in the texture image.



Pick Closest Point

Overrides the **From Point** option buttons (**Origin, Alignment, Shear**, and **Perspective**). If **Pick Closest** is set, Creator selects the **From** point closest to the place you click in the texture image.

Add to Mapping Palette

Controls whether the texture mapping defined during this function is saved in the Texture Mapping palette (see "The Texture Mapping Palette" on page 516). This is useful so you can reuse this texture mapping.

Mapping Name

Associates a text name with the new texture mapping. (See "The Texture Mapping Palette" on page 516).

Draw Red Texture Outline

Controls the display of a red grid showing the position and orientation of the applied texture. The red outline appears automatically on computers that do not display dynamic texture.

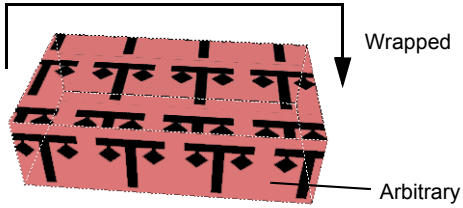
Follow Subnodes

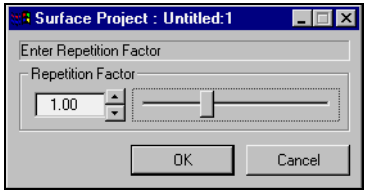
Controls whether subfaces are affected by **4 Point Put Texture**.

Surface Project Texture Tool



Surface Project wraps a texture around selected faces or all sides of a volume without shearing. It is particularly useful for texturing walls and fences. The texture is applied to the first face in the selected object then wrapped across contiguous linear polygons sharing common edges. Non-contiguous polygons are arbitrarily mapped.





To Apply a Texture Pattern with the Surface Project Tool

Step 1. Select a set of faces in the database.



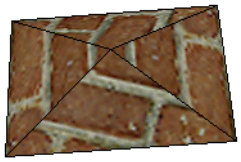
Step 2. Open the **Texture** toolbox.



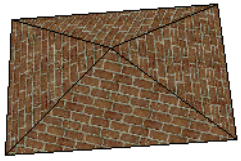
Step 3. Click the **Surface Project** tool.

Step 4. Enter the **Repetition Factor** or use the slider to specify the number of times a pattern repeats across the surface in both *u* and *v* in the database.

Valid values are from .01 to 10,000; the larger the number, the smaller the pattern. If you enter a number smaller than .01 or larger than 10,000, Creator defaults to .01 or 10,000.



Repetition Factor set to .01



Repetition Factor set to 6

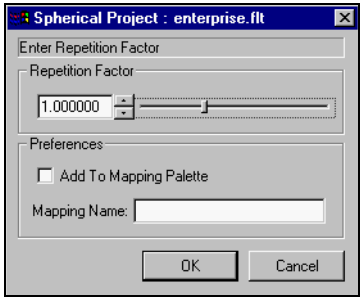
Step 5. Click **OK**.

Spherical Project Texture



Spherical Project covers a continuous surface with a smooth flow of texture by projecting the pattern spherically out from the center of all selected polygons. It is mainly used to wrap a texture around a sphere.





To Apply a Texture Pattern with the Spherical Project Tool

Step 1. In Object mode, select an object in the database.



Step 2. Open the **Texture** toolbox.



Step 3. Click the **Spherical Project** tool.

Step 4. Enter the **Repetition Factor** or use the slider to specify the number of times a pattern repeats across the surface in both *u* and *v* in the database.

Step 5. Set the **Add to Mapping Palette** checkbox if you want to add the texture mapping you define to the Texture Mapping palette (see "The Texture Mapping Palette" on page 516). This is useful if you want to reuse this texture mapping.

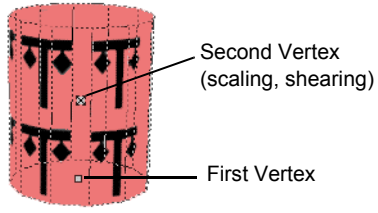
Step 6. If you have set **Add to Mapping Palette**, enter a name for the new texture map in the **Mapping Name** field.

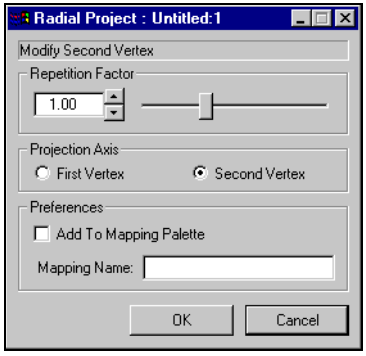
Step 7. Click **OK**.

Radial Project Texture Tool



Radial Project (cylindrical project) maps texture in a continuous flow around a surface by projecting the texture pattern outward from a construction edge. It is typically used to wrap texture around a cone, cylinder, or cube using a user-defined center point (first vertex) and a direction vector (second vertex).





To Apply a Texture Pattern with the Radial Project Tool

Step 1. In Object or Face mode, select an object or set of faces in the database.



Step 2. Open the **Texture** toolbox.

Step 3. Click the **Radial Project** tool.

Step 4. Enter the **Repetition Factor** or use the slider to set the number of times the pattern repeats across the surface in both *u* and *v* in the database.

Step 5. Set the **First Vertex**. This aligns the tracking plane to the axis along which you want to move the vector. To change the length of the edge without changing its orientation, point to the edge (not a vertex) and drag with the middle mouse button.

Step 6. Move the **Second Vertex** up or down vertically (scaling the texture) or to the right or left (shearing the texture).

Step 7. Set the **Add to Mapping Palette** checkbox if you want to add the texture mapping you define to the Texture Mapping palette (see "The Texture Mapping Palette" on page 516). This is useful if you want to reuse this texture mapping.

Step 8. If you have set **Add to Mapping Palette**, enter a name for the texture map in the **Mapping Name** field.

Step 9. Click **OK**.

Radial Project Parameters

The *Radial Project* dialog box sets the following parameters.

Repetition Factor

Determines the number of times a pattern repeats across the surface in both *u* and *v*.

First Vertex and Second Vertex

Define the temporary construction edge. The length of this edge scales the pattern in *v*. To change the length of the edge without changing its orientation, point to the edge (not a vertex) and drag with the middle mouse button.

Add to Mapping Palette

Controls whether the texture mapping you define is saved in the Texture Mapping palette (see "The Texture Mapping Palette" on page 516). This is useful if you want to reuse this texture mapping.

Mapping Name

Associates a text name with the new texture mapping. (See "The Texture Mapping Palette" on page 516.)

Environment Map Texture Tool



The **Environment Map Texture** tool maps the current texture to a face, object, or group, adding a mirror-like effect to the texture. This effect is sometimes called "reflection mapping." To create a more realistic mirror, you can map a simplified, two-dimensional picture of your 3D environment to a surface. This picture, usually a simple screen shot, simulates a reflection of objects that are placed in front of the surface.

The mirror-like effect can be enhanced by adding material properties, light sources, and/or using **Modify Vertex Normals** to modify the direction of the vertex normals after **Calculate Shading** is performed. The environment mapped texture will blend with shading if you choose the Blend Texture Attribute in the *Texture Attributes* window.

The *u*,*v* coordinates of a standard mapped texture stay static as you rotate the object. In Creator, the mirror-like effect of environmental mapping results when the texture's u,v coordinates are remapped as the object is rotated. The texture moves around the object as the eyepoint moves giving the illusion of reflection.

To Apply an Environment Mapped Texture

Step 1. Load a texture pattern into the Texture palette.

Step 2. Using the appropriate modeling mode, select a face, object, or group to which you will map the texture.

Step 3. Choose **Attributes/Calculate Shading** (Ctrl+G) to add vertex normals to the object for enhancing the reflective effect with shading.

Step 4. Select one of these shading attributes, and click OK:

- **Flat** - Shades faces using only the face colors
- **Lit** - Shades faces using both face colors and light
- **Gouraud** - Shades faces using only vertex colors. This is the default setting.
- **Lit Gouraud** - Shades faces using both vertex colors and light



Step 5. Open the **Texture** toolbox.



Step 6. Click **Environment Map Texture.** The texture is mapped to the surface.



Step 7. If **Draw Texture** is not set, open the View panel, and click **Draw Texture** (T) to view the effect.

Sky texture was Environment Mapped to the windows without Calculate Shading, below.



Sky texture was Environment Mapped to the windows using Gouraud shading to create lightly tinted windows, below. The top vertices of the windows were colored grey, and the bottom vertices were colored white.



Texture Tools in the Properties Toolbox

The **Properties** toolbox has two tools, **Insert Texture** and **Get Texture From Face** that apply the current texture and change the current texture, respectively. More information is available in the following topics:

"Insert Texture Tool" on page 461

"Get Texture From Face Tool" on page 462

Insert Texture Tool



The **Insert Texture** (Ctrl+T) tool, and the menu command **Edit/Insert Texture**, assign the current texture in the Texture palette to the selected faces. **Insert Texture** does not assign a texture mapping; you use it to change the texture on faces to which a texture has already been mapped, or to apply a texture using the current mapping in the Texture Mapping palette.

To Change a Texture on One or More Textured Faces

Step 1. Open the Texture palette and select the texture you want to use. (This makes it the current texture.)

Step 2. Select the face(s) whose texture you want to change.



Step 3. Open the **Properties** toolbox if you want to use the **Insert Texture** tool.

Step 4. Click **Insert Texture**, or choose **Edit/Insert Texture**. The current texture is applied to the selected faces.



To Use the Current Texture Mapping With Insert Texture

Step 1. Make sure the texture and texture mapping you want to use are selected in the Texture palette and Texture Mapping palette, respectively.

Step 2. Select the face(s) you want to texture.



Step 3. Open the **Properties** toolbox if you want to use the **Insert Texture** tool.

Step 4. Hold down Ctrl and click **Insert Texture**, or hold down Ctrl and choose **Edit/Insert Texture**. The current texture and texture mapping are applied to the selected faces.



Texture mapping information includes the original database coordinates used to apply the texture pattern. To reuse a texture mapping successfully, the new faces must positioned geometrically consistent with the original textured faces. To apply texture to faces that are at different orientations, use one of the **Texture** tools (see "Texture Tools" on page 444).

Get Texture From Face Tool



Get Texture From Face (Ctrl-Shift+T) sets a new current texture using the texture assigned to a face you select. You can hold down Ctrl and click **Get Texture From Face** to set the texture as a current texture mapping in the Texture Mapping palette.

To Use Get Texture From Face

Step 1. Select a face at any level in the hierarchy.



Step 2. Open the **Properties** toolbox.



Step 3. Click the **Get Texture From Face** tool.

Note: You can also select a face and then choose **Edit/Texture From Face** to change the current texture to that of the selected face.

Removing a Texture

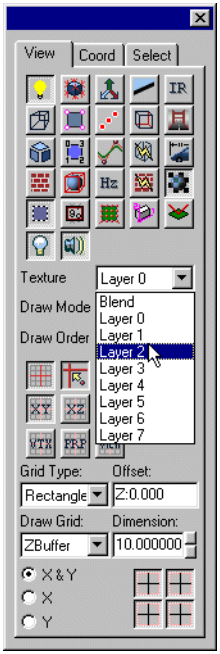
Remove Texture in the **Edit** menu detextures the currently-selected layer of the currently-selected faces. **Remove Texture** works by setting the Texture attribute for the selected layer of each selected face to -1 (no texture) and clearing the u,v texture mapping attributes for that layer. **Remove Texture** does not affect the Texture palettes, or the textures assigned to any other layers.

Step 1. Select the face(s) to be detextured.



Step 2. Open the View panel.

Step 3. Select the layer to be detextured.



Step 4. Choose **Edit/Remove Texture**.

If there are subfaces among the selected faces, a *Verify* dialog box appears. Click **Yes** to remove texture from subfaces or **No** to only remove texture from selected faces.

Using Chop Texture to Reduce Texture Size

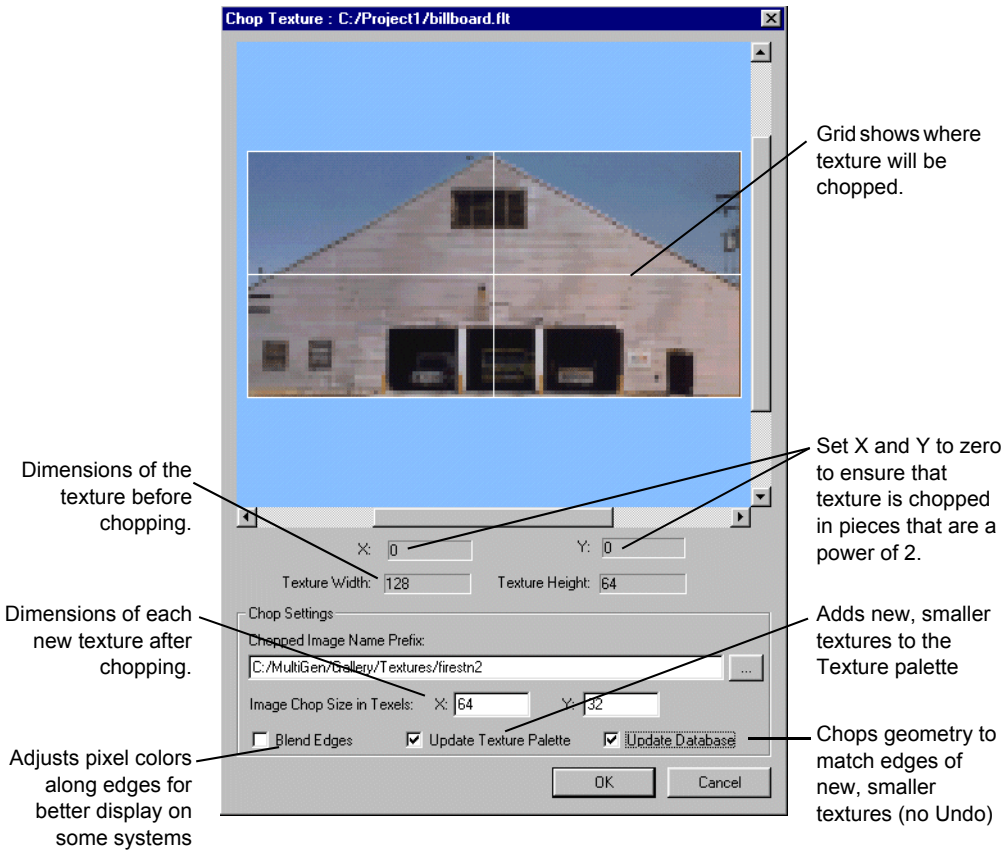
Chop Texture, in the Texture palette **Edit** menu, divides a texture into four smaller files, reducing the amount of texture memory needed for the realtime display. The original texture file is preserved.

If the texture already has been applied to faces in the database, **Chop Texture** can divide faces along the lines where the texture has been chopped, and update the texture pattern of each polygon so that the appropriate smaller texture is assigned.

To use **Chop Texture**:

Step 1. Click **Current Texture** on the Toolbar or choose **Palettes/Texture** to open the Texture palette.

Step 2. Select the texture you want to chop and choose **Chop Texture** from the Texture palette **Edit** menu. The *Chop Texture* dialog box appears.



Step 3. Click the lower-left corner of the texture display in this dialog box to position the grid over the texture. Adjust where you are clicking until both the X and Y settings are 0 (zero). If the texture you loaded into the Texture palette has dimensions that are powers of 2, **Chop Texture** will create smaller textures that also have dimensions that are powers of 2.

Step 4. Set **Update Texture Palette** to add the new textures to the Texture palette. If you do not set this option, **Chop Texture** creates four smaller texture files in the same directory as the original texture pattern, but does not add these textures to the Texture palette. (You will have to add them manually if you want to use them in the database.)

Step 5. When **Update Texture Palette** is set, **Update Database** becomes active. Set this option if (1) the texture has already been applied to faces in the database and (2) you want to replace the large texture with the new, smaller textures on these faces. A message warns you that there is no **Undo** for this function.

If you set **Update Database**, the polygon count will increase as the faces that are located along the edges of each smaller texture are cut to match the edges of the textures. The texture pattern number in the Face attributes of each affected face will be changed to the pattern number of the new texture that is applied.

Step 6. Set **Blend Edges** if you want Creator to adjust the pixel colors at the edges of each texture. This eliminates visible edges in some realtime systems.

Step 7. Click **OK**. **Chop Texture** creates four new texture files that correspond to the four grid quadrants. The new files are into named: [filename]_0.*, [filename]_1.*, [filename]_2.*, [filename]_3.*, where [filename] is the name of the original texture, and * is the file extension of the original texture.

Calculating Memory Requirements

Each component defined in a texture pattern (intensity, red, blue, green, and alpha) requires 8 bits (1 byte) of memory. Therefore, the number of bytes of texture memory needed to load a texture pattern is *(xsize) * (ysize) * (number of components)*, where *xsize* is the dimension in *x* and *ysize* is the dimension in *y*. For example, a 1024 x 1024 RGB_Alpha texture pattern requires 1024 * 1024 * 4 bytes (4 MB) of memory.

The formula for calculating the size of each supported texture pattern is as follows:

- For Intensity patterns:
  # of bytes of texture = *xsize* * *ysize**1
- For Intensity_Alpha patterns:
  # of bytes of texture = *xsize* * *ysize**2
- For RGB patterns:
  # of bytes of texture = *xsize* * *ysize**3
- For RGBA (RGB_Alpha) patterns:
  # of bytes of texture = *xsize* * *ysize**4

The dimension of every texture pattern (in *x* and *y*) must be a power of two. For example, a pattern can be 16 x 16, 16 x 32, etc. If you load an image that does not meet this requirement, it is automatically resized to the next higher power of 2. This does not affect its appearance, but may result in inefficient use of texture memory.

If any of these minification filters are used, the texture size is increased by one third.

- Mipmap Point
- Mipmap Linear
- Mipmap Bilinear
- Mipmap Trilinear

For example, a 32 x 32-pixel RGB_Alpha texture displayed using **Mipmap Trilinear** requires 32 x 32 x 4 x 1.33333 (5462 bytes) of memory.

Creator retains a list of all textures loaded for all the databases opened during a session. If two databases reference the same texture pattern, it is accessible to both databases, but only loaded into memory once. If both databases are closed, but Creator is still running, the loaded textures remain in memory, and do not have to be r-reread from disk when another database that references those patterns is loaded.

Tips for Working with Textures

The following are helpful tips for working with textures in Creator.

- Textures with a transparency (alpha) channel may display a light colored border, or halo, when applied. To eliminate this border, open the texture file in an image editor and set the color of the alpha pixels to the average color of the opaque pixel edges.
- Apply a texture pattern to all levels of detail within a database to ensure texture mapping consistency. To do this, select the top node, open the **Hierarchy** toolbox and click **Toggle Display** twice. This action turns the display of all geometry off and then on. All your LODs will be displayed and selected in the Graphics view, and you can then apply your texture.
- You can use the **Attribute Search** function to search for, and apply texture to, polygons with specific attributes (a terrain flag, for example)
- When textures are applied to polygons in a model, they should be mapped so that the resolution is approximately the same as the size the textures represent.

For example, if you apply a brick texture, where the bricks would be four inches by two inches, apply the texture pattern so that each brick covers approximately a two by four inch area of the polygon. This prevents unnatural visual effects when different textures are applied in close proximity to each other.

- Use high-resolution textures only if a scene will be viewed close-up. A texture resolution for a scene viewed at 3 meters or greater is approximately 0.005m/texel. For viewing distances of 100 meters or more, 0.164m/texel is a reasonable resolution.
- Size the textures (***u*** and ***v*** dimensions) to powers of 2 (32, 64, 128, an so on). Because some hardware cannot accurately display irregularly sized textures, this prevents mismatching problems.
- Use texture collages to apply texture patterns that do not repeat. They use less memory and are more efficient to process.
- For very large images (greater than 2048 x 2048), you can use the `mgattredit` utility to calculate the size of an image file, create a new `.attr` file and edit the image attributes without loading the file into memory (see the *MultiGen Creator Release Notes*).

Exercise: Applying a Texture

In the following exercise you will map a texture to a model and modify the texture after it has been applied.

Step 1. Open the database file `house.flt` in the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory.

Note: If you did not install the exercise files when you installed Creator, you can open this file directly from the Creator installation CD.

Step 2. Choose **LOD/Most Levels of Detail** (Ctrl+Shift+M) to display all levels of detail in the model.



Step 3. Click **Draw Texture** in the View panel (T) to display textures in the database.

Step 4. Choose **Palettes/Texture** to open the Texture palette.

Step 5. Choose **File/Read Pattern**.

Step 6. Enter an offset in the *Read New Pattern* dialog box or click **OK**. The **Read Pattern** dialog box appears.

Step 7. Select a new texture pattern such as `bigT.rgb` in the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory. The selected texture becomes the current texture.

Note: If you did not install the exercise files when you installed Creator, you can open this file directly from the Creator installation CD.

Step 8. Close the Texture palette.

Step 9. In Face mode (Shift+F), select a face that you want to texture, such as the side of the house.



Step 10. Open the View panel and click **Trackplane From Face**. Select the face to be textured.



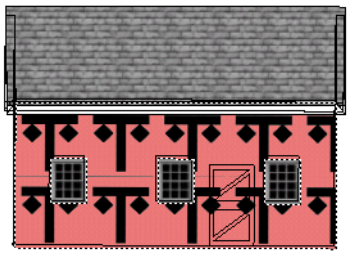
Step 11. Turn on the Grid and set the grid size to 1.



Step 12. Select **Put Texture** in the **Texture** toolbox.

Step 13. Set the **Origin** point to the lower left corner of the face, and the **Alignment** point two units to the right of the Origin. You do not need to set the **Shear** point.

Step 14. Click **OK**. The texture is applied to the face.





Step 15. Open the **Modify Texture** toolbox.



Step 16. Click **Translate Texture**.

Step 17. Click a position for the **To Point** on the selected face and click **OK** to move the texture pattern.

MultiTexture

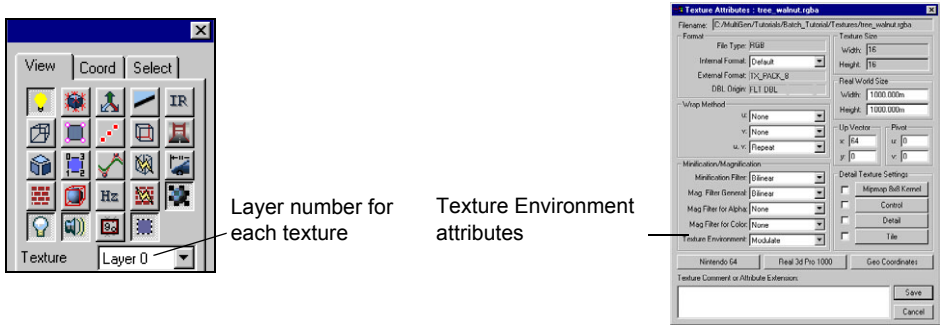
MultiTexture is an extension to face attributes. It applies textures in layers – up to 8 – to a face or faces in any combination.

Using MultiTexture, you can add local detail without adding additional polygons. For example, to show a hole in a wall, a brick pattern texture can be applied to the whole wall, and a texture of a hole with cracks applied to one part as another layer. MultiTexture is also useful when you have applied a repeating texture and want to make it appear random, such as clouds.

You apply MultiTexture layers using the View Control panel. The layers can be any supported graphic format, but textures must be either RGB-Alpha or Intensity-Alpha for blending to occur. You can blend textures with other textures or polygon colors using the Texture Environment algorithms (**Modulate, Blend, Decal, Replace**) in the *Texture Attributes* window.

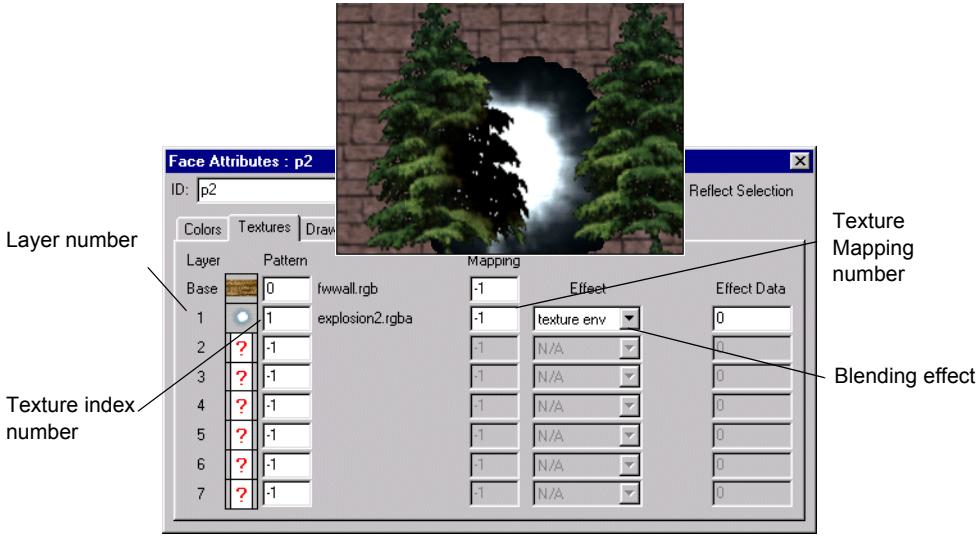
Note: Your hardware must be able to display multiple textures in order to see the blending effect.

To apply MultiTexture layers, you first select Layer 0 in the View Control panel, select the face to which it will be applied, and then apply a texture using Insert Texture or a texture mapping tool. To add a new layer, select a new layer number in the View Control panel, and apply the texture in the same manner. For each texture, you can open the *Texture Attributes* window from the Texture palette to set the blend algorithm. The texture blends with the base layer or other subsequent layers using the blend algorithm.



The layer number, texture index number, texture mapping number, and blending effect (**texture environment, bump map, custom)** are stored as face attributes in the **Textures** panel. Choose **texture environment** (the default blending effect) if you are using the Texture Environment attributes or **bump map** if your runtime system supports bump maps. You can select **custom** if you have created your own effect to display in the runtime system.

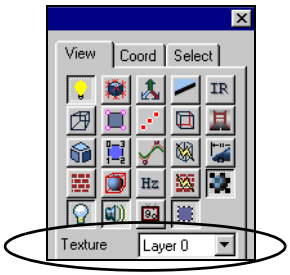
As with all textures, you can add the mapping scheme to the Texture Mapping palette. The Texture Mapping number can be instantly applied to any MultiTexture layer in the *Face Attributes* window.



To Apply Multiple Textures

Step 1. Load all the textures you want to apply into the Texture palette and select the texture you want to apply as the Current Texture.

Step 2. Open the View Grid Controls and select **Layer 0** from the **Texture** section.





Step 3. In Face mode, select the target face.

Step 4. Apply the base texture to the face using **Insert Texture** (Ctrl+T) or a texture mapping tool.



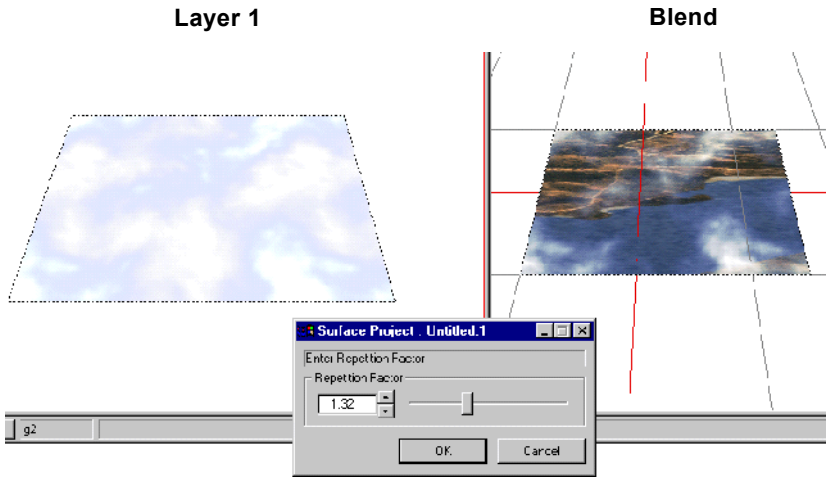
Step 5. Turn on **Display Texture** (T) to view the texture.

Step 6. Select another layer (1 through 7) from the **Texture** section in the View Grid controls. The new layer on the selected face will appear untextured.

Step 7. Select another texture from the Texture palette.

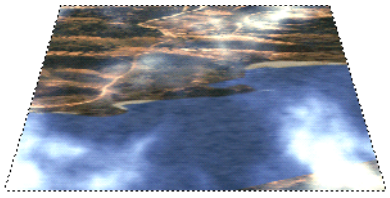
Step 8. Apply the texture to the face using Insert Texture (Ctrl+T) or a texture mapping tool.

Note: As an aid in placing your next texture, you can view both texture patterns together by splitting the graphics view window, selecting one pane and selecting **Blend** in the View Grid controls. Then, when you activate the other pane to apply the next layer, you can more accurately place and control its mapping.



Step 9. In the View Grid controls, select **Blend** in the **Texture** section to view the blended textures.

Step 10. Choose **Edit/Modify Attributes** from the Texture palette to open the *Texture Attributes* window. You can select a blend algorithm (**Modulate, Blend, Decal, Replace**) for the texture to blend it with the first texture.

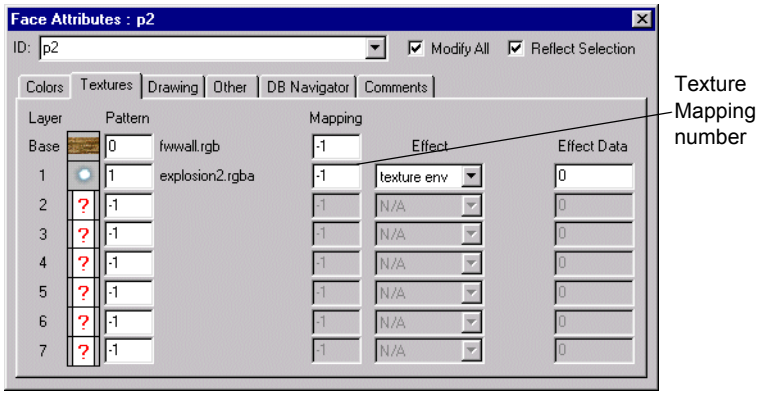


Texture Mapping with MultiTexture

You can instantly apply a texture layer to a selected face with an existing Texture Map. To do this:

Step 1. Select the target face.

Step 2. Open the *Face Attributes* window. Click the **Textures** tab.



Step 3. Delete the number in the appropriate Mapping field and enter the Texture Mapping number. The face will update automatically.

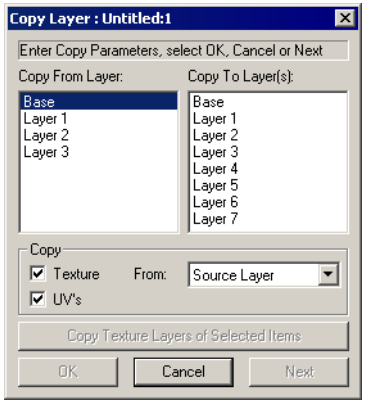
Copying Texture Layers

You can copy texture information (texture index, u,v coordinates, or both) from one layer of your geometry to one or more layers. You specify the source layer, destination layer, and the texture information that you want to copy.

To Copy Layers

Step 1. Select the textured geometry in the Graphics view. You can select either polygons or a mesh that has texture applied in one or more layers.

Step 2. Choose **Palettes/Texture Layers/Copy** to open the *Copy Layer* dialog box.



Step 3. In the **Copy From Layer** list, select the source texture layer that you want to copy. The selected source layer displays on the geometry in the Graphics view.

Step 4. Set either the **Texture**, **UV's**, or both that you want to copy from the source layer. If you want to copy the *u*,*v* coordinates but select a different texture pattern, for example, you can set only **UV's**. By default, both **Texture** and **UV's** are set.

Step 5. If you want to copy the currently selected texture pattern in the palette instead of a texture layer that is on your geometry, you can choose **Current Pattern** in the **From** list. By default, **Source Layer** is chosen.

Step 6. Select the destination layers in the **Copy To Layer(s)** list. You can copy to a single layer or to multiple layers at once. To select multiple layers, hold the Shift key down and drag over more than one layer using the left mouse button.

Step 7. After you have specified the copy layer, you can click **OK** to apply the changes and close the dialog box, click **Cancel** to close the dialog box without applying changes, or **Next** to apply the changes and leave the dialog box open. If you click **Next**, you can choose another layer to copy, or select a new set of geometry to work with.

After you select your next copy layer, you can click **Copy Texture Layers of Selected Items** to apply changes instead of clicking **Next**.

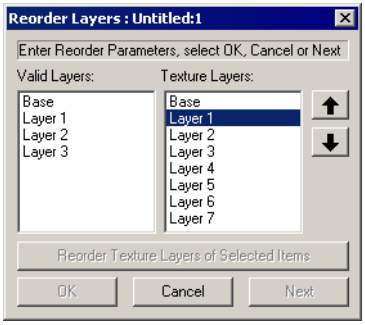
Reordering Texture Layers

You can change the order of the texture layers on your geometry. You select the layer that you want to move, and then click the up and down arrow to move the layer in the desired direction.

To Reorder Layers

Step 1. Select the textured geometry in the Graphics view. You can select either polygons or a mesh that has texture applied in one or more layers.

Step 2. Choose **Palettes/Texture Layers/Reorder** to open the *Reorder Layers* dialog box. The valid layers in the selected geometry display in the **Valid Layers** list.



Step 3. In the **Texture Layers** list, select the texture layer that you want to move. The selected source layer displays on the geometry in the Graphics view.

Step 4. Click the up or down arrow to move the layer to the desired location.

Step 5. After you have reordered the texture layers, you can click **OK** to apply the changes and close the dialog box, click **Cancel** to close the dialog box without applying changes, or **Next** to apply the changes and leave the dialog box open. If you click **Next**, you can choose another layer to move, or select a new set of geometry to work with.

After you select your next layer to move, you can click **Reorder Texture Layers of Selected Items** to apply changes instead of clicking **Next**.

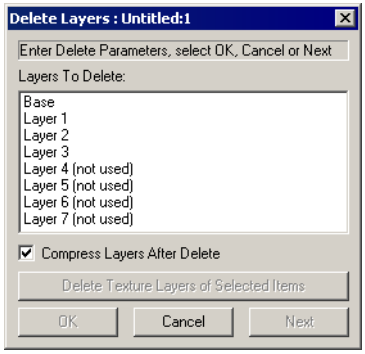
Deleting Texture Layers

You can delete one or more texture layers from your geometry. You select the layers that you want to delete. You can also compress the remaining texture layers.

To Delete Layers

Step 1. Select the textured geometry in the Graphics view. You can select either polygons or a mesh that has texture applied in one or more layers.

Step 2. Choose **Palettes/Texture Layers/Delete** to open the *Delete Layers* dialog box. The layers that you can delete are indicated in the **Layers to Delete** list.



Step 3. In the **Layers to Delete** list, select the texture layer that you want to delete. The selected source layer displays on the geometry in the Graphics view.

Step 4. Set **Compress Layers After Delete** to compress the remaining texture layers after the selected layers are deleted. This option moves textures in layers above those which were removed to lower indices which are vacant once layers are removed. For example, if a polygon has textures in layers 0, 1, 2, 3, and 4, you can delete layers 1 and 3. If **Compress Layers After Delete** is not set, the resulting polygon will have textures in layers 0, 2, and 4; if the box is checked, the layers will be compressed: 2 will be moved to 1 and 4 will be moved to 2, so that the resulting polygon will have textures in layers 0, 1, and 2.

Step 5. After you have deleted the texture layers, you can click **OK** to apply the changes and close the dialog box, click **Cancel** to close the dialog box without applying changes, or **Next** to apply the changes and leave the

dialog box open. If you click **Next**, you can choose another layer to delete, or select a new set of geometry to work with.

After you select your next layer to delete, you can click **Delete Texture Layers of Selected Items** to apply changes instead of clicking **Next**.

Subtexture

Subtexture lets you define a smaller area of a texture to apply to a face. When you define the subtexture, the **From** points that you use with the Texture Put tools (**3-Point Put** and **4-Point Put**) are also saved with the texture. The information for each subtexture is saved in the main texture's attribute (.attr) file, which includes the name, origin x, origin y, width and height.

Subtextures are useful for applying sections of a texture to different areas of a model. For example, you can define individual sections of a texture to apply to different areas of a house instead of using multiple textures. The use of subtextures saves system resources because the runtime uses fewer texture indices and names, and less switching of the currently bound texture.

You can group individual texture patterns into a single texture, or texture collage, that you can load into the Texture palette. Each texture pattern can be fence-selected and used as a subtexture. You can create the texture collage outside of Creator using Adobe PhotoShop, GIMP, or another image editor. You can use the Texture Compositor in the **Palettes** menu to group textures on selected geometry into a single texture. See "The Texture Compositor" on page 586 for more information.

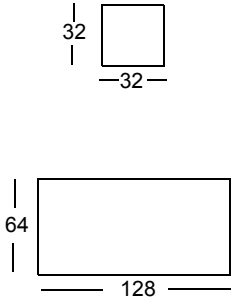
Subtexture u and v values are set with respect to the main texture. Because of this, they cannot be repeated across a face because the mapping would be incorrect.

Note: There is a trade-off between the size of the composite texture and the amount of memory it takes to load it and store it in memory. There is a point at which it is not beneficial to group the textures into one texture file. If you know what type of video card you are using you can check the card's available texture memory to determine whether your textures stretches or exceeds the memory allowed. See "Using Chop Texture to Reduce Texture Size" on page 464 for more information about calculating texture memory.

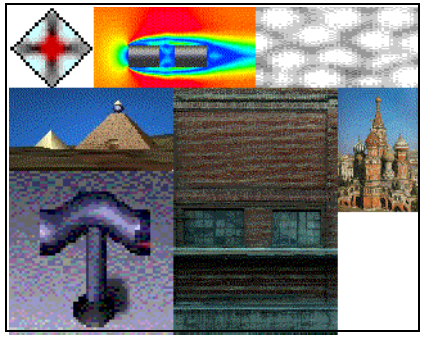
Creating a Texture Collage for Subtextures

It is efficient to group non-repeating texture patterns into a single composite texture called a texture collage. You can load this composite texture into Creator's Texture palette as one texture. Each texture pattern can be fence-selected and used as a subtexture.

You create a texture collage outside Creator using an image editor, such as Adobe Photoshop® or GIMP (Gnu Image Manipulation Program - see "Setting an Image Editor (IRIX)" on page 547). The subtextures must all be of the same type, such as RGB, and they must all contain the same texture attributes. The collage size can range from 8 to 256 texels, as long as the length of each side is a factor of 2 (8, 16, 32, 64, etc.). To be most efficient, the subtextures should completely fill the collage, leaving no unfilled space or borders.



Collage size must be a factor of 2



Avoid unfilled space in the collage

Defining and Applying Subtextures

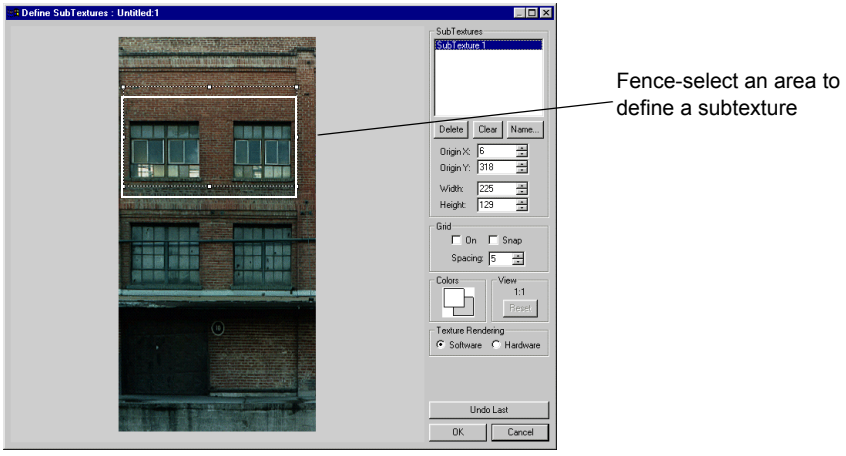
You can define one or more areas in the current texture as subtextures. Subtextures are saved with the main texture and can be applied to faces using the Put Texture tools. Vertices of the subtextures are saved as **From** points that you use to map to points on a face.

To define and apply a subtexture:

Step 1. Select a texture in the *Texture Palette* window as the current texture.

Step 2. Choose **Palettes/Define Subtextures** to open the *Define Subtextures* window.

Step 3. Fence-select an area in the texture with the left mouse button. The coordinates of the lower left corner of the subtexture rectangle appear in the **Origin X** and **Origin Y** fields.

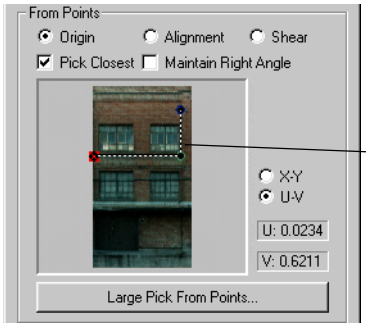


Step 4. Click the **Name** button in the **Subtextures** section to name the subtexture.

Step 5. In the **Grid** section, you can set the **Grid** checkbox to activate the grid, and set the **Snap** checkbox to snap the vertices of the subtexture to the grid. You can also define the spacing of the grid units.

Step 6. In the Texture palette, select a subtexture as the current texture. The subtexture rectangle appears in the main texture. The main texture will appear as the current texture on the Toolbar, however.

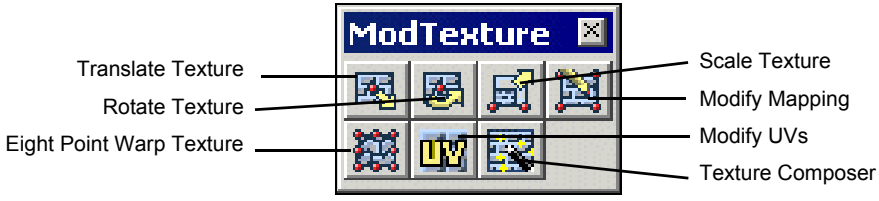
Step 7. Select either the **3-Point Put** or **4-Point Put** texture tool to map the subtexture to a face. You will notice that the **From** points that correspond to the subtexture rectangle are already selected on the main texture.



In the *Put Texture* window, the **From** points for the subtexture are already selected

Modify Texture Tools

The tools in the **Modify Texture** toolbox are used to adjust a texture's mapping any time after a texture has been applied.

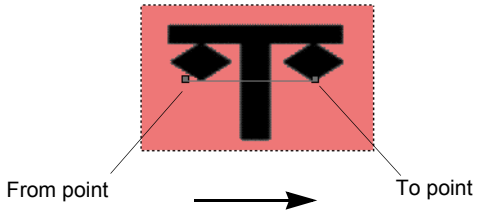


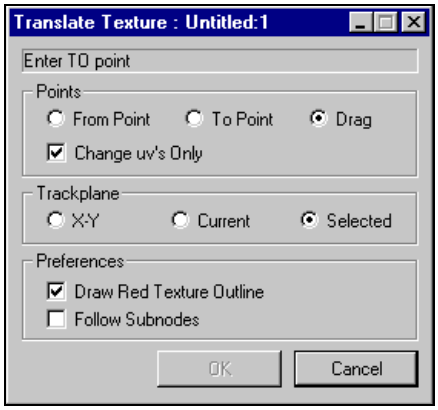
"The Texture Composer tool in the ModTexture toolbox is a powerful open-ended tool for creating, applying, and manipulating textures for models. The tool unwraps a model into a 2D texture map workspace that represents a u,v mapping for the geometry." on page 499

Translate Texture Tool



Translate Texture moves the applied texture pattern along a vector between two points.





To Translate an Applied Texture

Step 1. In Face mode, select the faces to which the texture is applied.



Step 2. Open the **Modify Texture** toolbox.



Step 3. Click the **Translate Texture** tool. The *Translate Texture* dialog box opens with the **From Point** displayed.

Step 4. Specify a new **From Point**, if you want. This point defines the origin of the translation.

Step 5. Click a **To Point** to define the amount and direction of translation, relative to **From Point**.

Step 6. Click **OK**.

Translate Texture Parameters

The *Translate Texture* dialog box sets the following parameters.

From Point

Defines the origin of the translation. The vector between this point and the **To** point determines the distance and direction of the translation.

To Point

Defines the amount and direction of translation, relative to the **From** point.

Drag

Enables you to translate the texture by dragging it with the left mouse button.

Change uv's Only

Translates the image by changing the *u*,*v* values of the selected faces and sets the **Texture Mapping** attribute of each selected face to -1 to indicate that the faces no longer reference the Texture Mapping palette. If **Change uv's Only** is cleared, **Translate Texture** changes the corresponding texture mapping (if any) in the Texture Mapping palette, affecting the texture mapping of all faces to which that mapping is assigned.

Trackplane

Controls the definition of the tracking plane while using **Translate Texture**. When you close the *Translate Texture* dialog box, the tracking plane returns to its previous position. The following controls are available:

- **X-Y** sets the tracking plane to the xy plane.
- **Current** uses the current tracking plane definition in effect when you start **Translate Texture**.
- **Selected** sets the tracking plane to the first face in the Select list.

Draw Select Outline During Edit

Controls the display of the dotted outline that normally appears around selected faces. Suppressing the select outline makes it easier to position **To** points on small faces.

Draw Red Texture Outline

Controls the display of a red grid showing the position and orientation of the applied texture. The red outline appears automatically on computers that do not display dynamic texture.

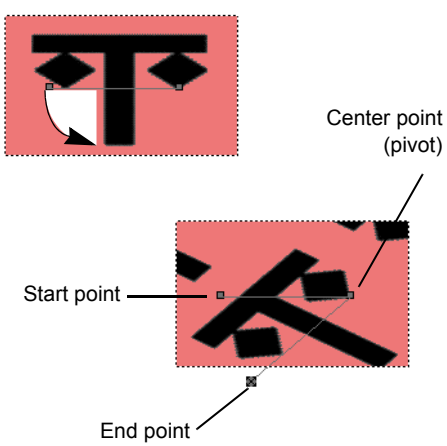
Follow Subnodes

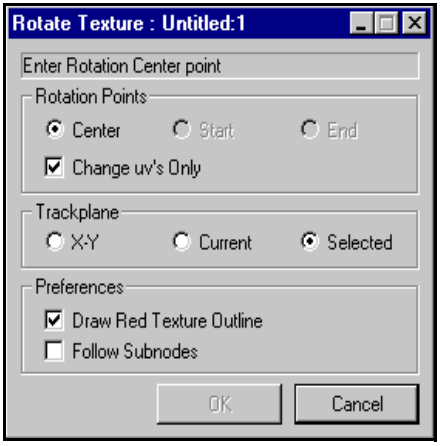
Controls whether subfaces are affected by the translation.

Rotate Texture Tool



Rotate Texture pivots an applied texture pattern about the **Center** of rotation from the **Start** point to the **End** point.





To Rotate an Applied Texture

Step 1. Select the faces to which the texture is applied.



Step 2. Open the **Modify Texture** toolbox.



Step 3. Click the **Rotate Texture** tool.

Step 4. Set the parameters in the *Rotate Texture* dialog box (see "Rotate Texture Parameters" on page 484).

Step 5. Click the positions for the **Center**, **Start**, and **End** points in the database.

Step 6. Drag the **End** point to the desired position.

Step 7. Click **OK**.

Rotate Texture Parameters

The *Rotate Texture* dialog box sets the following parameters.

Center

Specifies the center (pivot) point of rotation.

Start

Defines a reference edge for the rotation.

End

Defines the angle of rotation, relative to the reference edge.

Change uv's Only

Rotates the image by changing the *u*,*v* values of the selected faces and sets the **Texture Mapping** attribute of each selected face to -1 to indicate that the faces no longer reference the Texture Mapping palette. If **Change uv's Only** is cleared, **Rotate Texture** changes the corresponding texture mapping (if any) in the Texture Mapping palette, affecting the texture mapping of all faces to which that mapping is assigned.

Trackplane

Controls the definition of the tracking plane while using **Rotate Texture**. When you close the *Rotate Texture* dialog box, the tracking plane returns to its previous position. The following controls are available:

- **X-Y** sets the tracking plane to the xy plane.
- **Current** uses the current tracking plane definition in effect when you start **Rotate Texture**.
- **Selected** sets the tracking plane to the first face in the Select list.

Draw Red Texture Outline

Controls the display of a red grid showing the position and orientation of the applied texture. The red outline appears automatically on computers that do not display dynamic texture.

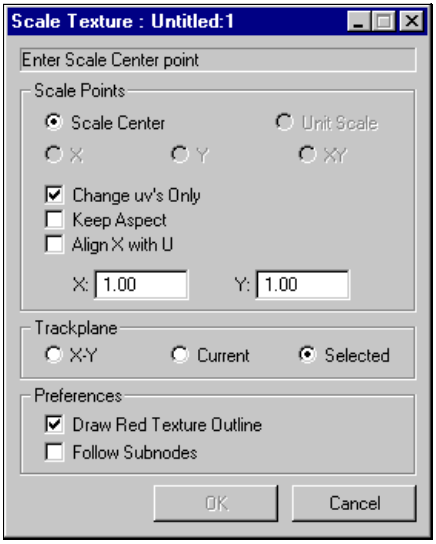
Follow Subnodes

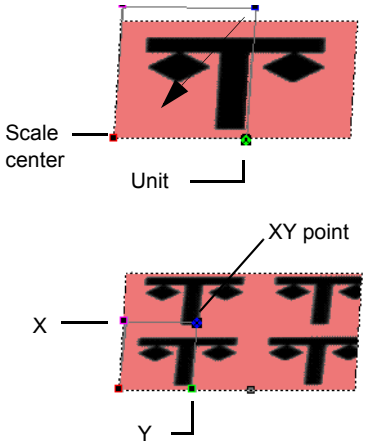
Controls whether subfaces are affected by the rotation.

Scale Texture Tool



Scale Texture scales an applied texture pattern along the *x* or *y* axis or along both axes simultaneously.





To Scale an Applied Texture

Step 1. In Face mode, select the faces to which the texture is applied.



Step 2. Open the **Modify Texture** toolbox.



Step 3. Click the **Scale Texture** tool.

Step 4. Set the controls in the *Scale Texture* dialog box.

Step 5. Click the positions for the **Scale Center** and **Unit Scale** points in the database.

Step 6. Click **XY** and drag to proportionately scale the texture or click **X** or **Y** to scale the texture in either axis alone.

Step 7. Click **OK**.

Scale Texture Parameters

The *Scale Texture* dialog box sets the following parameters.

Scale Center

Defines the origin of the *xy* texture-mapping axes.

Unit Scale

Marks an original aspect ratio for subsequent scale operations. Defines both the orientation of the *x* axis and the point where the scale in *x* is equal to 1.0. The unit scale along the *y* axis is automatically placed the same distance from the scale center (perpendicular to the *x* axis).

X,Y, XY (scale points)

Selects scale control points in the Graphics view. **X** scales the texture pattern along the *x* axis, **Y** along the *y* axis, and **XY** along both axes simultaneously. You can also scale the texture pattern numerically, by entering numbers in the **X:** and **Y:** fields.

Change uv's Only

Scales the image by changing the *u,v* values of the selected faces and sets the **Texture Mapping** attribute of each selected face to -1 to indicate that the faces no longer reference the Texture Mapping palette. If **Change uv's Only** is cleared, **Scale Texture** changes the corresponding texture mapping (if any) in the Texture Mapping palette, affecting the texture mapping of all faces to which that mapping is assigned.

Keep Aspect

Freezes the current relationship or ratio of *x* to *y*. For example, selecting this flag when the **X Scale** is 1.0 and the **Y Scale** is 0.5 fixes the ratio of *x* to *y* at 2:1.

Align X with U

Aligns the **Unit Scale** *x* with the texture pattern *u* to prevent accidental texture distortion. Select **Align X with U** before selecting **Unit Scale**.

X: and Y: fields

Numerically determine the scale in *x* and*y*.

Trackplane

Controls the definition of the tracking plane while using **Scale Texture**. When you close the *Scale Texture* dialog box, the tracking plane returns to its previous position. The following controls are available:

- **X-Y** sets the tracking plane to the xy plane.
- **Current** uses the current tracking plane definition in effect when you start **Scale Texture**.
- **Selected** sets the tracking plane to the first face in the Select list.

Preferences

- **Draw Red Texture Outline** controls the display of a red grid showing the position and orientation of the applied texture. The red outline appears automatically on computers that do not display dynamic texture.
- **Follow Subnodes** controls whether subfaces are affected by the rotation.

Modify Mapping Tool



Modify Mapping starts the texture mapping tool associated with the selected texture pattern indicated in the **Type** field in the Texture Mapping palette (see "The Texture Mapping Palette" on page 516).

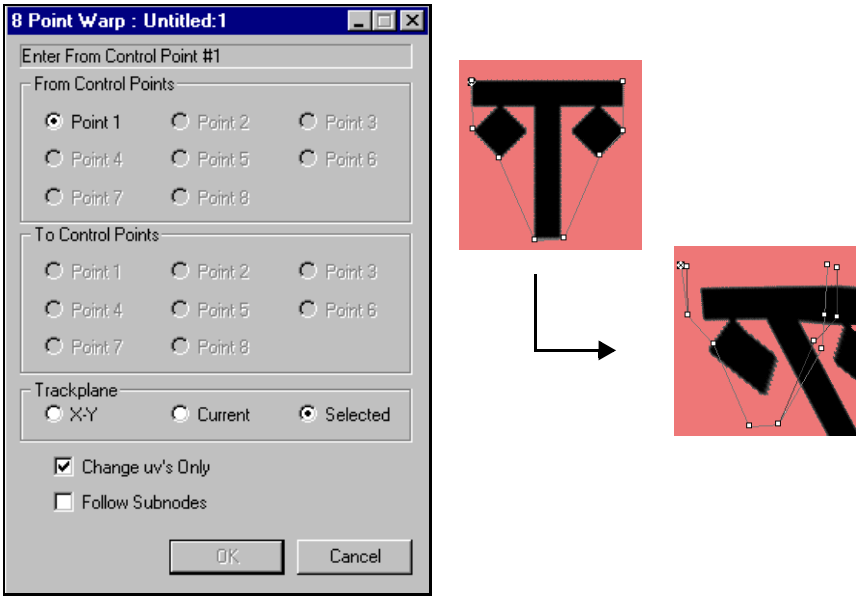
> **Note:** As long as the texture's **Type** has not changed, the tool restarts with the same parameter values in effect as when the texture mapping was saved. If the **Type** has changed, the tool restarts with default parameter values.

Clicking **Modify Mapping** in the **Modify Texture** toolbox is equivalent to choosing **Edit/ Modify Mapping** in the Texture Mapping palette.

Eight Point Warp Texture Tool



Eight Point Warp Texture remaps an applied texture using eight control points.



To Remap a Texture Using Eight Point Warp Texture

Step 1. Select the faces to which the texture is applied.



Step 2. Open the **Modify Texture** toolbox.

Step 3. Click the **Eight Point Warp Texture** tool.

Step 4. Set the controls in the *Eight Point Warp Texture* dialog box.

Step 5. Click the positions for eight **From Control Points** and then click the corresponding positions for the **To Control Points** in the database. Click **OK**.

Eight Point Warp Texture Parameters

The *Eight Point Warp Texture* dialog box sets the following parameters.

From and To Control Points

Remaps the applied texture pattern. Each **From** Control Point maps to the corresponding **To** Control Point in the database.

Trackplane

Controls the definition of the tracking plane while using **Eight Point Warp Texture**. When you close the *Eight Point Warp Texture* dialog box, the tracking plane returns to its previous position. The following controls are available:

- **X-Y** sets the tracking plane to the xy plane.
- **Current** uses the current tracking plane definition when you start **Scale Texture**.
- **Selected** sets the tracking plane to the first face in the Select list.

Change uv's Only

Warps the image by changing the *u*,*v* values of the selected faces and sets the **Texture Mapping** attribute of each selected face to -1 to indicate that the faces no longer reference the Texture Mapping palette. If **Change uv's Only** is cleared, **Eight Point Warp Texture** changes the corresponding texture mapping (if any) in the Texture Mapping palette, affecting the texture mapping of all faces to which that mapping is assigned.

Follow Subnodes

Controls whether subfaces are affected by the rotation.

Modify UVs Tool



Modify UVs allows you to edit texture coordinates on vertices directly in UV space. It displays the UV coordinates of selected vertices in the 2D image space of the texture applied, and lets you modify those UV coordinates using techniques such as drag, align, snap, rotate, and scale.

To Modify UVs

Step 1. Select the polygon(s) containing the vertices of the UV texture coordinates you want to edit. All selected polygons should have the same texture applied.

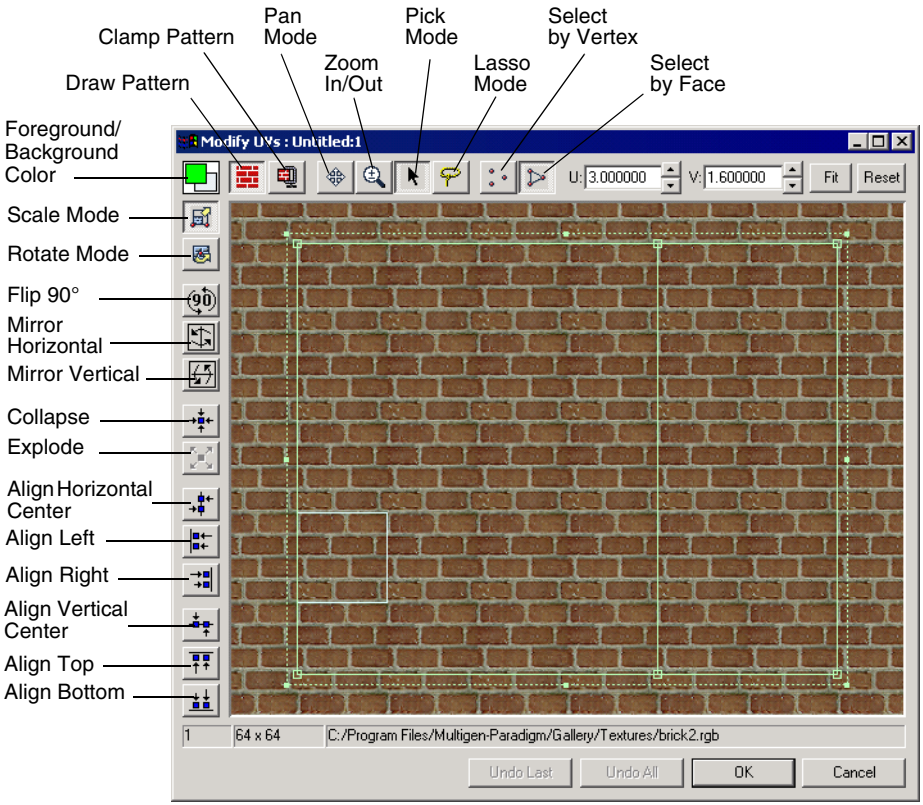
When it is invoked, the Modify UVs tool searches the select list for the first polygon that has a texture applied. Using the index of the texture applied to the first polygon it finds, it continues searching the select list for all other polygons that have the same texture applied. All polygons that do not have this texture applied are not considered for editing.



Step 2. Open the **Modify Texture** toolbox.



Step 3. Click the **Modify UVs** tool. The **Modify UVs** dialog box opens, providing numerous controls for changing the UV coordinates of selected vertices.



The UV view within the dialog box displays the polygons that share the same applied texture. The vertices (points) of the polygons are plotted using their UV coordinates, rather than their XYZ coordinates. Each

polygon appears as a wire frame outline, and each point is represented by a small solid square.

The UV view displays the UV coordinates in the 2D image space of the texture, and the texture pattern is drawn in the background. The lower left corner of the pattern has the UV coordinates (0,0), and the upper right has the coordinates (1,1).



The wireframe and points of the polygons are drawn in the color indicated by the foreground swatch in the upper left corner of the dialog box. The outline of the texture pattern is drawn in the color indicated by the background swatch in the upper left corner of the dialog box. If you are using a texture which makes the default colors hard to see, you can change colors by clicking on a swatch; when you click on a swatch, a color picker is displayed, providing the means to set a new color.

Step 4. Select the vertices you want to modify.



You can use either the **Pick Mode** or **Lasso Mode** to select vertices. In **Pick Mode**, select vertices individually by clicking on them, or drag a fence around multiple points by holding down the left mouse button. Simlarly, in **Lasso Mode**, select vertices individually by clicking on them, or lasso multiple points by holding down the mouse button and enclosing the vertices you want to modify. Additionally, if you press SHIFT while selecting a point that is not yet selected, you select that point without deselecting any points already selected. Conversely, if you press SHIFT while selecting a point that is already selected, you deselect that point without affecting any other selected points.



Your selection method is also governed by the selection controls at the top of the dialog box. When **Select by Vertex** is active, only individual vertices will be selected when you use the selection tools. When **Select by Face** is active, all the vertices of selected faces will be included.

Step 5. Modify the selected vertices (points), using the additional controls in the **Modify UV** dialog box to produce your desired results. The **Modify UV** controls can move, scale, rotate, and align the vertices with ease. (For more details on these operations, see “Moving Points in the UV View” on page 493, “Scaling Points in the UV View” on page 493, “Rotating Points in the UV View” on page 494, and “Alignment Tools” on page 495.)

Step 6. When you have finished making modifications, click **OK**; or choose **Cancel** to quit the operation without making changes.

Moving Points in the UV View

There are several ways to move points in the UV view. You can drag them using the mouse, type in UV coordinate values directly or use one of the many alignment tools. This section describes how to drag points around in the UV view. (To learn more about entering values directly or using the alignment tools, see "UV Coordinates" on page 496 and "Alignment Tools" on page 495.)



To drag points in the UV view, choose **Pick Mode** or **Lasso Mode**. You can drag a single point just as easily as you can drag a group of them. To drag a single point, press the left mouse button over the point you want to move (this will work whether or not the point is already selected) and simply drag the mouse to the new UV coordinate in the UV view. Release the mouse button when you are done dragging the point.

Before you can drag multiple points, you must first select all the points you want to move. When all the points you want to move are selected, press the left mouse button over any of the selected points and drag the mouse to the new UV coordinate. The point you pressed the mouse over will move to the new UV coordinate and all other selected points will move such that original spatial relationship between all the selected points is preserved.

Note: If you press CTRL while dragging, you can constrain the movement to a single axis.

Scaling Points in the UV View



To scale a group of points in the UV view, choose **Scale Mode**. Once you have selected multiple UV points, the select handles will be displayed. To scale the selected UV points, press the left mouse button over one of the select handles and drag it to a new location. How far you drag the handle determines how much the UV points are scaled. Also, which handle you drag determines the point about which the UV points are scaled. By default, the handle "opposite" the handle you drag is the point about which the UV points are scaled. For example, if you scale by dragging the upper right handle, the UV points are scaled relative to the lower left handle.

If you drag a corner handle, you can scale the UV points in two dimensions. If you drag an edge handle, you can scale the UV points in the dimension perpendicular to that edge (e.g., dragging the handle on either the left or right edge scales the UV points in the U direction only).

Note: If you press CTRL while scaling, you can scale about the center point of the selected UV points. This is useful for symmetric scaling effects.

Note: If you press SHIFT while scaling, the aspect ratio is maintained.

Rotating Points in the UV View



To rotate a group of points in the UV view, choose **Rotate Mode**. Once you have selected multiple UV points, the select handles will be displayed. To rotate the selected UV points, press the left mouse button over one of the select handles and drag it to a new location. How far you drag the handle determines how much the UV points are rotated. Also, which handle you drag determines the point about which the UV points are rotated. By default, the handle "opposite" the handle you drag is the point about which the UV points are rotated. For example, if you rotate by dragging the upper right handle, the UV points are scaled relative to the lower left handle.

Note: If you press CTRL while rotating, you can rotate about the center point of the selected UV points.

Changing the UV View

There are several ways to change the UV view. You can pan or zoom the image. You can fit on all or just the selected points. You can also resize the Modify UVs dialog; this resizes the UV view accordingly. (To learn how to fit within the UV view, see “View Buttons” on page 496.)



You can pan or zoom the image using either the left or right mouse button. To pan or zoom the image using the left mouse button, choose **Pan Mode** or **Zoom Mode**, respectively. In **Pan Mode**, dragging the left mouse within the UV view pans the image up, down, left or right. In **Zoom Mode**, dragging the left mouse up or down within the UV view zooms the image out and in, respectively. Note that both horizontal and vertical mouse motion affects panning. However, only vertical mouse motion (up and down) affects zooming.

Panning and zooming the image using the right mouse is very similar to using the left mouse. To pan or zoom using the right mouse button, you need only press the appropriate modifier key while dragging the right mouse. To pan, press SHIFT while dragging the right mouse. To zoom, press CTRL.

Note: The mouse motion model for panning and zooming the UV view (with either mouse) is identical to that used to pan and zoom a database within the Creator graphic view. If you are already comfortable using the right mouse motion model for panning and zooming a database, you will probably want to keep the UV view in pick mode so you can use the left mouse to select and move points and the right mouse to pan/zoom the image. This way, you will not have to change mouse modes to perform the different mouse operations.

Alignment Tools

These tools allow you to easily align and snap points to one another.

	**Draw Texture**	If this checkbox is selected, the texture pattern will be displayed in the UV view.
	**Clamp Pattern**	If this checkbox is selected, the texture pattern will be displayed repeated to fill the UV view. If this checkbox is not selected, only one repetition of the pattern is displayed.
	**Collapse**	Press this button to collapse the UV values of all selected points to the average UV values of the selected points. The average U and V values are the respective midpoints of the extents of the selected points.
	**Explode**	Press this button to "explode" coincident points. If you need to move coincident points independently, doing this will enable you to select the coincident points independently.
	**Align Left**	Press this button to set the U coordinates of all selected points to the minimum U value of the selected points.

	**Align Horizontal Center**	Press this button to set the U coordinates of all selected points to the average U value of the selected points. The average U value is the midpoint (in U) of the extents of the selected points.
	**Align Right**	Press this button to set the U coordinates of all selected points to the maximum U value of the selected points.
	**Align Bottom**	Press this button to set the V coordinates of all selected points to the minimum V value of the selected points.
	**Align Vertical Center**	Press this button to set the V coordinates of all selected points to the average V value of the selected points. The average V value is the midpoint (in V) of the extents of the selected points.
	**Align Top**	Press this button to set the V coordinates of all selected points to the maximum V value of the selected points.

UV Coordinates

This area is composed of two text fields that display the U and V coordinates of the selected point(s). When more than one point is selected, the values displayed in these text fields represent the average U and V values of the selected points.

U	You can enter a point's U value in this text field by typing or using the spin control.
V	You can enter a point's V value in this text field by typing or using the spin control.

View Buttons

These buttons are used to fit on different points within the UV view.

Fit	Press this button to fit the UV view to the extents of the selected points.

Reset	Press this button to fit the UV view to the extents of all the points.

Mouse Mode Buttons

The mouse mode buttons appear as push buttons but obey radio-button style behavior: that is, only one can be active at a time. You can use the mouse mode buttons to control which action is taken when you press the left mouse in the UV view. Selecting one of these radio buttons selects the corresponding mouse mode.

	**Pan Mode**	When this button is selected, you can drag/pan the contents of the UV view.
	**Zoom Mode**	When this button is selected, you can drag/zoom the contents of the UV view.
	**Pick Mode**	When this button is selected, you can select points in the UV view and drag them to new positions. When you drag the mouse to select points, a standard rectangular fence is used. This is the default action for the left mouse button when it is pressed in the UV view.
	**Lasso Mode**	When this button is selected, you can select points in the UV view and drag them to new positions. When you drag the mouse to select points, you can create an irregular shape inside which points will be selected.

Transform Mode Buttons

The transform mode buttons appear as push buttons but obey radio button style behavior: that is, only one can be active at a time. You can use the transform mode buttons to control which action is taken when you drag the select handles. You can

drag the select handles to either scale the selected UV points or to rotate them. Selecting one of these radio buttons selects the corresponding transform mode.

	**Scale Mode**	When this radio button is selected, dragging the select handles scales the UV points.
	**Rotate Mode**	When this radio button is selected, dragging the select handles rotates the UV points.

Color Swatches

This area displays color swatches to customize the display of the UV view. With certain texture patterns, the default colors used may be difficult to see. Use these swatches to select colors that work better for your specific pattern.

	**Foreground Color**	Specifies the color used to draw the polygons and points in the UV view. When you click the foreground color within the swatch, a standard color chooser dialog is displayed in which you can select a new foreground color to use.
	**Background Color**	Specifies the color used to draw the background of the UV view. When you click the background color within the swatch, a standard color chooser dialog is displayed in which you can select a new background color to use.

Control Buttons

Undo Last	Press this button to undo the last change you made.
Undo All	Press this button to undo all the changes you have made during this invocation of the tool. This button performs the same actions as the Cancel button but unlike the Cancel button which dismisses the dialog, this button leaves the dialog displayed.
Cancel	Press this button to undo all the changes you have made and dismiss the tool.
OK	Press this button to accept all the changes you have made and dismiss the tool.

Texture Composer Tool

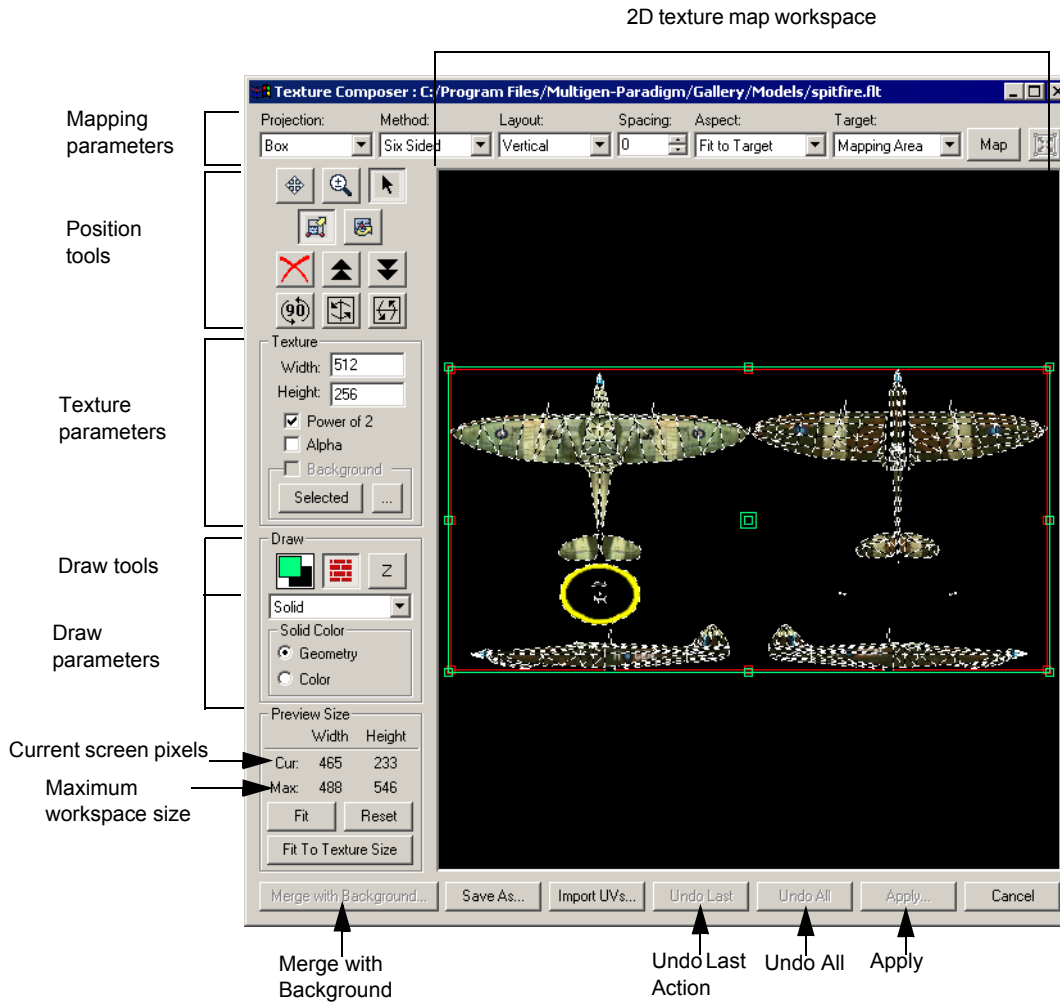


The **Texture Composer** tool in the **ModTexture** toolbox is a powerful open-ended tool for creating, applying, and manipulating textures for models. The tool unwraps a model into a 2D texture map workspace that represents a *u*,*v* mapping for the geometry.

The Texture Composer unwraps the geometry using a "best" projection method. For example, a Box format projects the six views (top, bottom, right side, left side, and back) of the model. You can easily change this projection, and you can also select individual faces to reproject using a different projection method. If the geometry is already textured, you can choose to keep the original *u*,*v* mappings, or display the geometry using a new projection.

There are many different ways to use the Texture Composer, but some common applications are:

- Generating a texture map template from geometry to create a new texture
- Modifying existing texture maps on geometry
- Creating a composite of multiple texture maps to save as a single texture



Creating a Template

One of the more powerful uses of the **Texture Composer** tool is for the creation of a texture template. A texture template consists of unwrapped geometry, typically in wireframe, projected as a 2D view in the texture *u,v* space. The texture template is saved as a RGB, RGBA, INT, or INTA texture file that is utilized as a guide in a 2D image editor, such as Creator's Texture Editor or Adobe Photoshop, to create a new texture map.

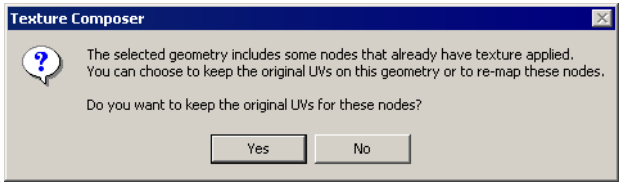
You can paint directly on the texture template, or manipulate a photo over the template, to create a new texture map. When you save the texture using the original texture name, the texture template that you previously applied on your geometry is replaced with your edited image in the Graphics view. Any subsequent changes made to the texture template will automatically show up on the 3D model in Creator the next time it is loaded or refreshed in the display.

To create a template:

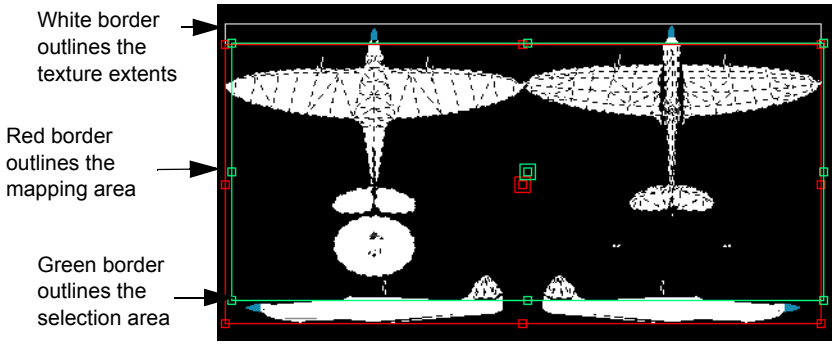
Step 1. Select the geometry in the Graphics view for the texture map template.



Step 2. Open the **Texture Composer** tool in the **ModTexture** toolbox. If the selected geometry is textured, the following message asks you if you want to to display the original texture *u,v* mappings as a background image.



Step 3. Click **No** to not display the original *u,v* mappings. The selected geometry is unwrapped into a flat 2D representation of the geometry.



The white border represents the *u,v* space that will become the texture extents; the red border represents the mapping area that receives any new

projections, or reprojections of geometry already in the workspace; and the green border (by default) outlines the selected area. You can change the color of the selection border, however.

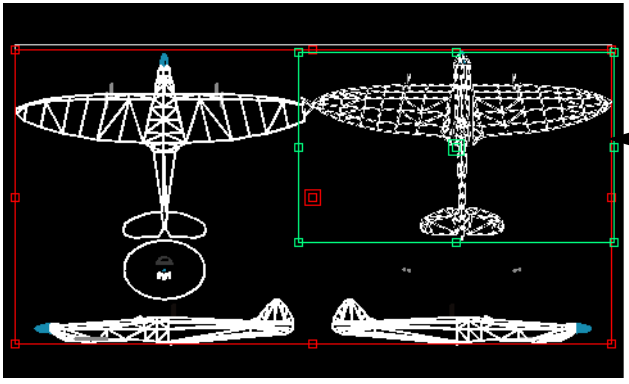
Note: By default, the mapping area is equal to the *u*,*v* space and covers the white border. You must resize the red border by dragging the control handles with the left mouse to see it.

Step 4. Change the Draw mode to **Wireframe**. **Wireframe** creates solid outlines in which you can paint.

Step 5. Set **Solid Color** to **Geometry**. The colors in the colored parts of your geometry will display instead of a different color that you select. The default color selected for the solid color (black) may blend with the background color, which hides parts of your geomtery from view.

Step 6. With the **Target** set to **Mapping Area**, choose a **Projection** method for your texture mapping needs. By default, the Texture Composer projects it using a "best" projection method.

Step 7. If you want to reproject geometric faces using a different type of projection, you can drag select them with the left mouse to create a new selection area. To add additional faces to your new selection area, press Shift while you drag select other faces.



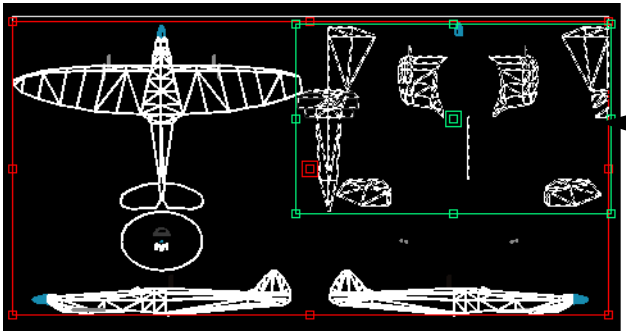
Selected faces can be reprojected using a different projection method.

When you choose **Selection Area** for the target, only those faces are reprojected.

Drag select the center box within the selection area to move it to a different location. This can be inside or outside of the mapping area. You can also press Shift and drag the control handles to scale the selection area or mapping area.

Step 8. Choose **Selection Area** for the target, and choose a Projection method. The selected faces will be automatically reprojected.

You can also move the mapping area, and then click **Map**. The selected geometry will then be added to the mapping area using the current Projection method.



Step 9. After you have laid out your geometry, select the **Width** and **Height** values for the texture size that is appropriate for your model. For example, a larger texture size results in more detail in your texture, but will require more texture memory in your simulation.

By default, the texture size will be set with **Power of 2**. A texture size with both dimensions that are a power of 2 (512 x 256, or 1024 x 1024, for example) is optimal for texture memory usage.

Step 10. If you want to see the actual texture size, click **Fit To Texture Size**. The screen resolution will be matched to your texture size.

Step 11. Set **Alpha** to add a transparent layer to your texture. This may be useful if you are going to be laying it over an image in an image editor, or...**[other reasons??]**.

Step 12. Click **Save As** to save the template as a texture file. You can save the texture in either RGB, RGBA, INT, or INTA format. If you have added an alpha channel, be sure to save the texture in either RGBA or INTA file format.

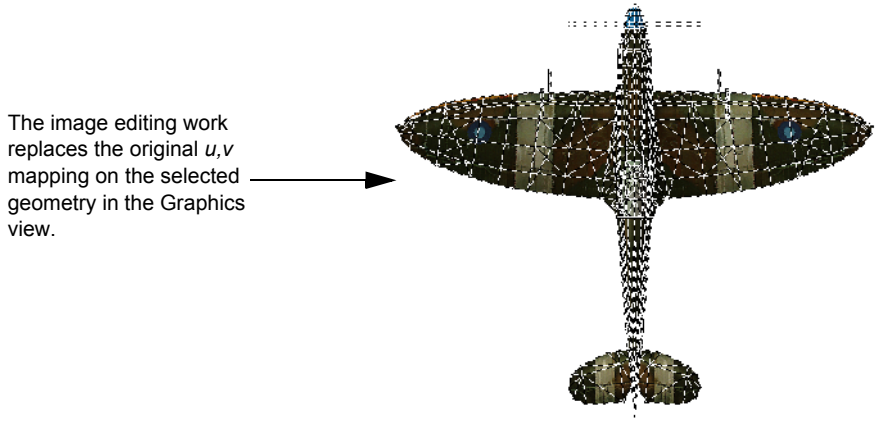
Step 13. Click **Apply** to "burn" the *u*,*v* coordinates of your template onto your geometry in the Graphics view.

You can choose a specific texture layer in the View panel to apply the texture to. For more information about using multiple texture layers on a model, see “MultiTexture” on page 470.

Note: You can select a different texture layer on the geometry as a test area for your new texture. If you want to reapply it to Layer 0 (the base layer), click **Apply** again.

Step 14. After you have applied your texture template, you can import it into an image editor, such as the Texture Editor or Adobe PhotoShop and apply photos or paint it. Be sure to maintain the original pixel aspect ratio and texture size so that the texture appears as expected on the geometry.

When you save the texture using the original texture name, the texture template that you previously applied on your geometry is replaced with your edited image in the Graphics view. You can open the Texture palette to verify that the texture is saved with the model.



Modifying a Texture Mapping

You can easily modify a texture in the **Texture Composer** tool that has been already mapped to geometry. The texture could have been mapped using any of the texture application tools, including the Texture Composer. When you apply the modified texture in the tool, the modified texture is applied to the geometry in the Graphics view.

Note: The selected geometry must have only one texture applied. If the geometry has multiple textures, only one of the images will display properly in the Texture Composer. All other texture mappings will be lost.

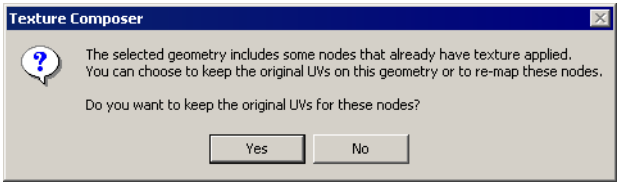
To modify an existing texture mapping:

Step 1. Select the geometry in the Graphics view with the texture that you want to modify.



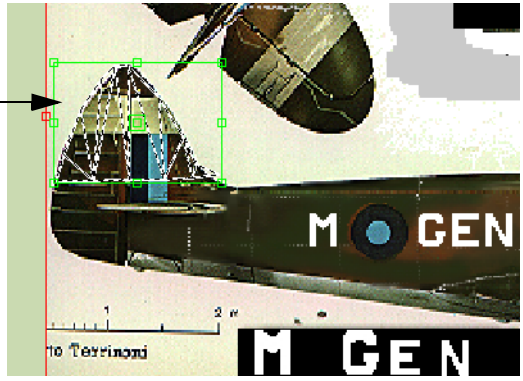


Step 2. Open the **Texture Composer** tool in the **ModTexture** toolbox. If the selected geometry is textured, the following message asks you if you want to keep the original *u*,*v* texture mappings.



Step 3. Click **Yes** to display the original *u,v* mappings. The selected geometry will display as a flat, 2D image with its *u,v* mapping. Choose **Wireframe**, and turn **Draw Mode** off so that you can see where to reposition geometry.

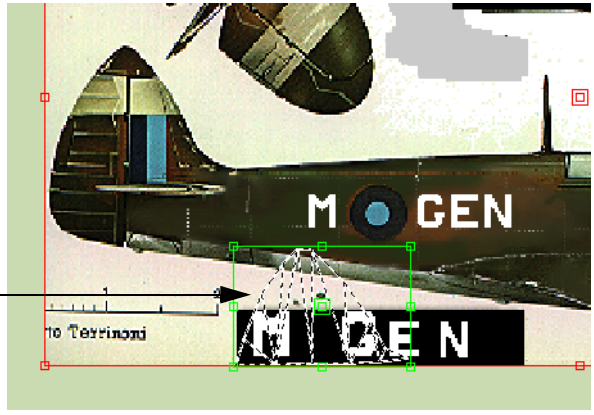
The tail object displays in wireframe mode with its original u,v texture mapping.



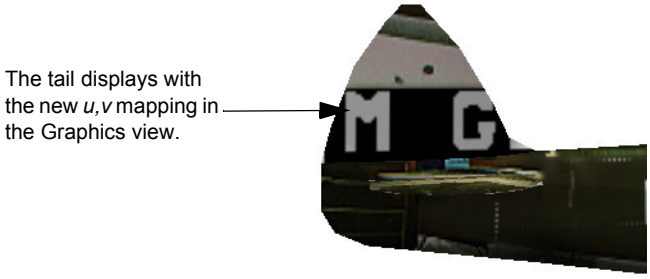
Note: If any of the selected faces were not textured, then those faces will appear as the default selected faces in the *Texture Composer* window. This is to indicate the faces that need to be textured.

Step 4. You can manipulate the texture mapping on the selected faces. For example, you can reshape and move it using the position tools, such as **Rotate Mode**, **Scale Mode**, **Pan** and **Zoom**, and mapping parameters, such as **Layout**, **Spacing**, and **Aspect**. See "Texture Composer Parameters" on page 510.

The geometry image was moved to a different location in the texture mapping. The geometry image will be burned into the *u,v*'s when applied.



Step 5. After you have modified the texture mapping, click **Apply**. The geometry image is burned into the new texture.



You can choose a specific texture layer in the View panel to apply the texture to. For more information about using multiple texture layers on a model, see "MultiTexture" on page 470.

Creating a Composite Texture

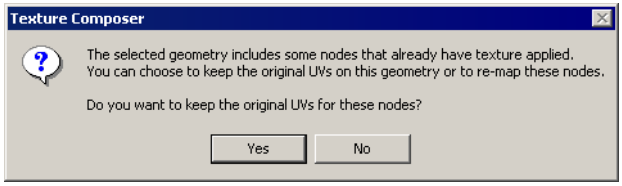
You can use the **Texture Composer** tool to generate a composite of texture mappings from a multi-textured model. Each texture must be a non-repeating texture. Using a single texture file optimizes your use of texture memory in Creator and in your runtime system.

To create a composite texture:

Step 1. Select all of the geometry in the Graphics view with the textures maps that you want to combine.



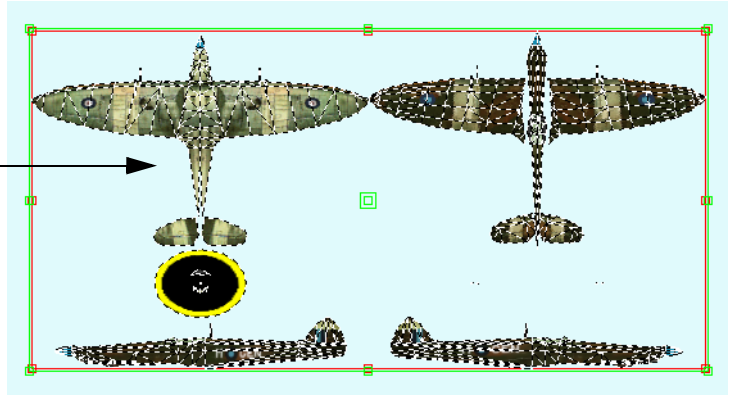
Step 2. Open the **Texture Composer** tool. The following message prompts you to keep the original *u,v* mappings, or to display the geometry without them.



Step 3. Click **No** to display the geometry without the original *u,v* mappings. Only one texture map will be loaded.

Step 4. Select **Draw Texture** and choose **Solid** mode so that the geometry displays with texture (this is the original texture referenced from the model).

The geometry displays with texture when **Draw Texture** and **Solid** are selected.



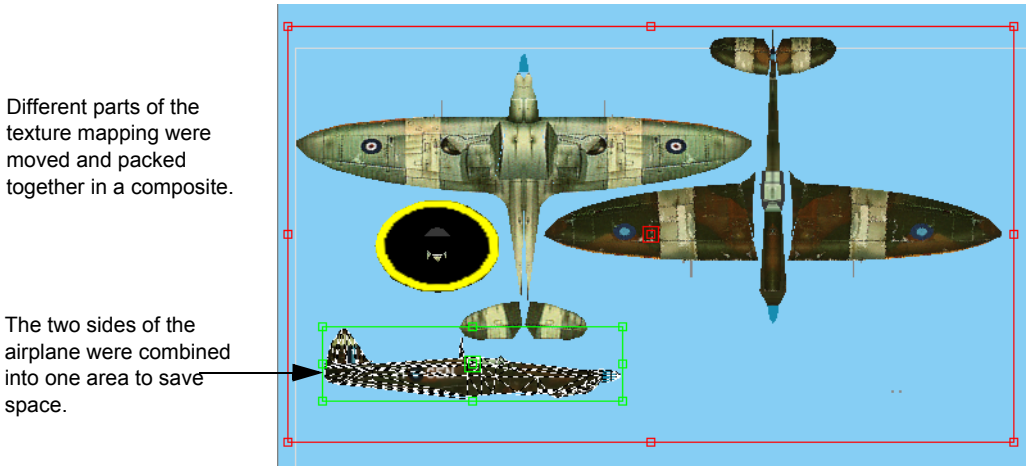
Step 5. Enter the **Width** and **Height** values for the final texture size. To optimize the use of texture memory, and maintain the highest resolution of the texture sources, the texture size dimensions should be less than or equal to the dimensions of your single largest texture on your model.

Note: If you change the size of the texture mapping, your geometry projection might be disproportionally scaled to the new texture size. Click **Map** with the Aspect mode set to **From Geometry** to correct the distortion.

Step 6. Drag select different *u,v*'s of the texture mapping with the left mouse to arrange them. These are some methods that you can use:

- Group *u,v*'s by size or type.
- Pack *u,v*'s tightly together to save space.
- Drag the control handles of the mapping area with the left mouse to enlarge it.
- Manipulate the *u,v*'s using the Position and Mapping parameters.
- Drag *u,v*'s outside of the *u,v* space temporarily while you arrange the *u,v*'s in the mapping area.

- If two parts of the geometry, such as sides of an airplane, are exactly the same but mirrored, you can select both sides, select the **One-Sided** projection method, the **From Geometry** aspect ratio, either the **Right** or **Left** plane for the mapping parameters, and then click **Map**. The two sides will be mapped on top of each other while still preserving the *u*,*v* coordinates.



Step 7. Click **Save As** to save the template as a new texture file. You can save the texture in either RGB, RGBA, INT, or INTA format. If you have added an alpha channel, be sure to save the texture in either RGBA or INTA file format.

Step 8. Click **Apply** to apply the new texture. The geometry is burned into the new texture.

You can choose a specific texture layer in the View panel to apply the texture to. For more information about using multiple texture layers on a model, see "MultiTexture" on page 470.

Note: If you open the Texture Palette, you will notice that the new composite texture is added, but may be blurrier than the original texture. The pixels were resampled to make the composite. The blurrier image may be suitable for your models.

Step 9. If you overwrote the original texture layer with the new composite texture, open the Texture Palette, and select **Remove Unused** in the **Edit** menu. The

original source texture will be removed, which is not necessary when the model is loaded.

Texture Composer Parameters

The *Texture Composer* window has the following parameters:

Projection

The general shape of the geometry when it is unwrapped into the 2D template. By default, the Texture Composer will use the **One Sided**, **Two Sided**, or **Box** projection method. You can choose one of the following:

- **One Sided** - Displays a customized one-sided planar view of the geometry. You can choose the Top (+XY plane), Bottom (-XY plane), Right (+YZ plane), Left (-YZ plane), Front (+XZ plane), or Back (-XZ plane) view. This is good for objects, such as terrain, that have one viewable side.
- **Two Sided** - Displays the opposite views along the X, Y, or Z axis. You specify the Front/Back, Left/Right, or Top/Bottom to create two one-sided projections into separate areas of the pattern. This is good for objects that have two viewable sides.
- **Box** - Projects the faces on a box, and then unwraps it showing either three or six sides. The pattern is produced by producing each polygon of the geometry into the "best" side of the box. This is good for objects that have six viewable sides.
- **Cylindrical** - Projects the faces on a cylinder, and then unwraps it.
- **Spherical** - Projects the faces on a sphere, and then unwraps it.
- **Unwrap** - Unwraps the geometry from a top-down view. The faces are displayed in their original shapes.
- **Face** - Displays an orthographic view of all of the faces in a random order.
- **Eyepoint** - Displays a customized one-sided planar view of the geometry. The eyepoint direction in the Graphics view is used, which you can change.

Method

The number of views, either Three Sided or Six Sided, that the **Box** projection displays.

Layout

The layout, either vertical or horizontal, of the projection.

Spacing

Creates a margin for minification or magnification of texels.

Aspect

Controls the aspect ratio of texels. Choose one of the following options:

- **From Geometry** - Keeps the original aspect ratio (proportions) of the selected geometry.
- **Fit To Target** - Scales selected geometry to fit the aspect ratio of the mapping area.

Target

The area that the mapping parameters (at the top of the window) affect. You can choose either **Selection Area** (green box) or **Mapping Area** (red box).

Map

Click to apply the mapping parameters to either the selection or mapping area.

Texture Width

The width of the output texture.

Texture Height

The height of the output texture.

Power of 2

Sets the texture dimensions to be a power of 2. If you enter a value that is not a power of 2, click in the other dimension field, or in the main window, to change it to the closest power of 2.

Alpha

Changes the background and all geometry to an alpha transparency. This may be useful to distinguish the geometry from the background color.

Background

Indicates that a background texture is displayed in the *Texture Composer* window. A background texture can be used to merge with geometry, or as a substitute for existing background texture.

Selected

Displays the original texture from the selected geometry as a background texture.

Wireframe

Draws only the wireframe outline of the geometry. The color of the wireframe is based on your solid color selection.

Solid

Draws a solid shaded model modified with either the texture map or the geometry color.

Wire/Solid

Draws the wireframe on top of the solid.

Solid Color

Select **Geometry** to display the original solid colors of the geometry, or select **Color** to choose a color from the Color palette for the original solid colors.

Fit

Maximizes the selection area to fill the *u,v* space.

Reset

Resets the selection area to its original size before **Fit**.

Fit To Texture Size

Fits the selected geometry to the selected texture size. If the texture size is too large to display at full resolution, increase the size of the *Texture Composer* window, or decrease the size of the texture. If you cannot increase the window size, then the texture will be scaled accordingly.

Merge with Background

Paints all geometry in the *u,v* space into the background texture. A message displays stating that this will overwrite the original background texture pattern.

Save As

Saves the template as a new texture that you specify.

Import UVs

Import the texture layers of the geometry into the Texture Composer.

Undo Last

Undos the last actions. The last undoable action is displayed on the button. The maximum number of actions that you can undo is unlimited.

Undo All

Undos all of the actions to the starting point, except for the **Save As** and the **Merge with Background** actions.

Apply

Applies the *u,v* mapping to the geometry in the Graphics view. You select the texture layer to apply it to.

Cancel

Cancels the Texture Composer operations and closes the window.

Texture Composer Tools

The *Texture Composer* window has the following tools:



Reset Mapping Area

If you change the mapping area, resets the default mapping area.



Pan Mode

Enables you to pan within the *u,v* space with the left mouse button, or Ctrl and the right mouse button.



Zoom Mode

Enables you to zoom within the *u,v* space with the left mouse button, or Shift and the right mouse button.



Pick Mode

Enables you to select items and create a new selection area, and adjust the control handles on the selection and mapping areas.



Scale Mode

Enables you to scale the geometry within the selection area when you drag the control handles with the left mouse. To scale proportionally, hold the Shift key down while dragging the control handles.



Rotate Mode

Enables you to rotate the geometry within the selection area when you drag a control handle with the left mouse. A diameter within the circle of rotation indicates the original starting location.



Remove Node

Deletes the selected geometry.



Bring to Front

If you have overlapping geometry in the *Texture Composer* window, all selected geometry is moved to the top draw layer. This is only active if z buffering is off.



Send to Back

If you have overlapping geometry in the *Texture Composer* window, all selected geometry is moved to the bottom draw layer.



Rotate 90

Rotates the selected geometry 90 degrees counterclockwise about the center point.



Mirror Horizontal

Flips the selected geometry horizontally.



Mirror Vertical

Flips the selected geometry vertically.



Browse

Enables you to browse for a texture file to load as a background texture.



Foreground/Background Color

Click the top color swatch to select a foreground color in the *Texture Composer* window. The foreground color affects the color of the selection area.

Click the bottom color swatch to select a background color. The pixels at the edge of the selected faces blend with the background color. The background color can also be saved with the geometry.



Draw Texture

Displays the original texture mappings on the geometry.



Draw Using Z-Buffer

Draws the geometry using z buffer sorting. If geometry overlaps other geometry, the geometry shown in the Graphics view is drawn according to the z-buffered display in Creator.

The Texture Mapping Palette

Texture application information can be stored in the Texture Mapping palette. The mapping information can be saved and reused. Using the Texture Mapping palette has two benefits:

- You can use a saved texture map to quickly texture faces without starting one of the texture application tools.
- You can restart a texture application tool and modify the mapping of an applied texture pattern. Any modifications made to a texture map in the Texture Mapping palette is automatically updated in the faces to which it was applied. The faces do not have to be selected in order for the changes to take place. If the texture mapping is not saved in the palette, you must restart the texture application tool from scratch.

One example of the usefulness of the Texture Mapping palette is when significant portions of a large-scale terrain must be covered with a specific texture. Rather than selecting groups of faces and applying the texture using Put Texture each time, you can apply the texture once, save the mapping in the Texture Mapping palette, and then continue to reapply the texture mapping from the Texture Mapping palette. If you adjust the texture mapping, all the faces will update with the change, so you don't have to modify the mapping for each group of faces.

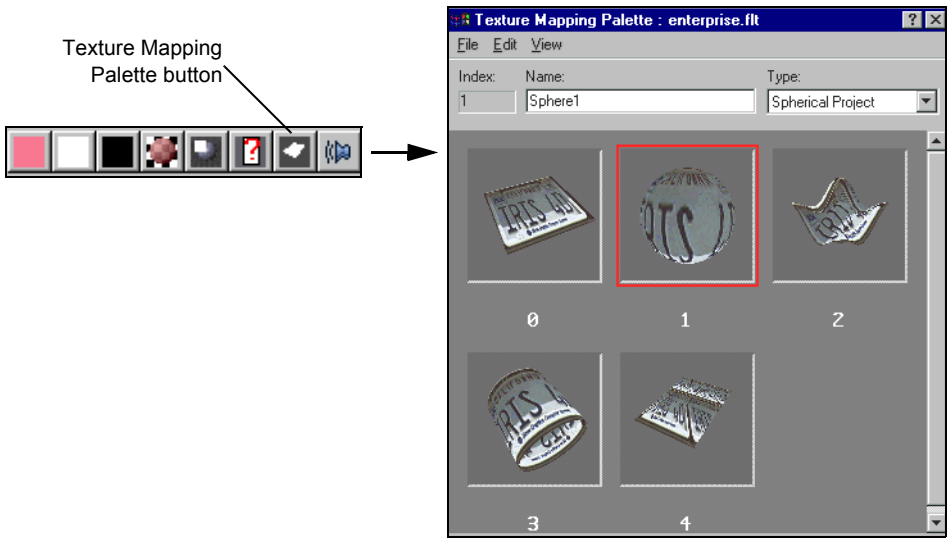
The realtime software needs only the *u,v* values stored in the vertices of textured faces to display an applied texture. However, in order to modify an applied texture mapping, Creator needs additional information, such as the control points used to apply the texture pattern. All of this information can be stored in a texture map in the Texture Mapping palette. Each texture map is assigned an index number, which is saved in the Texture Mapping field in each affected polygon's Face attributes.

You save texture mapping information to the Texture Mapping palette by setting the **Add to Mapping Palette** option when you apply the texture with any of the tools in the **Texture** toolbox (see "Texture Tools" on page 444).



To open the Texture Mapping palette, choose **Palettes/Texture Mapping** or click the **Texture Mapping Palette** button on the Toolbar.

Note: Because texture maps are not linked to specific texture patterns, each mapping is displayed using the current texture pattern.



The Texture Mapping palette displays the following information for each texture map.

- **Index -** Displays the index of the selected texture mapping in the Texture Mapping palette. Creator saves this number in the **Texture Mapping** attribute of each face to which the mapping is assigned.
- **Name -** Lets you assign a name of up to 31 characters to the selected texture mapping.
- **Type -** Indicates the type of the selected texture mapping. To change the type, choose a new option from this list and choose **Edit/Modify Mapping** in the Texture Mapping palette. Changes take effect when you close the texture mapping tool.

To Apply the Current Texture Mapping

Step 1. In Face or Object mode, select the faces to be textured.



Step 2. Open the **Properties** toolbox.

Step 3. Press Ctrl while clicking **Insert Texture**. The current texture will be applied to the faces using the current texture mapping.



To Change the Current Texture Mapping

The following steps gets the current texture mapping from a selected textured face.

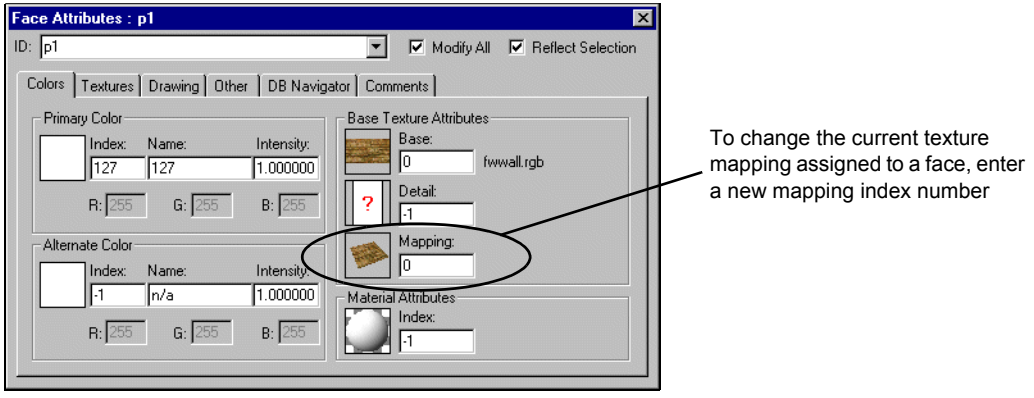
Step 1. In Face mode, select the textured face that you want to get the mapping from.

Step 2. Open the **Properties** toolbox.

Step 3. Press Alt while clicking **Get Texture From Face** to set the current texture mapping to that of the selected face.

Press Ctrl while clicking **Get Texture From Face** to set both the current texture pattern and the current texture mapping to the texture pattern and mapping of the selected face.

You can also apply or change a texture map for a selected face by opening the *Face Attributes* window and entering the texture map index number in the **Mapping** field in **Texture Attributes**.



To Create a New Texture Mapping

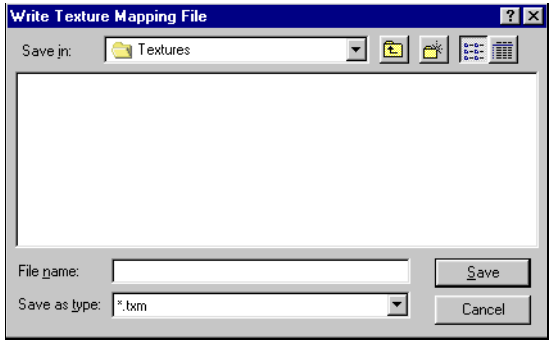
To create a new texture mapping, you can:

- Select a textured face and choose **Edit/New** in the Texture Mapping palette menu. A new texture mapping containing the same information as the texture mapping of the selected face will be added to the Texture Mapping palette.
- Select a textured face, and then hold down the Alt key and click the **Get Textures from Face** tool. A new texture mapping containing the same information as the texture mapping of the selected face will be added to the Texture Mapping palette.
- Click **Add to Mapping Palette** in any *Texture* tool window to add the current mapping to the Texture Mapping palette (see "Texture Tools" on page 444).

Once a new mapping has been applied, you can edit it as you want using the **Edit** menu functions **Modify Mapping**, **Scale**, **Translate**, **Rotate**, or **Eight Point Warp Texture**.

To Save a Texture Mapping Palette File

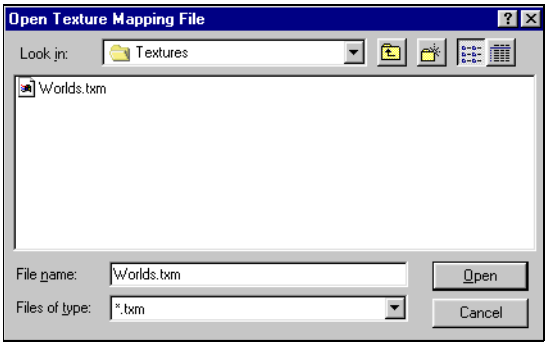
Step 1. In the Texture Mapping palette, choose **File/Save Palette**. The *Write Texture Mapping Palette* dialog box opens.



Step 2. Enter a name for the file and choose a directory, then click **Save**. The Texture Mapping Palette file will be appended with a `.txm` extension and saved to the specified directory.

To Load a Texture Mapping Palette File

Step 1. In the Texture Mapping palette, choose **File/Load Palette**. The *Open Texture Mapping File* dialog box opens.



Step 2. Select the file and then click **Open**. The Texture Mapping Palette file will be loaded into the Texture Mapping palette.

Texture Mapping Palette Menus

The Texture Mapping palette menus let you load and save palettes, edit a texture mapping, add a new mapping to the palette, and control the appearance of the Texture Mapping palette.

File Menu

The Texture Mapping palette **File** menu options save palette files to disk and load palette files.

Edit Menu

The options in the Texture Mapping palette **Edit** menu add new mappings to the palette, delete mappings, and edit existing mappings. You can cut or copy mappings from one Texture Mapping palette and paste them into another Texture Mapping palette.

The texture mapping modification tools, Modify, Translate, Scale, Rotate, and Eight Point Warp Texture, start the appropriate Modify Texture tools. Any modifications made to a texture mapping are automatically updated in all of the faces to which this mapping has been applied.

Modify Mapping

Starts the texture mapping tool associated with the selected texture pattern indicated **Type** option menu in the Texture Mapping palette. As long as the texture's **Type** has not changed, the tool restarts with the same parameter values in effect when the texture mapping was saved. If the **Type** has changed, the tool restarts with default parameter values.

Note: Choosing **Modify Mapping** is equivalent to clicking **Modify Mapping** in the **Modify Texture** toolbox.



View Menu

The **View** menu options in the Texture Mapping palette set the appearance of the Texture Mapping palette. The palette's default appearance is to show thumbnail images of each texture mapping in the palette. Clearing **Thumbnail** in the **View** menu displays larger versions of the mappings. Setting it again reduces the size of the mappings.

If there a large number of mappings, some may be hidden from view. **Fit on Select** brings the selected mapping into view.

Displaying Texture in the Database

These topics describe View settings and preferences that affect the way textures are displayed the Graphics view.

"Draw Texture" on page 521

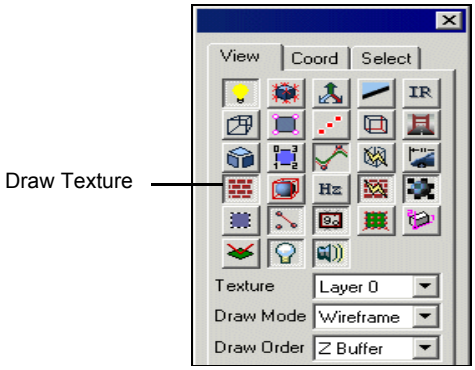
"Toggle Alpha Blending" on page 522

"Dynamic Texture" on page 523

Draw Texture

Because drawing textures increases the time required to draw the database window, you may want turn textures off. Applied textures can be displayed and hidden by clicking the **Draw Texture** button in the View panel or by pressing "T" on the keyboard.

If you want to have textures displayed by default, open the **Info/Preferences/State** panel and set the **Draw Texture** State preference.



Toggle Alpha Blending

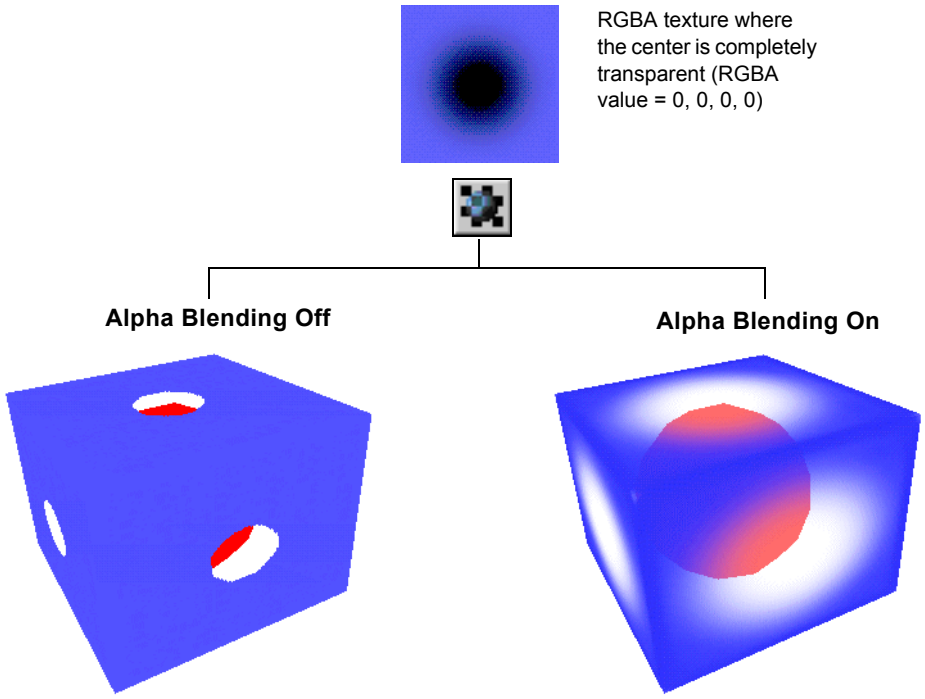


Toggle Alpha Blending, located in the View panel, toggles the ability to blend the transparency attribute (alpha channel) of RGBA or INTA texture images with faces behind the face to which the texture has been applied.

A texture's alpha component enhances a texture with transparency. An alpha value of 255 (white) is completely opaque. As the alpha value moves toward 0 (black), the RGB components become less visible (more transparent). A 0 alpha value indicates complete transparency.

> **Note:** Be sure the State preference **Alpha Blending** is set. If it is not set, alpha blending will not be visible when **Toggle Alpha Blending** is used.

In the figure below, an RGBA texture with a transparent center has been applied to the sides of a cube containing a sphere. The RGBA values in the texture gradually diminish toward 0 at the center. When **Toggle Alpha Blending** is off, only the very center of each side is transparent (where the RGBA value = 0, 0, 0, 0), producing a window-like effect, and displaying the sphere inside only at the transparent point. When **Toggle Alpha Blending** is on, the decreasing RGB values of the texture are modulated with the RGB values of the sphere. The sphere becomes increasingly more visible moving toward the center of the cube's side. The effect is a gradual fading of color and opacity.



Dynamic Texture



The **Dynamic Texture** control in the View panel causes applied textures to be displayed while the eyepoint is in motion. Normally texture is turned off while the eyepoint is moving in order to conserve drawing overhead. If the **Dynamic Texture** State preference is set, setting the **Dynamic Texture** control will turn it off.

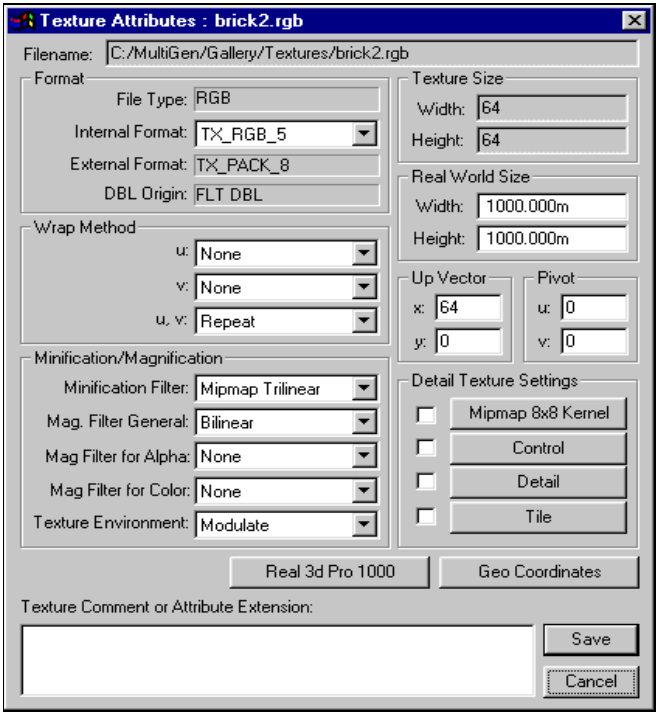
Texture Attributes

Creator uses two files to define each texture pattern: a *texture pattern file*, which contains the image (such as an `.rgb` or `.int` file), and a *texture attribute file*, which specifies how the file is loaded into memory and displayed.

The texture attribute file is automatically created and maintained by Creator. It is named after the original image file, with the extension `.attr`. For example, the image

file `wheel.rgb` is paired with the attribute file `wheel.attr`. This section describes the components of a texture attribute file.

To view the attributes of a texture pattern, open the Texture palette and double-click the texture pattern, or select the texture in the Texture palette and choose **Modify Attributes** from the Texture palette **Edit** menu.



Format Attributes

The **Format** area of the Texture Attributes window has the following settings:

Filename

Displays the directory path and file name of a texture pattern file.

File Type

For Intensity, Intensity-alpha, RGB and RGB-alpha, **File Type** indicates the actual format of the texture file. For all other supported image formats, **File Type** refers to the number of color channels the image file contains. The default file extension for supported formats are listed in "Texture" on page 421. The following types of texture pattern files are the most common:

- **Intensity** is an 8-bit grayscale (1-channel) format.
- **Intensity-alpha** is an 8-bit gray scale (2-channels) format that allows 8 bits of alpha.
- **RGB** is a 3-channel color image format with 8 bits each of red, green, and blue.
- **RGB-alpha** is a 4-channel color image format with 8 bits each of red, green, blue, and alpha.

Internal Format

Internal data formats dictate how the color components are packed into various byte configurations for performance, efficiency, or image quality when loaded into the realtime system's texture hardware. Internal format options include:

- **Default** automatically selects the internal format based on the texture type. For example, a 1-component intensity pattern is packed into an **Internal Format** of **TX_I_16**, an **INTA** is packed into a **TX_I_12A_4**, an **RGB** into a **TX_RGB_5**, and an **RGBA** into a **TX_RGBA_4**.
- **TX_I_12A_4** packs a 1- or 2-component texture into 16 bits, allowing 12 bits of intensity and 4 bits of alpha, producing better opaque image quality at the expense of transparent image quality.
- **TX_IA_8** packs a 2-component texture into 16 bits, allowing 8 bits of intensity and 8 bits of alpha, producing good opaque and transparent image quality.
- **TX_RGB_5** packs a 3-component texture into 16 bits, allowing 5 bits of red, 5 bits of blue, and 6 bits of green. This produces good drawing performance but moderately poor image quality.
- **TX_RGBA_4** packs a 4-component texture into 16 bits, allowing 4 bits per component. This produces good drawing performance but poor image quality.
- **TX_IA_12** packs a 2-component texture into 24 bits, allowing 12 bits per component. This may internally be rounded to 32 bits. This option produces good opaque and transparent image quality.

- **TX_RGBA_8** packs a 3- or 4-component texture into 32 bits, allowing 8 bits per component. This option produces good opaque and transparent image quality, at the expense of performance.
- **TX_RGBA_12** packs a 4-component texture into 64 bits, allowing 12 bits per component. This option produces very good image quality but poor drawing performance.
- **TX_I_16** packs a 1-component texture into 16 bits. This is normally used as a shadow map for realtime shadow computations. However, shadow maps are not supported by Creator.
- **TX_RGB_12** packs a 3-component texture into 64 bits, allowing 12 bits per component. This option produces very good image quality but poor drawing performance.

External Format

Determines the size of color components (intensity, or red, green, blue, and alpha) in the image. Creator uses **TX_PACK_8**, which assigns 8 bits to each component. This field cannot be changed.

DBL origin

Indicates the database format used by the software that created the texture pattern.

Wrap Method Attribute

Wrap methods **u** and **v** control the algorithm used to map textures onto polygons. Wrap method **u, v** specifies defaults for wrapping textures when either wrap method **u** or **v** is set to **None**. All three wrapping methods have the following options:

Clamp

Displays the texture pattern once. If the textured area is larger than the image, the outermost texels are repeated or "smeared" across the textured polygons.

Repeat

Displays copies of the texture pattern as many times as necessary to fill the textured polygons.

Size, Up Vector, and Pivot Attributes

Texture Size

Specifies the width and height of a texture pattern (in texels).

Real World Size

Specifies the width and height, in meters, used to map a texture pattern to polygons in the database. A sign, for example, might be one meter high by two meters wide.

Up Vector

Is included for backward compatibility with certain older realtime systems. For these systems, **Up Vector** defines an alignment vector (in terms of *x* and *y*) used to orient the texture pattern. A tree, for example, would have an up vector from the trunk toward the top of the tree.

Pivot

Designates a point (in terms of *u* and *v*) about which template polygons can be rotated in realtime. For example, if you have the TerrainPro option, and are using the TCT algorithm to generate terrain with cut-in features, you would use **Pivot** to align the side textures with the road surface on the bridges that are created automatically by TCT. (See the TCT lesson in the *Desktop Tutor* for details. The *Desktop Tutor* is included with Creator both in printed form and as an Acrobat file on the Creator CD.)

Minification/Magnification Attributes

Texture filters are algorithms used to soften, harden, or blend texels for more a pleasing visual effect when an object is resized or is reduced or enlarged by perspective as the eyepoint changes.

A *pixel* is the smallest unit of resolution available on the computer screen. When a texture pattern is rendered, the texels defining the pattern are mapped to screen pixels. As a result, the displayed texture is one of the following: *full size* (each texel maps to one pixel on the screen), *reduced* (several texels map to one screen pixel), or *magnified* (each texel maps to several screen pixels). The mapping of texels to pixels is controlled by the **Minification** and **Magnification** texture attributes.

Minification

Minification filters resolve situations in which multiple image texels are minimized to a point where they correspond to just one screen pixel, forcing the blending of texel colors. The **Minification Filter** setting in the *Texture Attributes* window identifies the algorithm that computes the color of each screen pixel when more than one texel maps to a single pixel on the screen.

If you select one of the **Mipmap** type minification filters, Creator internally generates a set of images that corresponds to the original texture pattern, as follows: the first image is produced by enlarging the texture pattern to dimensions equal to the nearest power of two; each consecutive image is produced by reducing the dimensions of the image to the next lower power of two. For example, if the texture pattern is 400 x 400 texels, the first mipmapped image is 512 x 512 texels, the second mipmapped image is 256 x 256 texels, and so on.

> **Note:** Mipmaps place an additional 1/3 load on memory at runtime to accommodate the generation of mipmap levels for each texture. Mipmaps are not saved to disk.

The following options are available for **Minification Filter**:

- **Point** computes the color of each screen pixel by selecting the texel nearest to the center of the pixel.
- **Bilinear** computes the color of each screen pixel from a weighted average of the four texels nearest to the center of the pixel.
- **Mipmap Point** computes the color of each screen pixel by first selecting the mipmapped image in which texels are closest in size to the texels of the applied texture, and then selecting (from that image) the texel nearest to the center of the pixel.
- **Mipmap Bilinear** selects the mipmapped image in which texels are closest in size to the texels of the applied texture, and computes the color of each screen pixel from a weighted average of the four texels in the selected image nearest to the center of the pixel.
- **Mipmap Trilinear** selects the two mipmapped images in which texels are closest in size to the texels of the applied texture. In each selected image, the four texels nearest to the center of each pixel are averaged. This creates two color maps for each screen pixel. **Mipmap Trilinear** computes the final color of each screen pixel by interpolating the corresponding pixels in both color maps.

- **Bicubic** computes the color of each screen pixel using a smooth (spline) weighting of the 4x4 area of texels nearest to the center of the pixel.
- **Bilinear greater/equal, Bilinear less/equal, Bicubic greater/equal**, and **Bicubic less/equal** can be used in realtime shadow computation.

  Note: With these four filters, you must choose the same setting for **Minification** and **Magnification** filters.

- **None** computes pixel color using the system default (generally, **Bilinear**).

Magnification Filter: General, Alpha, Color

Magnification filters resolve situations in which multiple screen pixels correspond to one image texel. This is likely to occur when the image is zoomed close to the viewer, magnifying the size of the mapped texels. The chosen magnification filter selects the algorithm used to compute the color of each screen pixel when one texel maps to more than one pixel on the screen.

These options are available for **Magnification Filter**:

- **Point** computes the color of each screen pixel by selecting the texel nearest to the center of the pixel.
- **Bilinear** computes the color of each screen pixel from a weighted average of the four texels nearest to the center of the pixel.
- **Bicubic** computes the color of each screen pixel from a smooth (spline) weighting of the eight texels surrounding the center of the pixel.
- **Sharpen**, for each of the two highest mipmap levels, computes the color of each pixel from a weighted average of the color of the four texels nearest to the center of the pixel, then extrapolates the final color of each screen pixel from the value of the corresponding pixels in both maps.
- **Add Detail** replaces this texture with the (larger) detail texture specified in the face attributes of each textured polygon (see "Detail Texture" on page 530. Detail texture is not available on all hardware platforms.
- **Modulate Detail** blends this texture with the detail texture specified in the face attributes of each textured polygon. Detail texture is not available on all hardware platforms.
- **Bilinear greater/equal, Bilinear less/equal, Bicubic greater/equal**, and **Bicubic less/equal** can be used in realtime shadow computation.

Note: With these four filters, you must choose the same setting for **Minification** and **Magnification** filters.

- **None** computes pixel color using the system default (generally, **Bilinear**).

Texture Environment

Controls the algorithm used to blend texture color with polygon color or other textures (see "MultiTexture" on page 470). The texture environment defines how a texture interacts with the color of the face or object to which it is applied and can be any one of the following:

- **Modulate** integrates texture values with polygon color. In this mode, face color affects the colors of certain kinds of texture patterns applied in the database.
- **Blend** modulates the primary and alternate color of the polygon based on the given texture element intensity value. For example, a texel with a grayscale value of 26 (10% of the range 0 to 255) has 10% of the alternate color blended with 90% of the primary color.
- **Decal** replaces the polygon color with the texture color. An RGBA texture blends with polygon colors in a ratio determined by the texture's alpha component. The polygon's alpha component is unchanged. Decal is only applicable to RGB and RGBA texture patterns.
- **Replace** replaces the polygon color with the texture color. An RGBA texture replaces polygon colors.

  Note: To prevent face color from affecting the color of the texture pattern, set the **Draw New Textured Polygons White** Header attribute *before* creating the face, or set the **Render Textured Polygons White** Face attribute *after* creating the face. The face's primary color will change to white. This applies to all texture environments except for **Replace.**

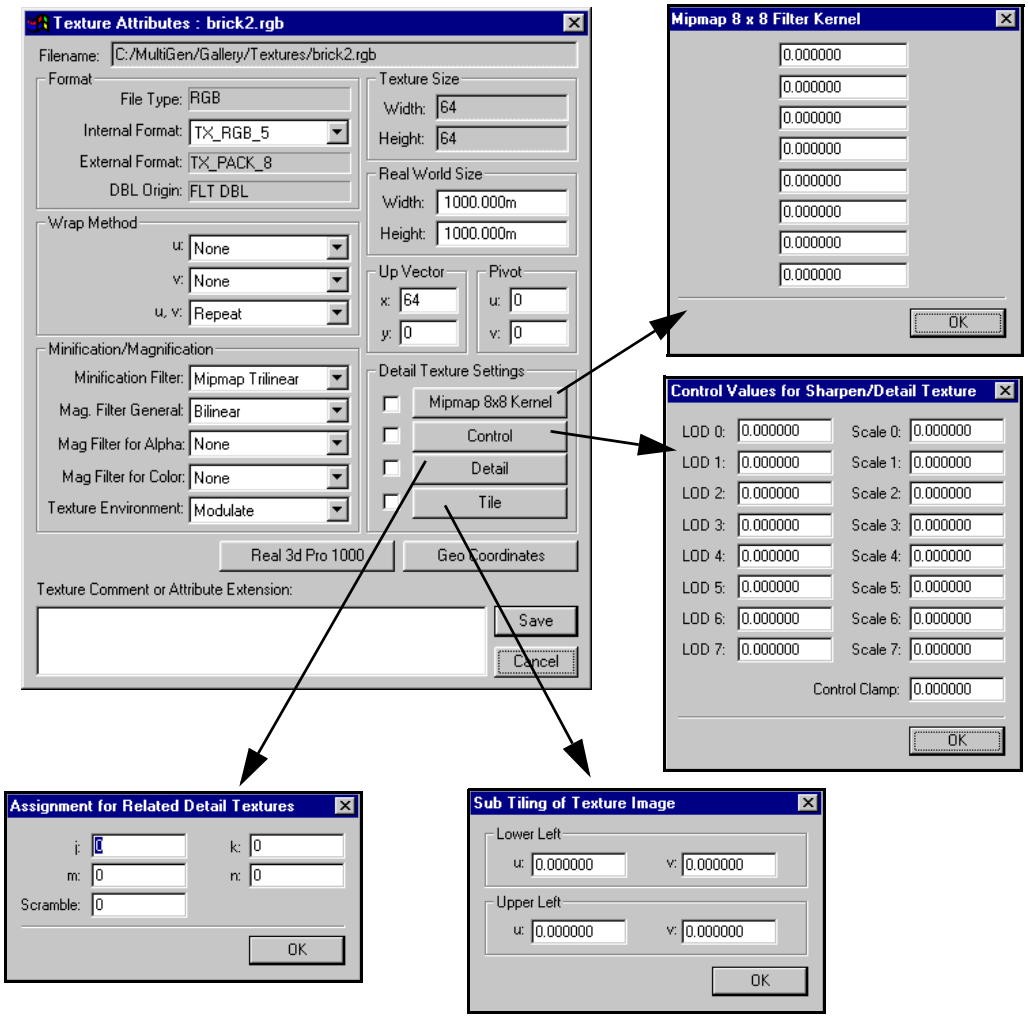
Detail Texture

Detail textures let you create successively more detailed texture images from a single image. These textures, when applied to a database with several levels of detail, give the appearance of greater and greater detail as the eyepoint moves through higher levels of detail. At a low level of detail, a low resolution mipmap of the texture image is applied. As the levels of detail increase, Creator applies higher and higher resolution mipmaps. Using detail textures instead of creating actual models provides a

tremendous polygon savings when flying over or driving through a database where things must look realistic to the eye but don't require touching or interacting.

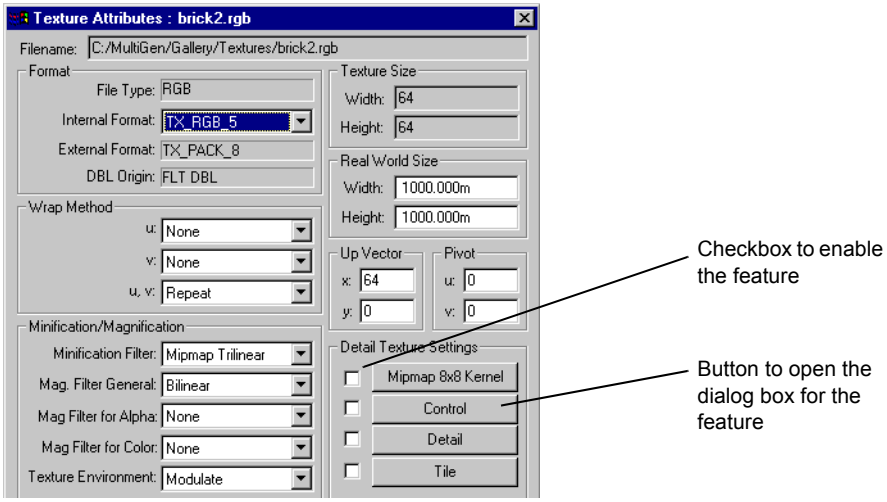
To create detail textures, load a texture image into the Texture palette and choose **Edit/ Texture Attributes**. The settings in the *Detail Texture* dialog boxes were developed from IRISGL specifications. OpenGL image generators can translate most of the IRISGL settings (exceptions are noted below); Creator, which is an OpenGL application, cannot. This means that, although you will not see the effects in Creator, you will see them in realtime.

The *Texture Attributes* dialog box in Creator has four dialog boxes for defining Detail Texture Settings: **Mipmap 8x8 Kernal**, **Control**, **Detail**, and **Tile**. The following sections describe each dialog box and how to use the settings.



To set up and enable a detail texture:

Step 1. In the *Texture Attribute* window, click the button for the feature.



Step 2. Enter suitable values, and then click **OK**.

Step 3. To enable the settings you just entered, check the box next to the feature. If the box is not checked, the settings are saved, but are not enabled.

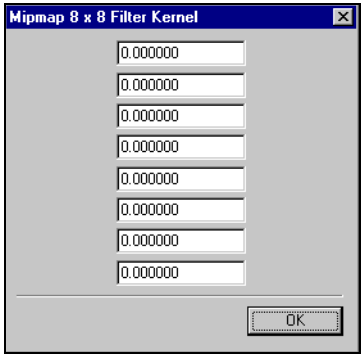
Mipmap 8 x 8 Filter Kernal Setting

The **Mipmap 8x8 Filter Kernal** values define an 8x8 separable symmetric convolution kernel used to generate mipmaps for a texture. The values in this dialog box create both a vertical matrix and a horizontal matrix. These matrices are then multiplied to form the convolution kernal that is used to calculate the pixel colors for mipmaps.

For example, if the numbers were 1, 2, 3, 4, 5, 6, 7, 8, the calculation for the matrix would be:

$$\begin{bmatrix} 1 & 2 \\ 3 & 4 \\ 5 & 6 \\ 7 & 8 \end{bmatrix} \times \begin{bmatrix} 1 & 2 & 3 & 4 & 5 & 6 & 7 & 8 \end{bmatrix}$$

The convolution kernal gives you some fine control over the appearance of the hardware-generated mipmaps. The IRISGL default values are 0.0, 0.0, 0.125, 0.375, 0.375, 0.125, 0.0, 0.0. Creator initially sets all values to 0.000000.



Note: In OpenGL, the realtime application creates the mipmaps, and no fine control is available. The *Mipmap 8x8 Filter Kernal* dialog box is provided for backward compatibility only.

Control Values for Sharpen/Detail Texture

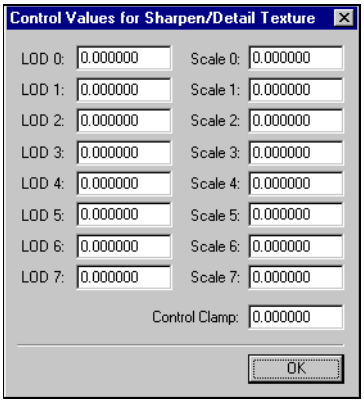
In IRISGL, when a texture is rendered in the frame buffer, an LOD value is computed as follows:

```
LOD = log2 (rendered_texel_scale / texel_scale)
```

For example, if the full resolution texture is rendered one-to-one, LOD = 0.0. If the rendered texture is minified, LOD is negative, and if the rendered texture is magnified, LOD is positive.

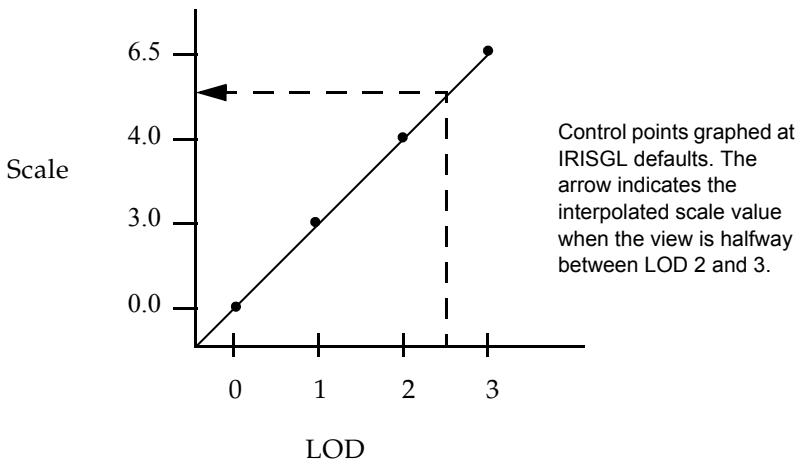
For images that are minified, the LOD is used to select the correct mipmap level. For images that are magnified with the filter set to **Sharpen**, **Add Detail**, or **Modulate Detail**, the fields in the *Control Values for Sharpen/Detail Texture* dialog box allow users to enter control points for each LOD that give them greater control over the way the hardware sharpens, adds, or blends the detail texture during rendering.

In this dialog box, you enter values for **LOD** and **Scale**. The corresponding values (LOD 0 and Scale 0, for example) are the coordinates of a control point. The default values in IRISGL are ((0.0, 0.0), (3.0, 3.0), (4.0, 4.0), (6.5, 6.5)). As you can see in the figure below, the defaults are 0.0 for all points. The IRISGL function call for this feature was written to accept up to 8 values, but only 4 were ever implemented in Performer, so the values for LOD 4 - LOD 7 are meaningless.



The **Control Clamp** specifies a maximum scale value. The IRISGL default value is 6.5.

The IRISGL Scale function computes the scale value by linearly interpolating between the control points (LOD, Scale pairs).



The way the interpolated scale value, Scale (LOD), is used depends on the **Magnification Filter** setting in the *Texture Attributes* window:

- If the **Magnification Filter** is set to **Sharpen**:

Rendered_texture = (1 + `Scale ( LOD )`) *
bilinearly_interpolated_texture_Mipmap_level_0 –

(`Scale ( LOD )`) * bilinearly_interpolated_texture_Mipmap_level_1

This uses the difference between the full resolution texture and the half resolution texture to sharpen the magnified full resolution texture.

- If the **Magnification Filter** is set to **Add Detail**:

  Rendered_texture = bilinearly_interpolated_texture_Mipmap_level_0 +
  (Bilinearly_interpolated_detail_texture – max_pixel_value * 0.5) * `Scale (LOD)`

  This gradually adds the detail texture as the rendered texture gets bigger.

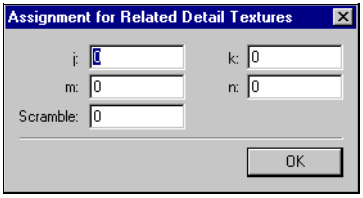
- If the **Magnification Filter** is set to **Modulate Detail**:

  Rendered_texture = bilinearly_interpolated_texture_Mipmap_level_0 + (
  Bilinearly_interpolated_detail_texture – max_pixel_value * 0.5) *
  Bilinearly_interpolated_texture_Mipmap_level_0 * `Scale ( LOD )`

  This gradually blends in the detail texture as the rendered texture gets bigger.

Assignment for Related Detail Textures

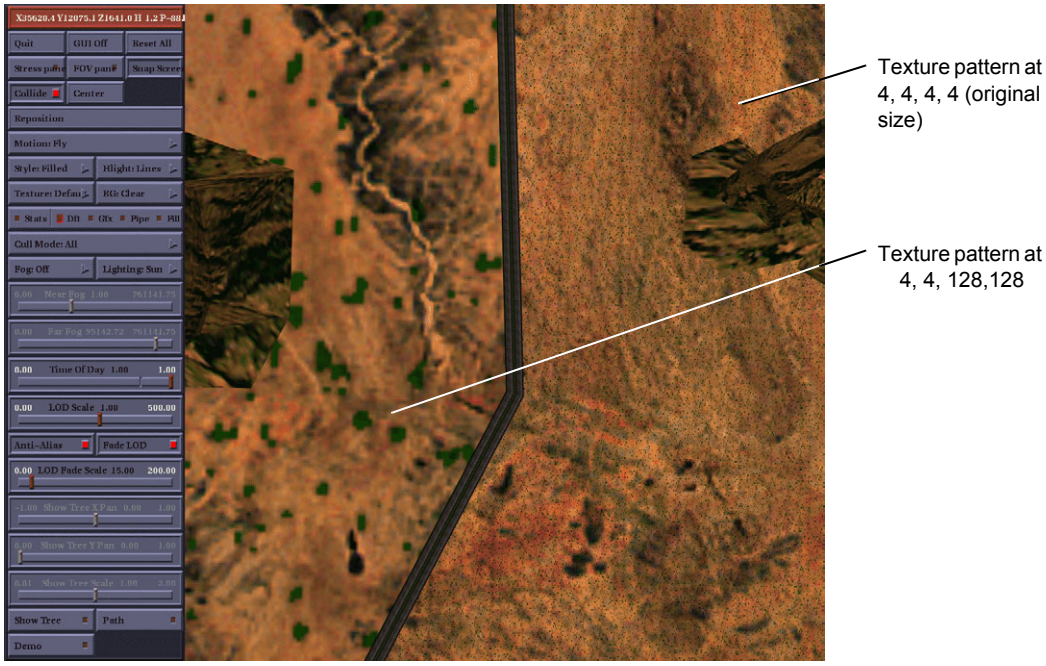
The settings in this dialog box define the relationship between the size of the detail texture and the size of the main texture.



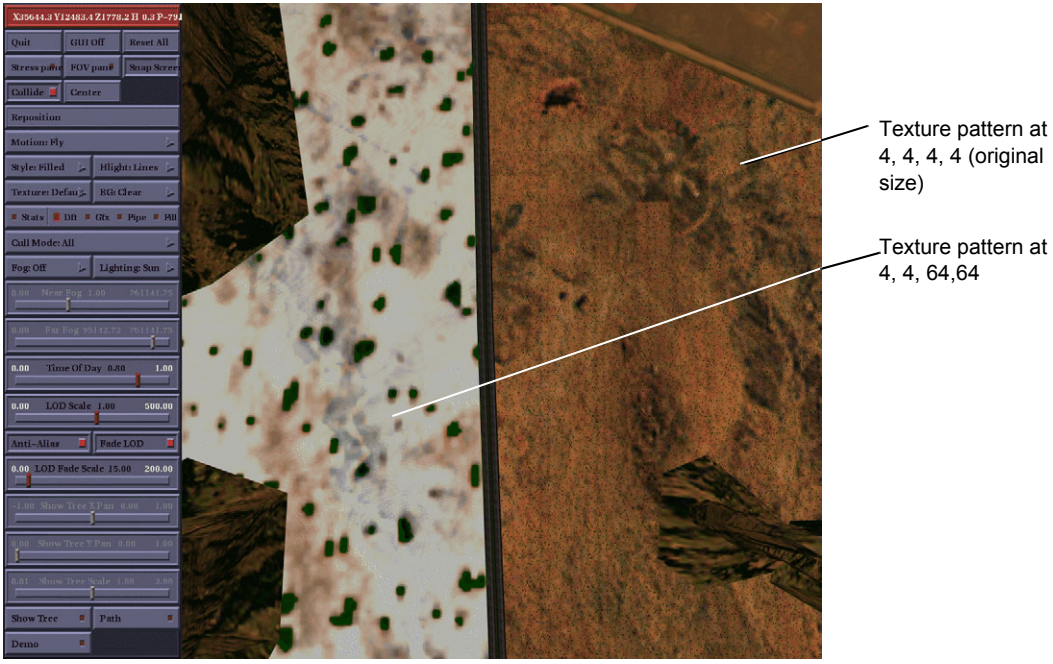
A *j* x *k* block of detail texture modulates or adds to an *m* x *n* block of main textures (whether the detail texture modulates or adds depends on whether **Mag. Filter General** is set to **Modulate Detail** or **Add Detail**).

The values *j* and *k* must always equal 4; anything else you might enter for these parameters is ignored. For *m* and *n*, valid values are 4, 8, 16, 32, and 64. As you increase the number, the pattern's texels are replicated to make the texture pattern bigger.

If you enter a value between 4, 8, 16, 32, and 64, IRISGL takes the next highest number. Numbers greater than 64 cause the pattern to wrap. For example, at 4, 4, 128, 128, the pattern looked like the figure below when **Mag. Filter General** is set to **Modulate Detail**:



When **Mag. Filter General** is set to **Add Detail**, the RGBA values in the texture pattern are added; the result is a lighter image. The following figure shows how the texture in the previous figure looks when you change the filter setting to **Add Detail**.

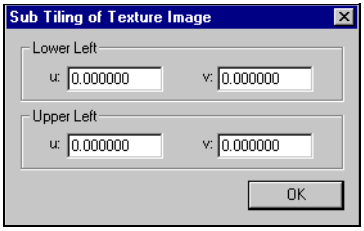


The **Scramble** parameter is not used. It is included only for backward compatibility.

Sub-Tiling of Texture Image

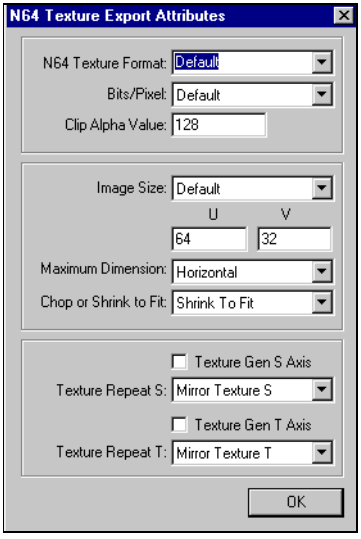
You use the settings in this dialog box to specify a region in the texture pattern for display; the rest of the texture will be displayed in 100% alpha, rendering it invisible.

To specify the area you want to display, enter the coordinates for the **Lower Left** and **Upper Right** corners of the area.



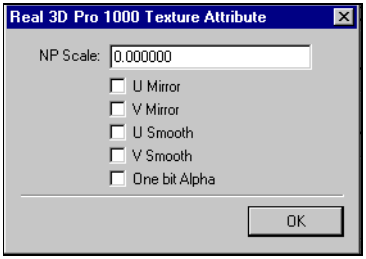
Nintendo 64 Texture Attributes

Opens the *N64 Texture Export Attributes* dialog box.



Real 3d Pro 1000 Texture Attributes

Opens the *Real 3D Pro 1000 Attributes* dialog box. See the Real 3D documentation for an explanation of the attributes.



Texture Variables and Preferences

The following topics explain texture-related environment variables and preferences.

The MPI_CREATOR_TXTPATH Environment Variable

It is good practice to organize all the textures for a database in a single directory. To make it easy to find and open these textures, you can set the environment variable `MPI_CREATOR_TXTPATH` to the directory where you are storing the textures for your current project. `MPI_CREATOR_TXTPATH` specifies the default path for loading and saving texture patterns; when you want to load a new texture into the Texture palette, for example, this is the directory that is first displayed in the *Read Pattern* dialog box.

When you open a database, Creator searches for the file `default.txt`, which contains the default Texture palette. If it does not find this file in the *working directory*, Creator looks for it in the directory specified by the MPI_CREATOR_TXTPATH environment variable. The working directory for Windows is the directory where Creator is installed, and for IRIX it is the directory where Creator was launched.

For instructions on setting environment variables on your platform, see “Environment Variables” on page 43.

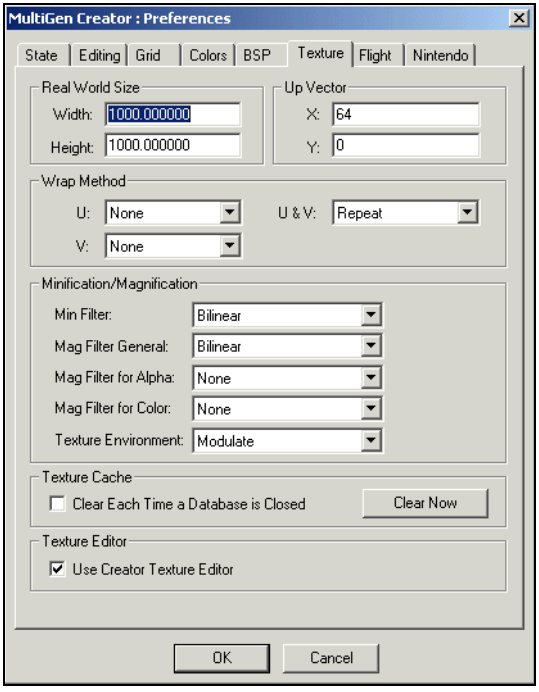
Texture Preferences

In the *Preferences* dialog box, the Texture panel contains options that control the default attributes assigned to newly-loaded textures, State panel preferences control the default texture display when you open a new database, and an Edit panel setting

determines if the current texture and texture mapping will be applied automatically when you create new faces. The following sections describe these preferences in detail.

Texture Preferences Panel

To open the *Preferences* dialog box, choose **Info/Preferences** (Ctrl+Shift+P). Click the **Texture** tab to view the attributes that Creator assigns to newly-loaded textures.



For an explanation of the **Real World Size**, **Up Vector**, **Wrap Method** and **Minification/ Magnification** settings in this panel, see "Texture Attributes" on page 523.

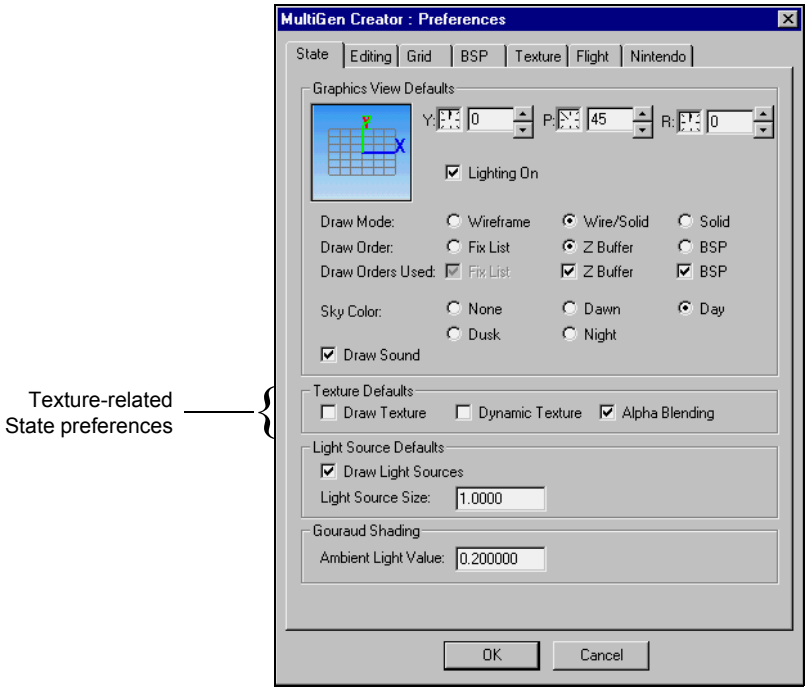
When **Clear Each Time a Database is Closed** is set, Creator automatically removes any unused texture patterns from the internal texture cache when a database is closed. If this preference is not set, you must purge the palettes manually by clicking the **Clear Now** button.

Texture patterns that are loaded into Texture palettes remain in memory when you close the current database and open another database. After opening and closing several databases and loading new textures, textures that build up in the internal texture cache can affect performance.

State Preferences Panel

The **State** panel in the Preferences window has **Texture Defaults** settings that affect texture-related behavior in newly-created databases.

State Preferences



- When **Dynamic Texture** is checked, texture is displayed when the eyepoint is in motion. If you find that the eyepoint does not move smoothly when **Draw Texture** is set, try turning off **Dynamic Texture**. This reduces drawing overhead by automatically switching texture off when the eyepoint is in motion and switching it back on when the eyepoint comes to rest.

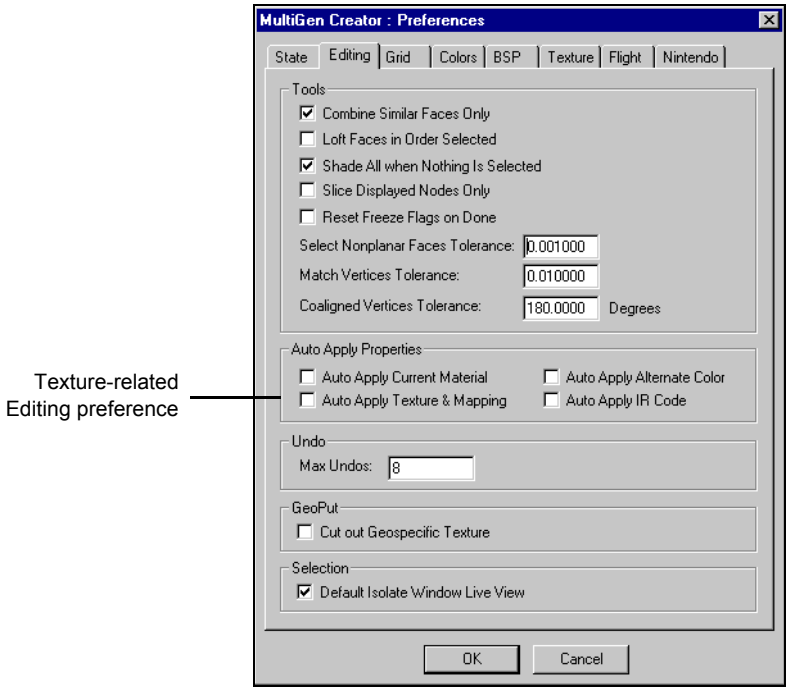
Note: Setting the **Dynamic Texture** State preference is equivalent to setting **Dynamic Texture** in the View controls.

- When **Draw Texture** is checked, texture is displayed when a Graphics view is first opened.
- When **Alpha Blending** is checked, textures that have an alpha component (indicating transparency) are blended with other faces in the database. Transparent areas are blended with the face behind the textured face in the Graphics view.

Editing Preferences Panel

The **Editing** preferences panel lets you set automatic application of the current texture when creating new faces.

Editing Preferences



Auto Apply Texture & Mapping

If the **Auto Apply Texture & Mapping** Editing preference is set, the current texture pattern and texture mapping are automatically assigned to a new face when it is created. If this preference is off, you have to assign textures manually.

Editing Texture Patterns

You can edit your textures in an image editor before applying them to geometry in a Creator database. Image editors include MultiGen Creator's Texture Editor or external image editors such as Adobe Photoshop or GIMP (GNU Image Manipulation Program). After you set an image editor for Creator, the image editor will open when you choose **Edit/Edit Pattern** from the Texture palette (or double-click the **Current Texture** button on the Toolbar).

Installing Adobe Photoshop Plug-ins (Windows Only)

Creator includes two plug-ins for Adobe Photoshop, `RGBFormat.8BI` and `PBMFormat.8BI`, which enable Photoshop to read, edit, and save files in SGI RGB and Portable Bitmap formats, respectively. If Adobe Photoshop is available on your PC *at the time you install Creator*, these files will be installed in the Photoshop Plug-ins directory, and Photoshop will automatically be associated with these file types. If you install Photoshop after you install Creator, drag these plug-ins from the Creator installation CD to the Photoshop Plug-ins directory, and manually associate the file types with Photoshop using the procedure in the following section.

Setting an Image Editor (Windows)

The Texture Editor is set in the *Preferences* window to open all texture file types that Creator supports. To set an external image editor instead, you can clear the checkbox and specify a program to open the particular texture file types.

To set an external image editor:

Step 1. Choose **Info/Preferences** in the Creator database window to open the *Preferences* window. Click the **Texture** tab.

Step 2. Clear the **Use Creator Texture Editor** checkbox in the **Texture** panel.

Step 3. *Windows 2000/XP Professional*: In **My Computer** or **Windows NT Explorer**, choose **Tools/Folder Options**.
Windows NT: In **My Computer** or **Windows NT Explorer**, choose **View/Options**.

Step 4. Click the **File Types** tab.

Step 5. Click the file type you want to change in the **Registered file types** list.

Windows 2000/XP Professional: The settings for that file type appear in the **Details for extension** group.
Windows NT: The settings for that file type appear in the **File type details** group.

Step 6. *Windows 2000/XP Professional*: Click **Advanced**.
Windows NT: Click **Edit...**

Step 7. In the *Edit File Type* dialog box, click **Open** in the **Actions** box.

Step 8. Click **Edit...**

Step 9. In the *Editing action for type* dialog box, specify the program that will open files with this filename extension in the **Application used to perform action** field.

Note: If there are no existing associations with an application, **Open** will not be available; choose **New** instead. Then type `Open` in the **Action** field and specify the program that will open files with this filename extension.

Setting an Image Editor (IRIX)

The Texture Editor is set in the *Preferences* window to open all texture file types that Creator supports. To set an external image editor instead, you can clear the checkbox and specify a program, such as GIMP, to open the particular texture file types.

To set an external image editor:

Step 1. Choose **Info/Preferences** to display Creator preferences. Click the **Texture** tab.

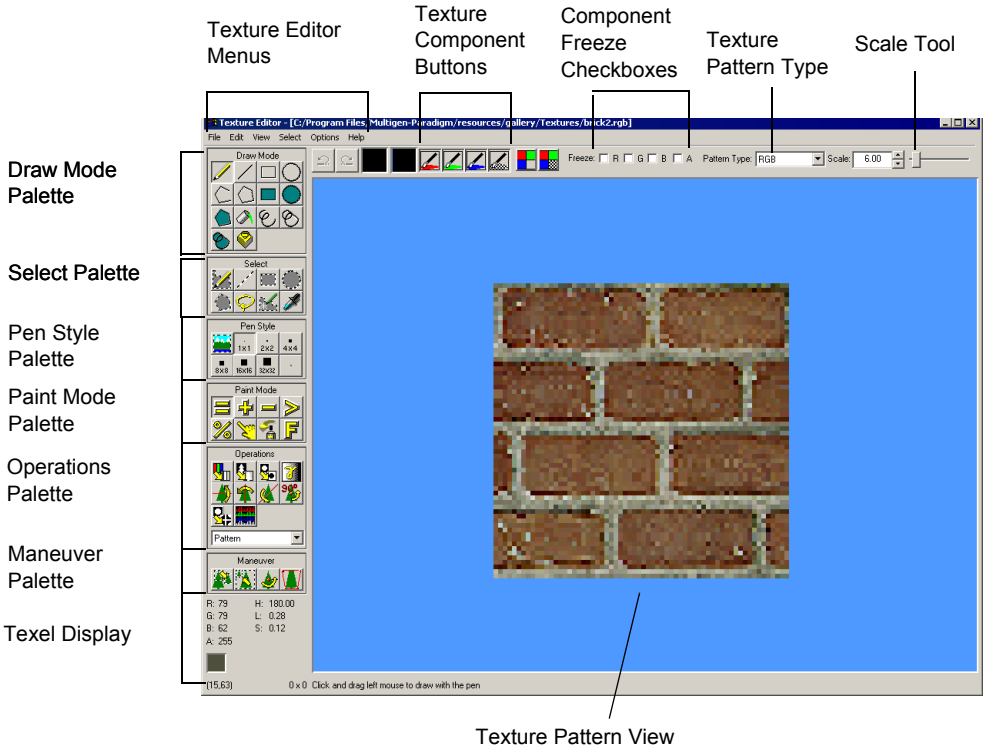
Step 2. Clear the **Use Creator Texture Editor** checkbox in the **Texture** panel.

Step 3. Enter the path to the image editor's executable file, including option switches, in the **Texture Editor** field. For example, `/usr/imgworks -p`.

Step 4. Click **OK** to update the preferences.

The Texture Editor

The Texture Editor is the MultiGen Creator image editor that lets you edit a texture pattern that has been loaded into the Texture palette. You can view an applied texture in the database window as you edit and make changes to the texture. The Texture Editor includes image editing tools to modify, resize, or repeat texture patterns.



Related Topics

"Starting the Texture Editor" on page 549

"Reading the Texel Display" on page 550

"Controlling the Eyepoint" on page 550

"Supported Image Formats" on page 551

"Texture Editor Menus" on page 552

"Using the Draw Tools" on page 558

"Selecting Texels" on page 563

"Defining the Pen Tip" on page 567

"Choosing a Paint Mode" on page 570

"Displaying and Protecting Texels" on page 577

"Using Texture Operations Tools" on page 578

Starting the Texture Editor

To edit a texture in the Texture Editor, you first load a texture pattern in the Texture palette, and then choose **Edit Pattern** in the Texture palette **Edit** menu. You can also double-click the **Current Texture** button on the Toolbar to open the texture in the Texture Editor.

You can only edit one texture at a time, which is the current texture, in the *Texture Editor* window. You can, however, copy elements from one texture to a different texture in a new *Texture Editor* window.

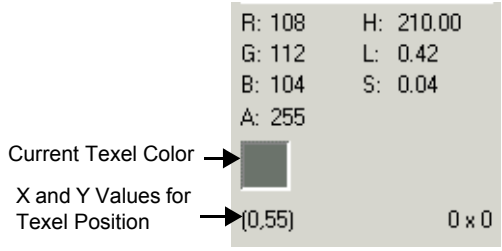
To start the Texture Editor:

Step 1. Load a texture into the Texture palette. See "Texture Palettes" on page 15-422 for information about loading and saving textures in the Texture palette.

Step 2. Select the texture and choose **Edit Pattern** in the Texture palette **Edit** menu, or double-click the **Current Texture** button on the Toolbar in the database window. The texture opens in the *Texture Editor* window.

Reading the Texel Display

The Texel display at the bottom of the *Texture Editor* window provides information about the texel under the pointer. As you move the mouse, the information about that texel changes dynamically.



The Texel display shows the color of the current texel. **R**, **G**, **B**, **A** values indicate the red, green, blue, and alpha components of the texel color; **H**, **L**, and **S** are the equivalent hue, lightness, and saturation values. **X** and **Y** values on the bottom of the Texel display give the texel position relative to the lower-left corner of the texture pattern.

Controlling the Eyepoint

You can change the eyepoint, such as zooming in, zooming out, or panning the eyepoint (moving the view horizontally or vertically), by doing the following:

- To zoom in or zoom out, press Shift+right mouse while dragging the mouse up and down, or slide the **Scale** tool slider at the top-right of the *Texture Editor* window.
- To pan the eyepoint, press the right mouse button while dragging the mouse in any direction.
- To reset the *Texture Editor* window to the default view, choose **Reset View** in the **View** menu.

Supported Image Formats

Any texture pattern that can be read into the Creator Texture palette can be edited. These include image file formats supported natively by Creator as well as image file formats supported by currently loaded image importer plug-ins. The Texture Editor can only save files to SGI file formats.

The image file formats supported natively by Creator include:

- SGI formats RGB, RGBA, INT, INTA
- TIFF
- GIF
- JPEG/JFIF
- PCX
- IFF/ILBM
- Alias Pix
- BMP/DIB
- PNG
- PPM
- Sun Raster
- Clip Texture Version 1

The image file formats supported by image importer plug-ins developed and distributed by MultiGen-Paradigm include:

- TGA
- Clip Texture Version 2

The Texture Editor saves textures only in SGI formats. For example, if you edit a JPG file, you must save it in one of the following formats:

- RGB
- RGBA
- INT
- INTA

Texture Editor Menus

The Texture Editor menus include functions to save, edit, and manage texture patterns. Click a menu option, such as **File** or **Edit**, on the Texture Editor Toolbar to see its menu.

Related Topics

"File Menu" on page 553

"Edit Menu" on page 554

"View Menu" on page 555

"Select Menu" on page 556

"Options Menu" on page 557

File Menu

The **File** menu includes the following functions.

"Save" on page 553

"Save As" on page 553

"Resize" on page 553

"Revert" on page 554

"Close" on page 554

Save

Choose **Save** to save the texture pattern in the *Texture Editor* window.

Save As

Choose **Save As** to save the texture pattern in a different file.

To save the texture pattern in a different file:

Step 1. Choose **Save As** in the **File** menu. A standard file chooser appears in which you specify the output file and file type.

Step 2. Click the **Save** button.

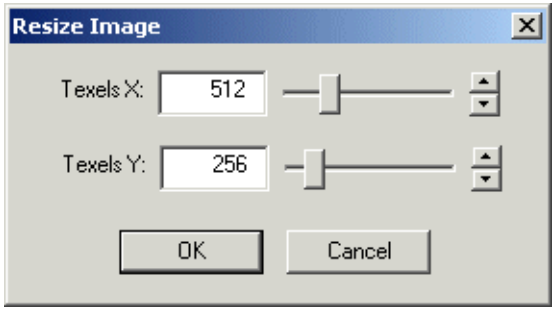
Note: You must add the texture pattern that you saved to the database Texture palette before you can apply the texture pattern to the database.

Resize

You can change the dimensions (width and height) of the texture pattern, which is measured in texels.

To resize the texture pattern:

Step 1. Choose **Resize** in the **File** menu to display the *Resize Image* dialog box.



Step 2. Enter the number of texels (between 0 and 2048) for the new width in the **Texels X** field, or slide the slider bar.

Step 3. Enter the number of texels (between 0 and 2048) for the new length in the **Texels Y** field, or slide the slider bar.

Step 4. Click **OK**.

Revert

Choose **Revert** to remove all edits and reload the original version of the texture pattern in the *Texture Editor* window.

Close

Choose **Close** to close the *Texture Editor* window without saving any changes.

Edit Menu

The **Edit** menu includes the following functions.

"Undo" on page 555

"Redo" on page 555

"Modify Alpha" on page 555

Undo

Choose **Undo** to remove any number of changes to the texture pattern that have not been saved.



You can alternatively click the **Undo** button at the top of the *Texture Editor* window. When there are no more changes to undo, this button will become gray.

Redo

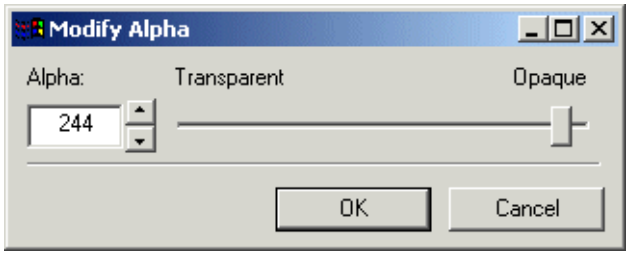
Choose **Redo** to replace any number of changes to the texture pattern that you removed with **Undo**.



You can alternatively click the **Redo** button at the top of the *Texture Editor* window. When there are no more changes to redo, this button will become unavailable.

Modify Alpha

Choose **Modify Alpha** to modify the alpha component value for transparency and opaqueness. You can enter an alpha value or use the slider bar to enter a value from 0-255 in the *Modify Alpha* dialog box. A value of 0 indicates complete transparency, and a value of 255 indicates complete opaqueness.



View Menu

The **View** menu includes the following functions.

"Fit on Select" on page 556

"Reset View" on page 556

"Pen Window" on page 556

"Render" on page 556

"Tile" on page 556

Fit on Select

Choose **Fit on Select** to center the eyepoint over selected texels in the texture pattern, and zoom the eyepoint in or out to make the selection fill the view.

Reset View

Choose **Reset View** to reset the *Texture Editor* window to the default view.

Pen Window

Choose **Pen Window** to view the current pen tip. You can see the pen tip color, shape, and texels. If you leave this window open while you change the pen tip, you can see the pen tip change dynamically.

Render

Choose **Render** to render the texture using hardware or software, whichever is faster for your computer. You choose either **Software** or **Hardware** in the **Render** menu.

Tile

Choose **Tile** to display one copy, or three or nine repetitions of the texture pattern. Texture patterns often repeat across surfaces in a database. This function lets you see how the pattern looks if it is repeated. You choose either **1 x 1**, **3 x 3**, or **9 x 9** in the **Tile** menu. Zoom out in the *Texture Editor* window if the tiled pattern does not fit in view.

Select Menu

The **Select** menu includes the following functions.

"Select All" on page 557

"Deselect All" on page 557

"Invert Select" on page 557

"Show Select Outline" on page 557

Select All

Choose **Select All** to select the entire texture pattern.

Deselect All

Choose **Deselect All** to clear the set of selected texels.

Invert Select

Choose **Invert Select** to clear each texel that was previously selected, and select each texel that was not selected.

Show Select Outline

Choose **Show Select Outline** to view or hide the outline that is drawn around selected texels.

Options Menu

The **Options** menu includes the following functions.

"Dynamic Update" on page 557

"Update Database" on page 558

Dynamic Update

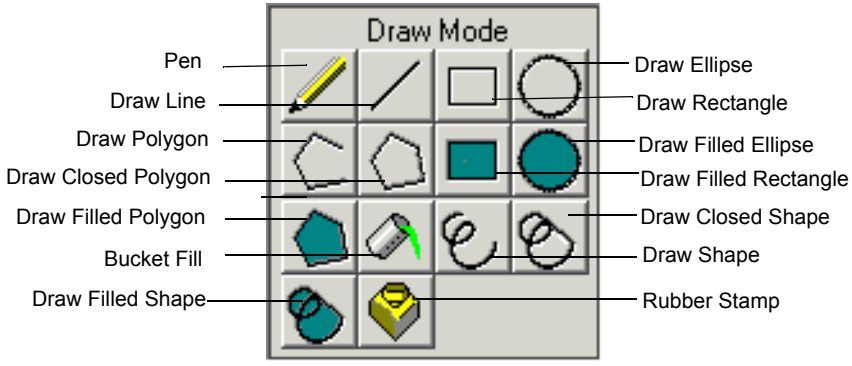
Choose **Dynamic Update** to dynamically apply the changes you make in the Texture Editor to the texture in the Graphics view. To view the texture in the Graphics view, **Draw Texture** in the View panel must be set (see "Draw Texture" on page 15-521).

Update Database

Choose **Update Database** if **Dynamic Update** is off to update the texture in the Graphics view with all changes that you have made in the Texture Editor. Changes are also updated in the Graphics view when you save the texture pattern.

Using the Draw Tools

The Draw tools in the Draw Mode palette apply the *pen tip* to the texture pattern when you click or drag the mouse in the Texture Editor. A pen tip is basically the shape and pattern that the pen draws (see "Defining the Pen Tip" on page 15-567). You can define a pen tip with a pen color, pen style, and a selection of the texture pattern. You then use the Draw tools to apply the pen tip in several shapes, for example, a straight line with the **Draw Line** tool, a filled rectangle with the **Draw Filled Rectangle** tool, or a copy of one portion of the texture pattern to another with the **Rubber Stamp** tool.



After you select and define a Draw tool with the pen tip, you enter points by pressing the mouse button, dragging the pointer to a new location in the texture pattern, and then releasing the mouse button. A red outline of the shape that you are defining appears in the Texture Pattern view as you drag the mouse, and the dimensions of the shape appear in the **X** and **Y** fields in the **Texel Display** at the bottom of the *Texture Editor* window.

Draw Tools

The Draw Mode palette provides the following tools.

"Pen" on page 559

"Draw Line" on page 559

"Draw Rectangle" on page 560

"Draw Filled Rectangle" on page 560

"Draw Ellipse" on page 560

"Draw Filled Ellipse" on page 560

"Draw Polygon" on page 560

"Draw Closed Polygon" on page 560

"Draw Filled Polygon" on page 560

"Draw Shape" on page 561

"Draw Closed Shape" on page 561

"Draw Filled Shape" on page 561

"Rubber Stamp" on page 561

"Bucket Fill" on page 562

Pen



The **Pen** tool draws in the texture pattern using the current pen tip.

Draw Line



Draw Line draws a straight line between two endpoints using the current pen tip. The first endpoint is the texel under the pointer when you first click the mouse button. The second endpoint is created the second time you click the mouse button.

Draw Rectangle



Draw Rectangle draws an unfilled rectangle using the current pen tip.

Draw Filled Rectangle



Draw Filled Rectangle draws a rectangle with an interior fill using the current pen tip.

Draw Ellipse



Draw Ellipse draws an unfilled ellipse using the current pen tip.

Draw Filled Ellipse



Draw Filled Ellipse draws an ellipse with an interior fill using the current pen tip.

Draw Polygon



Draw Polygon draws an open polygon as you enter vertex points with the mouse button. The current pen tip is traced along the polygon outline when you double-click the mouse button.

Draw Closed Polygon



Draw Closed Polygon draws an unfilled, closed polygon as you enter vertex points with the mouse button. Double-click the mouse button to enter the last vertex; the first and last vertices are automatically joined to close the polygon.

Draw Filled Polygon



Draw Filled Polygon draws a closed polygon as you enter vertex points with the mouse button. The polygon is filled with the current pen tip when you double-click the mouse button.

Draw Shape



Draw Shape draws a freeform, unclosed shape with the current pen tip. To use this tool, click the **Draw Shape** tool, and then hold down the mouse button and drag the mouse to draw. A red line appears, showing the path that will be painted when you release the mouse button.

Draw Closed Shape



Draw Closed Shape draws a freeform, closed, unfilled shape with the current pen tip. To use this tool, click the **Draw Closed Shape** tool, and then hold down the mouse button and drag the mouse to draw. A red line appears, showing the path that will be painted when you release the mouse button. The first and last vertices of the freehand shape are connected automatically when you release the mouse button.

Draw Filled Shape



Draw Filled Shape draws a freeform, closed, filled shape with the current pen tip. To use this tool, click the **Draw Filled Shape** tool, and then hold down the mouse button and drag the mouse to draw. A red line appears, showing the path that will be painted when you release the mouse button. The first and last vertices of the freehand shape are connected automatically when you release the mouse button.

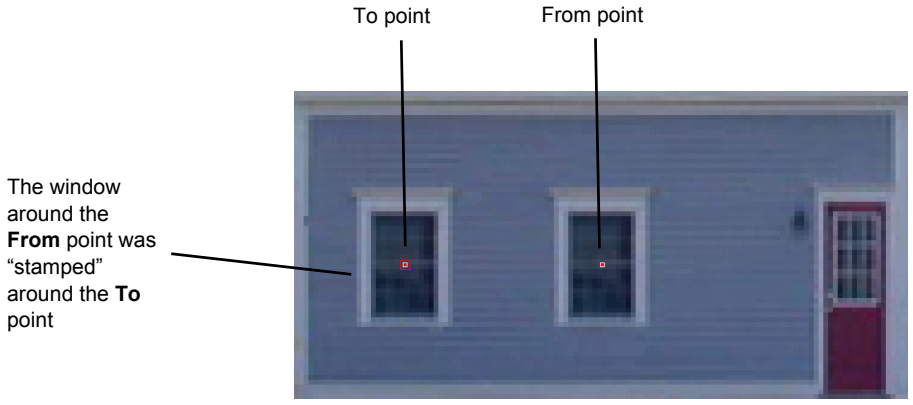
Rubber Stamp



Rubber Stamp copies one portion of the texture pattern onto another portion using the shape of the pen tip. After you select a pen tip shape, you click a **From** point with the mouse button, and then click a **To** point. The texels around the **From** point are "lifted" and "stamped" around the **To** point with each mouse click.

After the **From** and **To** points have been entered, each subsequent mouse click causes the offset between the **To** point and the current mouse position to be calculated. This offset is applied to the **From** point, and a set of texels in the shape of the pen tip is copied from that area to the current mouse position. The effect is that you can "sweep"

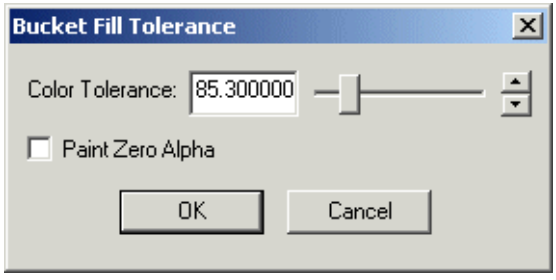
out a copy of the portion around the **From** point as you drag the cursor. Each stroke of the **Rubber Stamp** tool paints more of the portion.



Bucket Fill



Bucket Fill assigns the current pen color to texels that match the texel color that you choose. Double-click the **Bucket Fill** tool to open the *Bucket Fill Tolerance* dialog box. Parameters in the dialog box control which texels are assigned the color.



The *Bucket Fill Tolerance* dialog box has these parameters:

Color Tolerance

Controls the amount of variation allowed in each component of a color for it to match the texel color that you click. You can either enter a value or slide the slider bar for a value between 0 and 512, which is the amount of distribution in color space. A large tolerance allows more variation in color. If you dynamically change the tolerance value

after clicking a texel, the texels that are filled will update dynamically to reflect the new value of tolerance.

Paint Zero Alpha

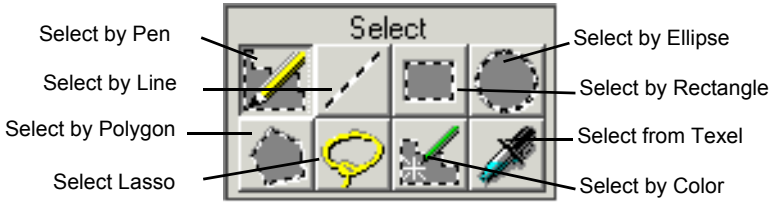
Check the **Paint Zero Alpha** checkbox if you want color assigned to texels with a zero-alpha component (transparency).

Selecting Texels

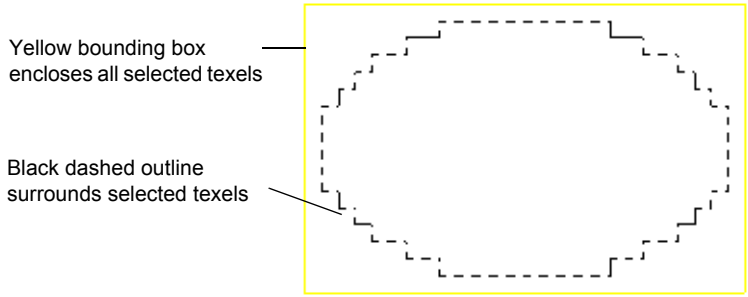
The Texture Editor has a number of functions that select texels for image processing or for loading into the pen tip. For example, you can use **Select** tools to select texels by a line, a freehand path that you draw, or by a color that you select in the texture pattern. You can also select all texels in the texture pattern using functions in the **Select** menu (see "Select Menu" on page 15-556). After you select texels, you can load the selection into the pen tip, blend the pen tip color, or apply other functions to modify the selection of the texture pattern.

Select Tools

Most of the **Select** tools let you select texels by holding down the mouse button and dragging an outline in the texture pattern. When you release the mouse button, all texels enclosed in the outline are selected. The Select palette contains the following **Select** tools:



A dotted black outline appears around each area of selected texels in the texture pattern. A yellow bounding box appears to show the set of all selected texels.



You can use Shift, Ctrl, or both while you use the Select tools to control your selection, as described below.

Adding to the Set of Selected Texels

Hold down Shift while making a selection to add texels to the set.

Subtracting From the Set of Selected Texels

Hold down Ctrl while making a selection to subtract texels from the set.

Inverting the Set of Selected Texels

Hold down both Shift and Ctrl while making a selection. Texels in the selected area become deselected; texels that were not previously selected become selected.

The Select palette provides these tools.

“Select by Drawing” on page 565

“Select by Line” on page 565

“Select by Rectangle” on page 565

“Select by Ellipse” on page 565

“Select by Polygon” on page 565

"Select Lasso" on page 565

"Select by Color" on page 566

"Color From Texel" on page 566

Select by Drawing



Selects a portion of the texture pattern using the shape of the pen as you drag the pen over the texture pattern.

Select by Line



Selects a portion of the texture pattern as you drag a line between two endpoints.

Select by Rectangle



Selects a portion of the texture pattern as you drag a rectangular border between two corners.

Select by Ellipse



Selects a portion of the texture pattern as you drag an elliptical border between two corners.

Select by Polygon



Selects an area of the texture pattern as you select each vertex of a polygon with the mouse button. Double-click the last vertex to close the polygon.

Select Lasso

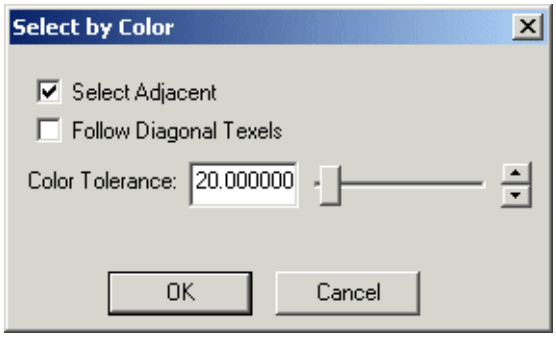


Selects the portion of the texture pattern bounded by the texels over which you drag the mouse. The first and last texels that you select connect automatically, and the selected area closes when you release the mouse button.

Select by Color



Selects texels that match the color of a texel that you click in the texture pattern. Double-click the **Select by Color** tool to open the *Select by Color* dialog box. Parameters in the dialog box control which texels are selected.



The *Select by Color* dialog box has these parameters:

Select Adjacent

Only texels that are adjacent to the texel that you click are selected.

Follow Diagonal Texels

Texels that are diagonally adjacent to the texel that you click are selected.

Color Tolerance

Controls the amount of variation allowed in each component of a color for it to match the texel color that you click. You can either enter a value or slide the slider bar for a value between 0 and 512, which is the amount of distribution in color space. A large tolerance allows more variation in color. If you dynamically change the tolerance value after clicking a texel, the texels that are selected will change dynamically to reflect the new value of tolerance.

Color From Texel



Sets the pen color to the color of the texel that you click in the texel pattern.

Defining the Pen Tip

You define the pen tip for the **Draw** tools with a *pen color* (RGB, RGBA, Intensity, Intensity-alpha), *pen style* (shape), or a *pen selection*. The pen color is a color of a texel in the texture pattern; the pen style defines the shape of the pen that draws the pen color onto the texture pattern; the pen selection is a portion of the texture pattern that you select.

"Pen Color" on page 567

"Pen Style" on page 568

"Pen Selection" on page 569

Pen Color

The pen color is a color that you select from either the texture pattern or the Windows *Color* dialog box. For IRIX, Creator uses the color picker. After you select a pen color, you can use a **Draw** tool to apply that color.

To select a color from the texture:



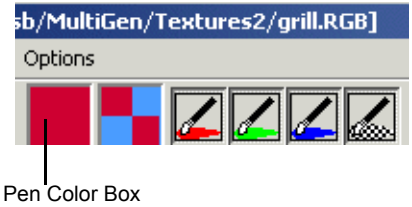
Step 1. Click the **Color From Texel** tool in the Select palette to select a color from the texture pattern.

Step 2. Position the pointer over a texel with the desired color in the texture pattern, and click the mouse button.

Step 3. Choose **View/Pen Window** to see the set pen color.

To select a color from the Windows *Color* dialog box:

Step 1. Double-click the Pen Color box at the top of the *Texture Editor* window to open the Windows *Color* dialog box.



Step 2. Select a color from the color chooser. The selected color appears in the Pen Color box at the top-left of the *Texture Editor* window. You can also choose **View/Pen Window** to see the set pen color.

Note: If you are editing an intensity texture pattern, the grayscale equivalent of the pen color you select is used in the Texture Editor.

Pen Style

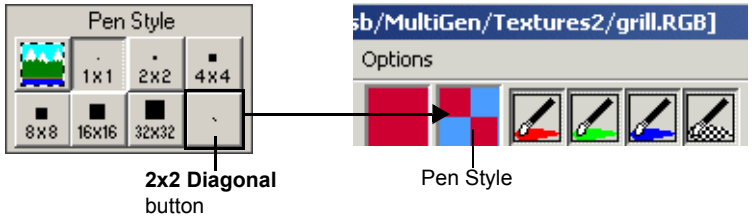
The pen style is the shape of the pen that draws the pen color onto the texture pattern. The pen style also determines the shape of the pen for the **Rubber Stamp** tool that stamps a selection onto the texture. The size of the texture pattern in texels defines the shape. For example, if you select **2 x 2** in the Pen Style palette and use a **Draw** tool, you will draw a line that is 2 texels wide. If you use the **Rubber Stamp** tool, a 2 x 2 area of the texture pattern will be copied and stamped (see "Rubber Stamp" on page 15-561).

To set the pen style:

Step 1. Set the pen color as described in "Pen Color" on page 15-567, or click the **Rubber Stamp** tool in the Draw Mode palette.

Step 2. Click a texel size in the Pen Style palette. You can choose **1 x 1**, **2 x 2**, **4 x 4**, **8 x 8**, **16 x 16**, **32 x 32**, or a **2 x 2 Diagonal** texel pattern in which a diagonal

pattern is selected. The pen style appears in the Pen Style box at the top of the *Texture Editor* window.



Pen Selection

You can define the pen tip with a selection of the texture pattern. After you select a portion of the texture pattern, you can use any **Draw** tool to apply the selection on the texture pattern.

To set the pen selection:

Step 1. Click a **Select** tool in the Select palette. See "Select Tools" on page 15-563 for descriptions of the **Select** tools. You can also choose functions in the **Select** menu. See "Select Menu" on page 15-556 for a description of the **Select** menu.

Step 2. Select a portion of the texture pattern with a **Select** tool.

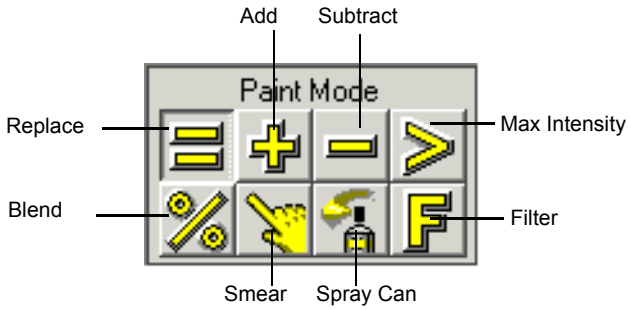
Step 3. Click the **Load Pen With Selection** tool in the Pen Style palette. The pen selection appears in the Pen Style box at the top of the *Texture Editor* window.





Choosing a Paint Mode

The **Paint** tools in the Paint Mode palette determine the effects that are produced by the **Draw** tools. For example, you can add the pen color to the texture pattern by a specified percentage, blend the pen tip into the texture pattern by a specified percentage, or apply a filter to the texture pattern.



Paint Mode Tools

The Paint Mode palette provides these tools.

"Replace Mode" on page 571

"Add Mode" on page 571

"Subtract Mode" on page 571

"Max Intensity Mode" on page 572

"Blend Mode" on page 572

"Smear Mode" on page 573

"Spray Can Mode" on page 574

"Filter Mode" on page 574

Replace Mode

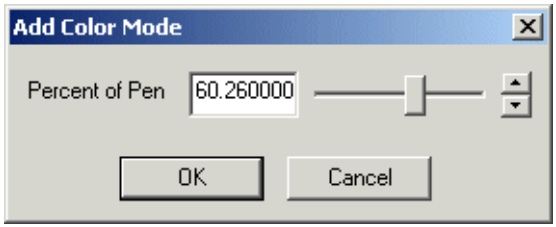


Replaces the texels under the pointer with the texels in the pen tip. This is the default setting.

Add Mode



Adds a percentage of the color intensities of texels in the pen tip to the texels in the texture pattern that are touched by a **Draw** tool. The area of the texture pattern becomes brighter as you add the texel colors. To change the percentage of color intensity, double-click the **Add Mode** tool to open the *Add Color Mode* dialog box.



The *Add Color Mode* dialog box has this parameter:

Percent of Pen

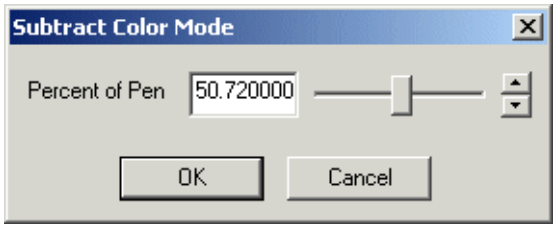
Controls the percentage of texel color intensities in the pen tip that are added to the texels in the texture pattern touched by a **Draw** tool. You can either enter a value or slide the slider bar for a value between 0 and 100.

Subtract Mode



Subtracts a percentage of the color intensities of texels in the pen tip from the texels in the texture pattern that are touched by a **Draw** tool. To change the percentage of color

intensity, double-click the **Subtract Mode** tool to open the *Subtract Color Mode* dialog box.



The *Subtract Color Mode* dialog box has this parameter:

Percent of Pen

Controls the percentage of texel color intensities in the pen tip that are subtracted from the texels in the texture pattern touched by a **Draw** tool. You can either enter a value or slide the slider bar for a value between 0 and 100.

Max Intensity Mode

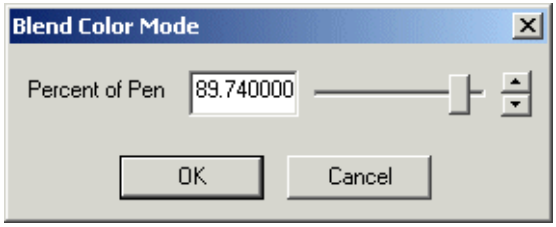


Compares each texel in the texture pattern affected by a **Draw** tool to the corresponding texel in the pen tip and sets the color in the texture to the maximum hue, lightness, and saturation (HLS) value. For example, if the pen tip contains a white texel and the corresponding texel in the texture is gray, the white texel replaces the gray one. But if the pen tip texel is gray and the texel in the texture is white, no change occurs.

Blend Mode



Blends the pen tip into the texture pattern by adding a percentage of the color intensities of texels in the pen tip to the texels in the texture pattern that are touched by a **Draw** tool. To change the blend percentage, double-click the **Blend Mode** tool to open the *Blend Color Mode* dialog box.



The *Blend Color Mode* dialog box has this parameter:

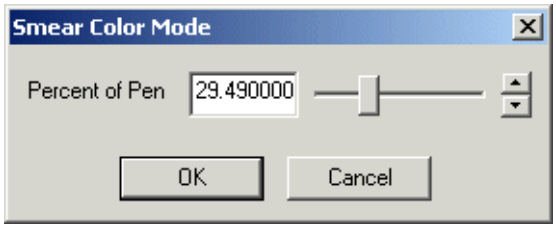
Percent of Pen

Controls the percentage of texel color intensities in the pen tip that are blended with the texels in the texture pattern touched by a **Draw** tool. You can either enter a value or slide the slider bar for a value between 0 and 100.

Smear Mode



Blends the colors in the texture as you drag the pointer over them. The effect is similar to smearing wet paint with your finger. The pen style defines the shape of the "finger." To change the smear percentage, double-click the **Smear Mode** tool to open the *Smear Color Mode* dialog box.



The *Smear Color Mode* dialog box has this parameter:

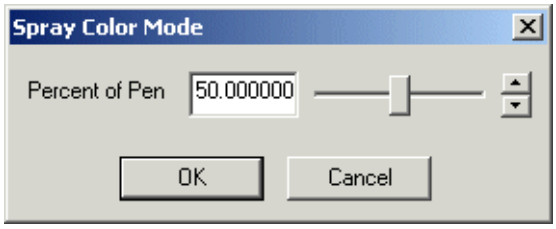
Percent of Pen

Controls the percentage of texel color intensities in the pen tip that are blended with the texels in the texture pattern touched by a **Draw** tool. You can either enter a value or slide the slider bar for a value between 0 and 100.

Spray Can Mode



"Sprays" the pen tip onto the texture pattern. To change the percentage of color intensity, double-click the **Spray Can Mode** tool to open the *Spray Color Mode* dialog box.



The *Spray Color Mode* dialog box has this parameter:

Percent of Pen

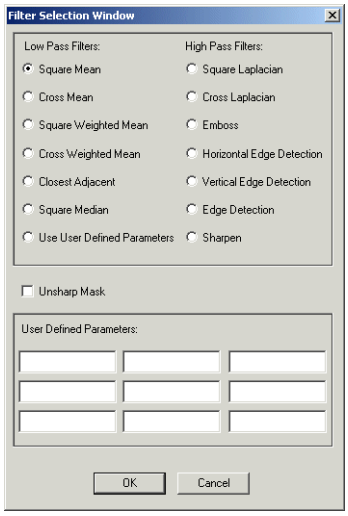
Controls the percentage of texel color intensities in the pen tip that are "sprayed" onto the texture pattern touched by a **Draw** tool. You can either enter a value or slide the slider bar for a value between 0 and 100.

Filter Mode



Applies the current filter to the area of the texture affected by the **Draw** tool. Filters control whether a high or low frequency (amount of variation of light and dark colors)

is passed through to the texture. To change the filter applied by this tool, double-click the **Filter Mode** tool to open the *Filter Selection* window.



The *Filter Selection* window contains filters (3 x 3 convolution values) for blurring or sharpening images. Click one of the options to select either a **Low Pass** or **High Pass** filter. The **Low Pass** filters are used to blur textures, and the filters at the top of the list blur more than those towards the bottom. The **High Pass** filters are used to sharpen textures. You can select the **Use User Defined Parameters** option in the **Low Pass Filters** list, and then enter your own 3 x 3 convolution values in **User Defined Parameters** at the bottom of this window.

Set the **Unsharp Mask** checkbox to highlight details in the area of the texture affected by the **Draw** tool. A low frequency filter is passed through, and the unfiltered texels are added to the original texture resulting in a higher contrast.

Exercise: Drawing With a Pen Tip

In this exercise, you will load a model in Creator and draw a shape on its texture. You will define a pen tip to use for the **Draw Filled Ellipse** tool, and then use a **Paint** tool to create a special effect.

Step 1. In the Creator database window, open a model from the Gallery folder that uses textures, such as `commanche.flt` or `racecar.flt`. The Gallery folder

is in the MultiGen directory, or in the directory where Creator is installed. Keep the database window open.

Step 2. Open the Texture palette, and select a texture. Choose **Edit/Edit Pattern** to open the texture in the Texture Editor.

Step 3. In the *Texture Editor* window, choose **Options/Dynamic Update** to dynamically apply the changes you make to the texture in the Graphics view.

Step 4. To define your pen tip, click the **Color From Texel** tool in the Select palette, and click a texel in the texture pattern with the desired color. The Pen Color box at the top of the *Texture Editor* window displays the selected color.



Step 5. Click a texel size in the Pen Style palette. The Pen Style box at the top of the *Texture Editor* window displays the selected pen style.

Step 6. Click the **Blend Mode** tool in the Paint Mode palette. Double-click it to open the *Blend Color Mode* dialog box. Set the **Percent of Pen** to *.50* so that the color in the pen tip blends 50 percent with the texture pattern color.



Step 7. Click the **Draw Filled Ellipse** tool and drag the mouse to draw an ellipse anywhere on the texture. Draw ellipses in several areas of the texture to see the color blend with different texels.



Step 8. Click the database window to see the texture changes on the model. You can also save the texture in the Texture Palette to save the changes.

Displaying and Protecting Texels

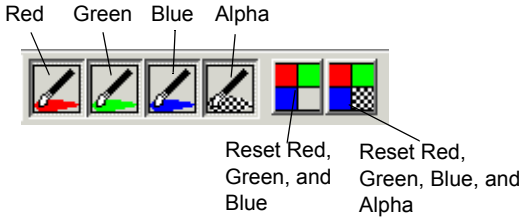
The Texture Editor has several controls for displaying and hiding texel colors, as well as protecting them from editing:

- **Texture Component** buttons selectively display the red, green, blue, and alpha components of every texel color in a texture pattern. These buttons also turn off the display of components.
- **Component Freeze** checkboxes protect the red, green, blue, and alpha components of texel colors without turning off their display. Like **Texture Component** buttons, **Component Freeze** checkboxes affect the entire texture pattern.

Displaying Texture Components

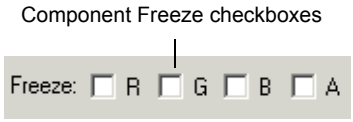
Texture Component buttons turn off the display of color and transparency components in every texel of the texture pattern. For example, if **G** is on and **R**, **B**, and **A** are off, only the green intensities of the texture pattern are displayed.

To suppress a component display, click the corresponding **Texture Component** button. To return a component to its normal state, click the component button again. To simultaneously display red, green, and blue components; or red, green, blue, and alpha components, click the **Reset Red, Green, and Blue** button or the **Reset Red, Green, Blue,** and **Alpha** button.



Freezing Texture Components

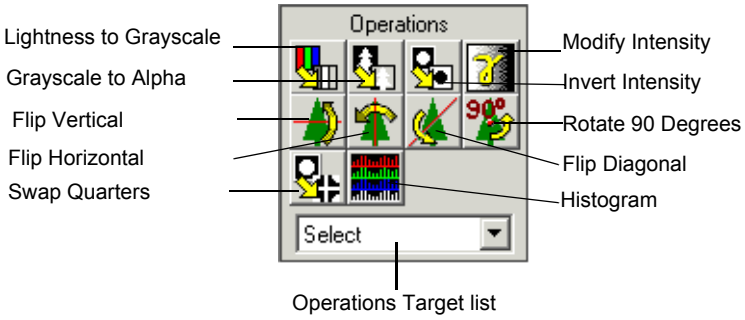
Component Freeze checkboxes protect color components from being edited, without affecting the way colors are displayed. To freeze the red, green, blue, or alpha component, set the **R**, **G**, **B**, or **A** checkbox, respectively.



Using Texture Operations Tools

The Operations palette contains tools for changing the orientation of a selected area of texels and for changing texel color components. You can rotate selected texels, flip texels vertically or horizontally, or swap the positions of each quarter of the texture pattern. Texels can be in the entire texture pattern, texels stored in the pen tip, or selected texels. You can also edit the components of the texel colors and convert the colors in the selected texels to a different color mode (RGB to grayscale, for example).

To use the tools in the **Operations** palette, you first select which texels you want modified in the **Operations Target** list. You can choose **Pattern** to select all texels in the entire texture pattern, **Select** to select texels with a **Select** tool, or **Pen** to select those texels stored in the pen tip. After you select the texels to change, click an **Operations** tool in the **Operations** palette.



"Operations Target" on page 579

"Flipping, Rotating, and Swapping Texels" on page 579

"Changing Intensity and Color" on page 580

Operations Target

Choose an **Operations Target** in the **Operations Target** list to specify the set of the texels that you want the **Operations** tools to process. You can choose one of the following:

- **Pattern** - Performs an operation on all texels in the texture pattern
- **Select** - Performs an operation on texels selected with a **Select** tool
- **Pen** - Performs an operation on texels in the current pen tip

Flipping, Rotating, and Swapping Texels

Flip, **Rotate,** and **Swap** tools change the orientation of texels you select. You can mirror texels across centerlines, rotate all affected texels, or swap all quarters of the texture across diagonal lines. To apply the changes, select the texels that you want to change or the entire texture pattern, and then click one of these tools.

"Flip Vertical" on page 579

"Flip Horizontal" on page 579

"Flip Diagonal" on page 580

"Rotate 90°" on page 580

"Swap Quarters" on page 580

Flip Vertical



Mirrors texels across the horizontal centerline of the affected area.

Flip Horizontal



Mirrors texels across the vertical centerline of the affected area.

Flip Diagonal



Mirrors texels across the diagonal centerline (from lower-left to upper-right) of the affected area.

Rotate 90°



Rotates all affected texels 90 degrees counter-clockwise.

Swap Quarters



Divides the affected area into four equal sections and switches the positions of the sections diagonally. The following figure shows the effects of swapping quarters:

Before using Swap Quarters



After using Swap Quarters. Each quarter of the texture is switched across its diagonal line.



Changing Intensity and Color

Image Processing tools change the intensity and color of texels that you select. To apply the changes, select the texels that you want to change or the entire texture pattern, and then click one of these tools.

"RGB to Grayscale" on page 581

"Grayscale to Alpha" on page 581

"Invert Intensity" on page 581

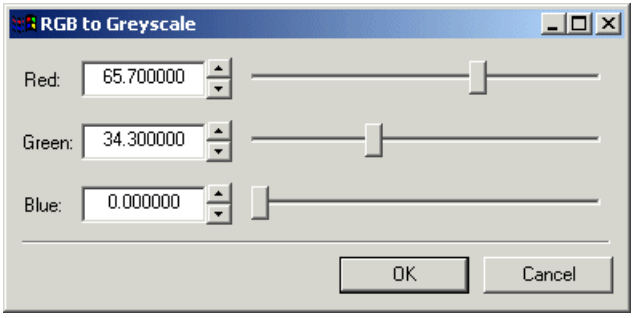
"Modify Intensity" on page 582

"Histogram" on page 582

RGB to Grayscale



Uses the intensity of the red, green, and blue components of each selected texel in an RGB or RGBA texture to convert the texture to grayscale. To improve the resolution of the grayscale image, you can click the **RGB to Grayscale** tool to adjust one or more intensities of red, green, and blue texel components in the *RGB to Grayscale* dialog box.



Texel areas that were distinct as full-color areas may have too little or too much contrast when converted to grayscale.You can change the percentage of intensity for each texel color by entering a number or moving the slider bar for a percentage between 0 and 100. The selected area of the texture pattern changes dynamically as you adjust color intensities.

Grayscale to Alpha



Uses the lightness component of each selected texel to generate its alpha component. The original image must be in grayscale.

Invert Intensity



Reverses the intensities of red, green, blue, and alpha components for each selected texel to produce a “negative” image.

Modify Intensity



Changes the intensity (brightness) of selected texels according to a gamma value. Click the **Modify Intensity** tool to enter a gamma value in the *Modify Gamma* dialog box.



You can enter a value or move the slider bar for a gamma value between 0 and 5. The selected area of the texture pattern changes dynamically. As you decrease the value of gamma below 1, the texel colors become darker than the original. As you increase the value of gamma above 1, the texel colors become lighter. The intensity of the selected texels using a gamma value can be expressed in the following equation:

$$\text{Newintensity} = \left(\frac{\text{Oldintensity}}{255}\right)^{1/g}$$

where g=Gamma

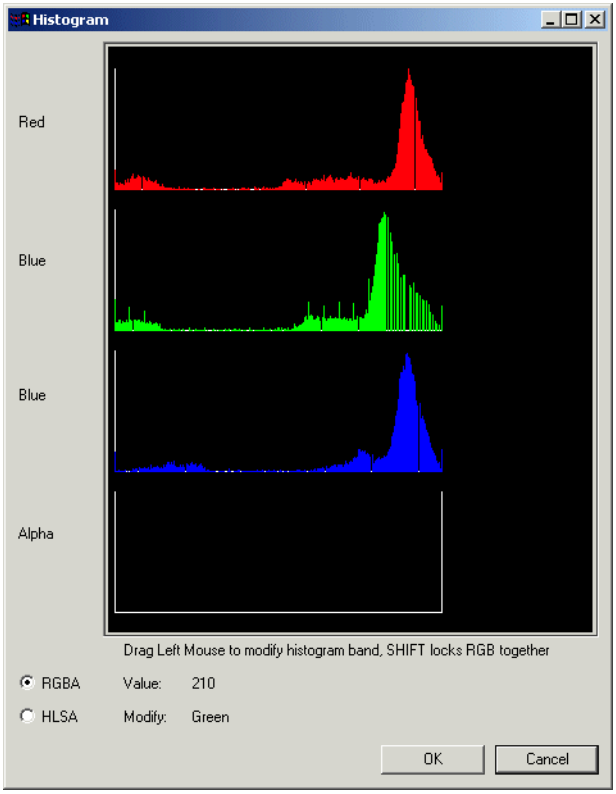
Histogram



Opens the *Histogram* dialog box and generates a frequency distribution graph for either the RGB-alpha or HLS-alpha bands of the affected texels. For each red, green, blue, or alpha color component, or hue, lightness, and saturation component, the horizontal axis represents color intensity, and the vertical axis represents the amount of texels for each intensity.

To change the intensities of the colors in the selected texels, drag the pointer over the histogram. As you drag the pointer across the **Hue** graph, the hues for the red, green, and blue components change. The RGBA or HLSA value changes dynamically at the

bottom of the *Histogram* dialog box. Press the Shift key and the mouse button to lock RGBA intensities together while you change them.



Exercise: Selecting and Flipping Texels

In the following exercise, you will load a texture in the Texture Editor and select an area of the texture with a **Select** tool. You will flip the texels across the vertical centerline in the selected areas with an **Operations** tool.

Step 1. In the Creator database window, open the Texture palette, and select a texture.

Step 2. Choose **Edit/Edit Pattern** to open the texture in the Texture Editor.

Step 3. Click the **Select by Rectangle** tool, and drag the mouse to select a rectangular area anywhere on the texture.

Step 4. In the **Operations** palette, choose **Select** from the **Operations Target** list to modify the texels in the selected area.

Step 5. Click the **Flip Horizontal** tool to flip the texels across the vertical centerline.

Step 6. Save the texture. The changes in the selected area are saved as well.

Using Texture Maneuvering Tools

The **Maneuver** tools let you translate, scale, rotate, and skew selected texels. To avoid leaving a hole in the texture pattern, these functions fill the original footprint of the selected texels with a copy of the original texels.

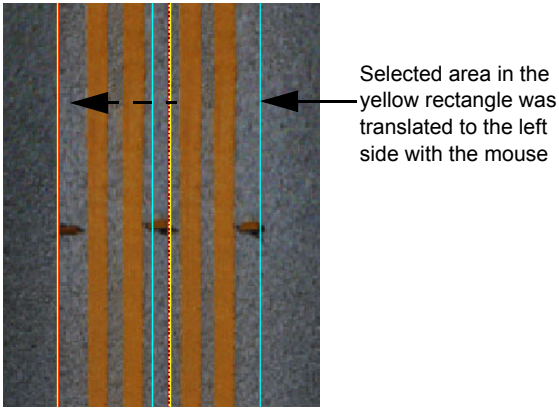
To use the tools in the **Maneuver** palette, you first select the texels in your texture pattern using a **Select** tool. A yellow rectangle surrounds the selected texels. After you select the texels to change, click a **Maneuver** tool in the **Maneuver** palette. A red rectangle is added with four vertices to drag with the mouse. Both rectangles are removed when you select a new tool.



Translate



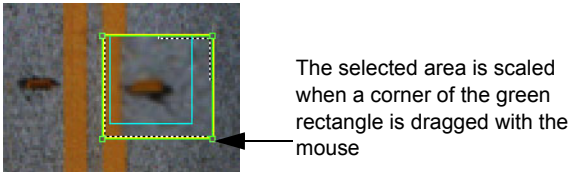
Moves selected texels to a new location in the texture pattern as you drag them with a mouse.



Scale



Resizes a selected set of texels as you drag one of the corners of the green rectangle surrounding the selection in either the *x* or *y* direction.

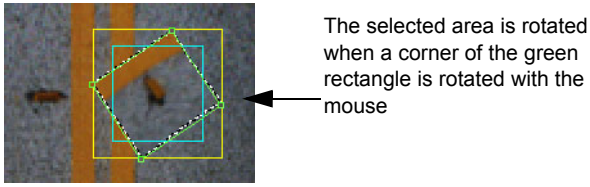


Note: If you are going to scale and translate a set of texels, it is best to translate the texels to the desired area first, and then scale them.

Rotate



Rotates a selected set of texels as you drag one of the corners of the green rectangle surrounding the selection.

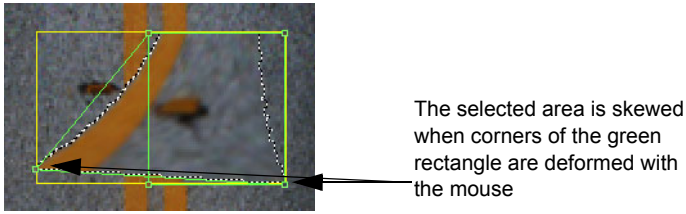


Note: If you are going to rotate and translate a set of texels, it is best to translate the texels to the desired area first, and then rotate them.

Warp



Skews a selected set of texels by deforming the green rectangle surrounding the selection in any direction.



Note: If you are going to warp and translate a set of texels, it is best to translate the texels to the desired area first, and then warp them.

The Texture Compositor

The Texture Compositor in the **ModTexture** toolbox enables you to easily combine multiple non-repeating textures on selected geometry into a single texture. Within the Creator modeling environment, a single texture file is generated for easier processing, paging, and display at runtime. It is also efficient to load a single texture file into Creator.

Each original texture in the single image is saved as a subtexture. See "Subtexture" on page 478 for more information about how they can be used for texturing other geometry.

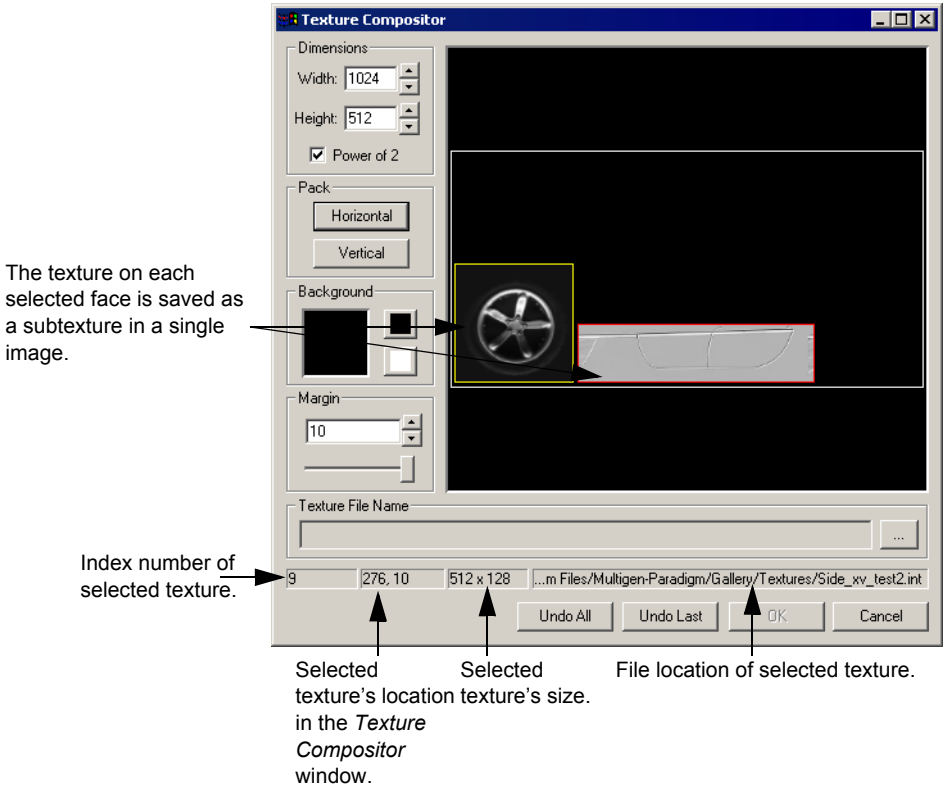
Note: There is a trade-off between the size of the composite texture and the amount of memory it takes to load it and store it in memory. There is a point at which it is not beneficial to group the textures into one texture file. If you know what type of video card you are using you can check the card's available texture memory to determine whether your textures stretches or exceeds the memory allowed. See "Using Chop Texture to Reduce Texture Size" on page 464 for more information about calculating texture memory.

To combine selected textures:

Step 1. In the Graphics view, select at least two faces with the textures that you want to combine. The textures must be non-repeating.

Step 2. Choose **Palettes/Texture Compositor** to open the *Texture Compositor* window. Each subtexture is outlined in yellow in the single image (indicated by a white border). A selected texture is outlined in red, and the

index number, location in the *Texture Compositor* window, size in pixels, and file location appears in the status bar at the bottom.



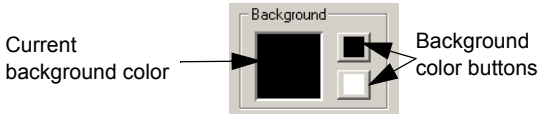
Step 3. Enter or select the width and height of the image. If you set **Power of 2**, each dimension is a factor of 2, and the image size can range from 512 x 1024 texels up to 4096 x 4096 texels.

Step 4. You can pack the textures together with no margins between them either horizontally or vertically. The textures are aligned in the bottom left corner of the single image.

If you want the textures in specific locations in the *Texture Compositor* window, you can drag them with the left mouse button. The location of a selected texture appears in the status bar at the bottom of the window.

Step 5. You can select the background color of the single image by clicking either the black or white color button, or clicking the current background color to

select a different color in the Color palette. The background color becomes blended with the edges.



Step 6. Enter a value between 0 and 10 to set the size of the margin between the textures. You can also drag the slider with the left mouse button.

Step 7. Click **Undo Last** to undo the last action, or **Undo All** to undo all the actions at the same time.

Step 8. Click the **Browse** button to save the composite image in a texture file. When you save the new image, the texture's appearance on the geometry is not altered.



Step 9. Click **OK** to save the changes and close the *Texture Compositor* window, or **Cancel** to close the window without saving changes.

The Texture Sequencer

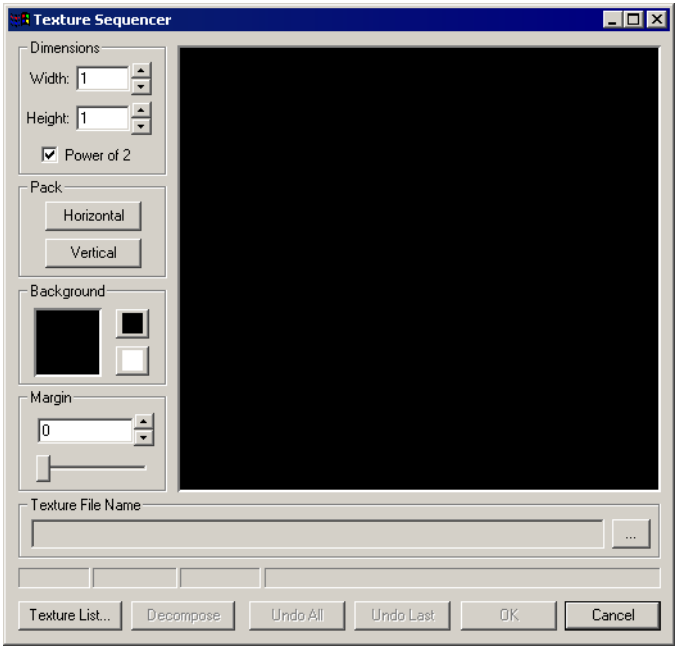
The **Texture Sequencer** tool in the **ModTexture** toolbox enables you to place a set of sequential textures, such as a series of frames of an explosion, into a single image file. Each texture is a subtexture of the image file. The subtextures in the image file can be used as animation frames in a runtime system.

Within the Texture Sequencer, you can group the subtextures or pack them closely together to save texture space. You can set up the subtexture frames in an automated loop to preview an animation.

You can load the single image file into Creator and apply the subtextures to polygons using the Flip Book animation tool. See "Flip Book Tool" on page 128 for more information.

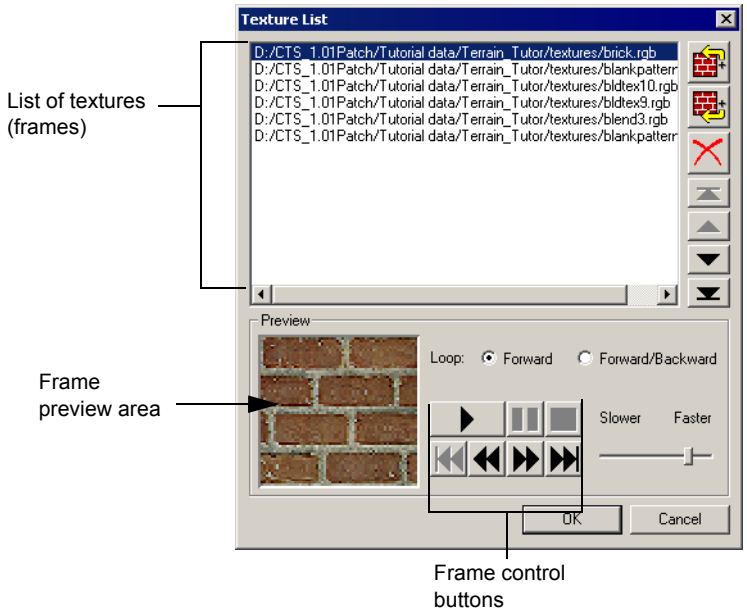
To create a sequence of textures:

Step 1. Choose **Palettes/Texture Sequencer** to open the *Texture Sequencer* window.



If the current texture in the Texture palette has subtextures, a message will appear that asks if you want to load it when you open the Texture Sequencer. You can select the texture in the *Texture Sequencer* window, and then click **Decompose** to decompose all of the subtextures and rearrange them in an animation loop.

Step 2. Click **Texture List** to load textures in the *Texture List* window. If you decomposed subtextures in a texture file, all of the subtextures will already be listed in this window.





Step 3. Click **Add Frame Before** to add a texture before the selected texture in the texture list. If you hold down the Shift key and select several textures in the **Open** dialog box to load at once, they will appear in decreasing order in the texture list.



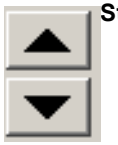
Step 4. Click **Add Frame After** to add a texture after the selected texture in the texture list. If you hold down the Shift key and select several textures in the **Open** dialog box to load at once, they will appear in decreasing order in the texture list.



Step 5. Click **Delete** to delete the selected texture from the texture list.



Step 6. Click **Move Frame To Top** to move the selected texture to the top of the texture list. This is inactive when the first texture is selected in the texture list. Click **Move Frame To Bottom** to move the selected texture to the bottom of the texture list. This is inactive when the last texture is selected in the texture list.



Step 7. Click **Move Frame Up** or **Move Frame Down** to move the selected frame up or down one level in the texture list. **Move Frame Up** is inactive when the first texture is selected in the texture list. **Move Frame Down** is inactive when the last texture is selected in the texture list.

Step 8. To create an animation loop of frames to preview, select either **Forward** to animate the frames sequentially in the texture list, or **Forward/Backward** to animate the frames forward sequentially, and then backward in the texture list.

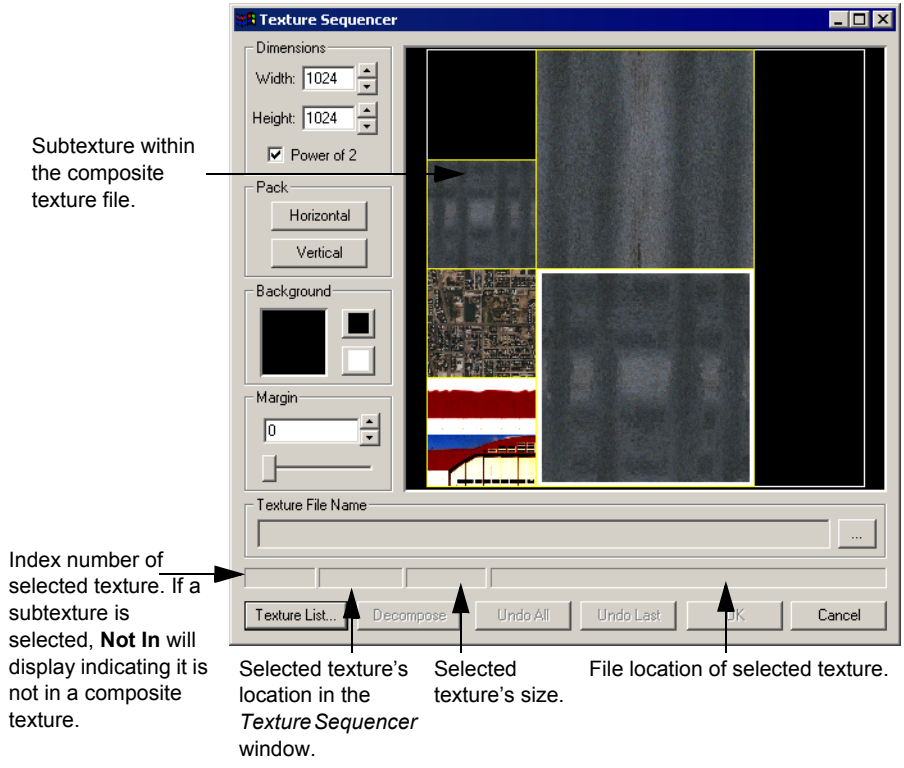
Step 9. Click the frame control buttons, such as **Play**, **Pause**, and **Next Frame** to control the animation loop.

Step 10. Drag the slider between **Slower** and **Faster** to control the speed of the animation.

Step 11. Click **OK** to save the changes and close the *Texture List* window, or **Cancel** to close the window without saving changes.

The subtextures appear in the *Texture Sequencer* window. You can use the different functions to arrange them and set the dimensions of the final tex-

ture file. You can also drag subtextures to different areas with the left mouse. A red border outlines the selected textures.

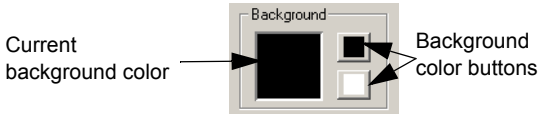


Step 12. Enter or select the width and height of the image. If you set **Power of 2**, each dimension is a factor of 2, and the image size can range from 512 x 1024 texels up to 4096 x 4096 texels.

Step 13. You can pack the textures together with no margins between them either horizontally or vertically. The textures are aligned in the bottom left corner of the single image.

Step 14. You can select the background color of the single image by clicking either the black or white color button, or clicking the current background color to

select a different color in the Color palette. The background color becomes blended with the edges.



Step 15. Enter a value between 0 and 10 to set the size of the margin between the textures. You can also drag the slider with the left mouse button.

Step 16. Click **Undo Last** to undo the last action, or **Undo All** to undo all the actions at the same time.



Step 17. Click the **Browse** button to save the composite image in a texture file.

Step 18. Click **OK** to save the changes and close the *Texture Sequencer* window, or **Cancel** to close the window without saving changes.

The Fractal Texture

The **Fractal Texture** tool in the **Palettes** menu enables you to create texture patterns using a random fractal algorithm. The patterns generated are particularly useful for sky textures but can be applied to many different applications such as terrain. Opposite edges within the patterns match, so they repeat seamlessly when applied in a scene. When you have completed the texture pattern, you can save the texture and load it into the texture palette of the current database.

The pattern is generated using random variations of two colors. These two colors default to blue and white, which make nice looking sky textures, but you can choose any two colors for different effects.

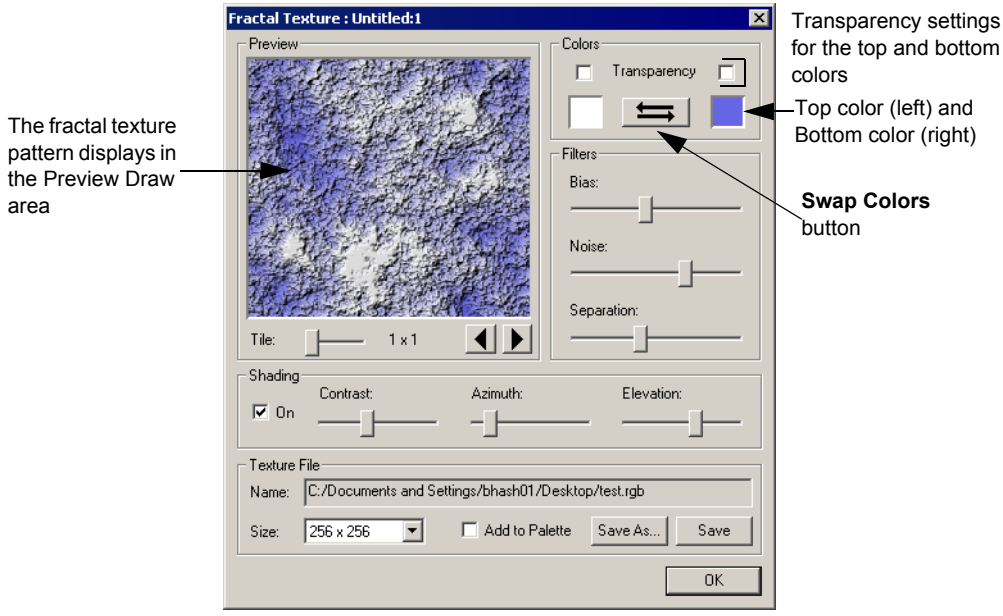
After selecting the colors that you want to use, you can adjust the filter controls for blending effects of the two colors in the pattern. You can also create lighting and shading effects to show relief in the pattern.

After you apply the texture to geometry in your database, you can select the texture in the texture palette to make it the current texture, and re-edit the fractal pattern in the

Fractal Texture tool. You will simultaneously see the texture change in the Graphics view.

To create a fractal texture:

Step 1. Choose **Palettes/Fractal Texture** to open the *Fractal Texture* window. The pattern using the current settings displays in the Preview Draw area. It automatically updates as you change the settings.

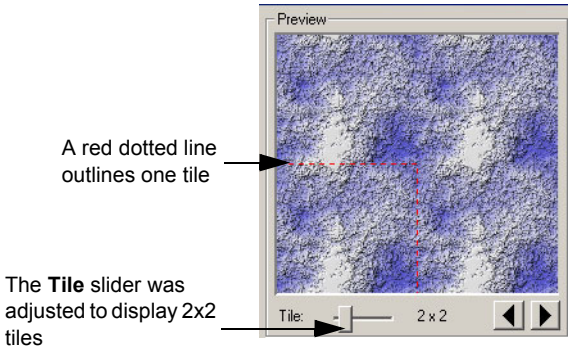


Step 2. To change the default colors (blue and white) in the pattern, click the left color swatch to choose the top color, and the right color swatch to choose the bottom color. You can click the **Swap Colors** button to switch the top and bottom colors.

Step 3. Set a **Transparency** checkbox to add an alpha component to either the top or bottom color. Alpha is added to the pattern so that the intensity of the alpha in any texel is equal to the intensity of the color in that texel. For example, when the color of a texel is 100 percent of the color, the alpha value of that texel is 100 percent transparent.

Step 4. You can adjust the **Tile** slider bar to control the number of tiles (or repetitions of the pattern) that display in the Preview Draw area. You can

choose a value from 1x1 to 10x10 tiles to preview. A red dotted line outlines one tile.



Step 5. You can click the right and left arrow buttons to generate new and different patterns using a random fractal algorithm. The algorithm uses an integer value that can be varied for random effects.

Step 6. You can adjust these filters for different effects:

- **Bias** - Controls how much of each color is mixed into the pattern. Slide this slider to the right to mix in more of the top color or to the left to mix in more of the bottom color.
- **Noise** - Controls how much noise is introduced into the pattern.
- **Separation** - Controls how sharply the colors blend together.

Step 7. You can create lighting and shading in the pattern by adjusting these settings:

- **Shading On** -Sets shading on and off. If shading is on, then the slider controls in this area are enabled. You can use them to create different lighting effects in the pattern. Shading adds a relief effect to the pattern.
- **Contrast** - Controls the contrast in the pattern.
- **Azimuth** - Controls the direction the light source is facing when applying lighting to the pattern. As you change this value, a small icon representing the light source (position and direction) displays within the Preview Draw area.
- **Elevation** - Controls the elevation of the light source when applying lighting to the pattern. As you change this value, a small icon

representing the light source (position and direction) displays within the Preview Draw area.You can also adjust the azimuth and elevation values of the light source simultaneously by pressing and dragging the left mouse button within the Preview Draw area. As you do this, the light source follows the mouse, and a small icon representing the light source (position and direction) displays on top of the pattern within the Preview Draw area.

When the mouse is over the center of the pattern, the light source will be at maximum elevation (pointing straight down). When the mouse is closer to any edge of the pattern, the light source will lower in elevation and point toward the center of the pattern. As you move closer to the edge of the pattern, the elevation becomes lower.

Step 8. Slide the **Size** slider to specify the size of the texture pattern. The texture must be square with dimensions that are a power of two in the range of 16x16 to 512x512.

Step 9. Set **Add to Palette** to load the texture pattern into the texture palette of the current database when it is saved. If the pattern is already in the texture palette, the existing pattern will be replaced.

Step 10. Click **Save As** to select a new file name and location for the texture pattern. The path displays in the **Name** field.

Click **Save** to save the pattern to the current file name and location.

Note: If you have specified transparency, be sure to save the file in an alpha format such as .rgba or .inta. If you do not, any transparency will be lost when you save the file.

Step 11. Click **OK** to save your changes and close the window.

16 Light Sources and Database Shading

Light and shade define and highlight objects. They can set a mood or create drama in an otherwise uninteresting model. In Creator, you shade a database using a combination of three types of light sources and four shading modes.

Some of the following topics describe the different types of lights and light sources and how to create, apply, and modify them. Other topics describe how to shade a database, and the different shade attributes that give the models in your database a more real-world look.

Light Sources



Light sources illuminate all or part of a database, and produce shadows. You can create three types of light sources in Creator:

- *infinite light sources* create general lighting over the entire database
- *local light sources* radiate light in all directions from a specific location
- *spot light sources* radiate light in a specific direction from a specific location.

A light source is defined in the database at all times. The current light source is displayed on the **Current Light Source** button in the Toolbar. The default light source type is infinite.

You create, define, modify, and store light sources in the Light Source palette. Each database has its own Light Source palette, which can be saved and used in other databases. Each light source in the palette is assigned a unique index number.

To select a light source as the current light source, select it in the Light Source palette.



Light sources are enabled in a database as modeling lights or by creating and positioning a *light source node*. Modeling lights illuminate the entire database and are attached to the eyepoint .Light source nodes are created with the **Create Light Source** tool in the **Create Tools** toolbox. They are attached to a group node parent regardless of the current modeling mode. Light source nodes are assigned the properties of the current light source when they are created. A maximum of eight enabled light sources can be rendered at one time although you can have more than eight in the database hierarchy.



Once a light source node has been created and enabled, you must perform **Calculate Shading** to view its effect in the database (described in "Shading the Database" on page 622). The **Lights On/Off** View control turns the effects of lighting on and off. When lighting is on, an L appears in the upper left corner of the Graphics view. The keyboard shortcut to toggle lights on or off is the L key.



By default, Creator displays Local and Spot light source nodes in the Graphics view. Local lights are represented by closed boxes. Spot lights are represented by boxes that open in the direction of the lighting vector. Infinite light sources have no graphical representation. You can toggle the display of light source nodes on and off by clicking the **Draw Light Sources** icon in the View panel.



You can modify a light source node's attributes or reposition a light source node in the database by setting the modeling mode to **Light Source**, selecting it, and then choosing **Attributes/Modify Attributes** or **Attributes/Place Light Source**.

Using the Light Source Palette

Each database has its own Light Source palette that can contain up to 64 independent light source definitions. Light Source palette file names have a .lts extension.

You use the Light Source palette to create, modify and store light source definitions and palettes. One light source is selected in the Light Source palette at any given time. To select a different light source, click it in the Light Source palette.



To open the active database's Light Source palette, click the **Current Light Source** button on the Toolbar, or choose **Palettes/Light Source**.

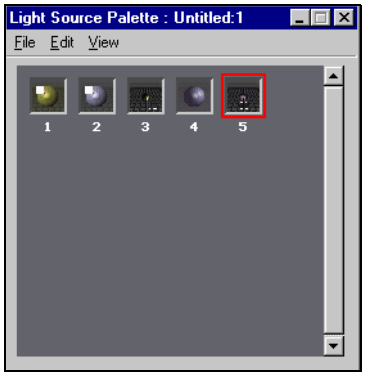
Using the menus in the Light Source palette, you can load palettes, work with light sources, and specify how you want the light sources to appear in the palette. From the File menu, you can load saved light source palettes, save new and existing light source palettes, and close the Light Source palette. Using the **Edit** menu, you can create new light sources, and cut, copy, paste, delete and modify existing light source definitions. (The **Cut**, **Copy**, and **Paste** functions can be used between different light source palettes.)

You can use the **Cut**, **Copy**, **Paste** and **Delete** functions in the Light Source palette's **Edit** menu to duplicate existing light source definitions in the current palette, add

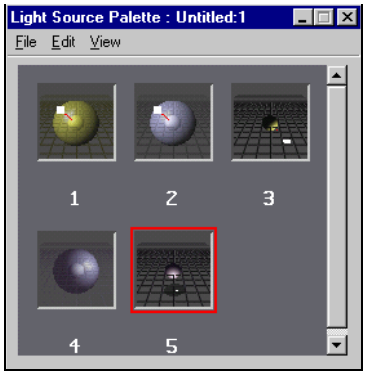
definitions from another Light Source palette to the current palette, or delete a definition.

The View menu controls the ways you can view light source definitions:

- **Thumbnail** displays small thumbnail views of each light source in the current Light Source palette. This is the default setting. You can deselect this preference to show larger views of the light sources in the palette.



Thumbnail on



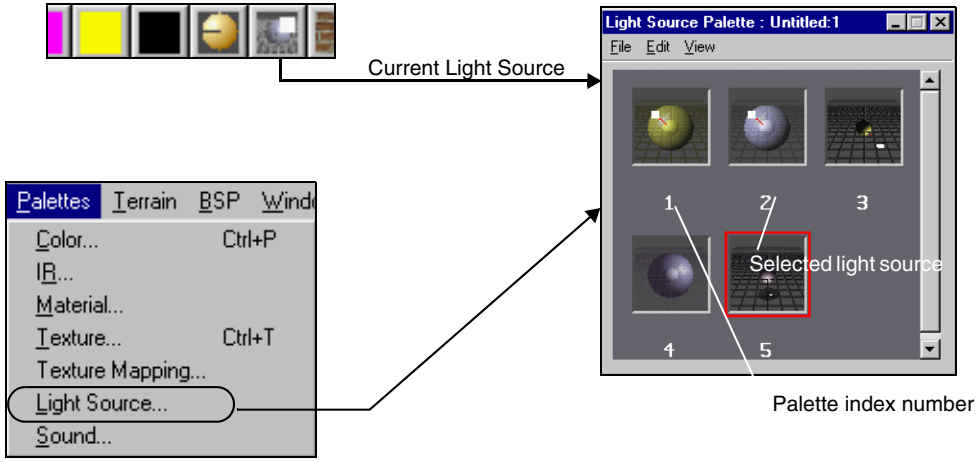
Thumbnail off

- When **Thumbnail** is not set, only a portion of the light sources are visible at any time in a Light Source palette. Choosing **Fit on Select** immediately displays the current light source.

The following sections describe how to create, modify, and store light sources, and save and load light source palettes.

Creating Light Sources

Every light source in the Light Source palette has a light source palette index number. A light source is associated with a light source node by assigning this index number to the node's Light Source attribute.



To Create a New Light Source

Step 1. Open the Light Source palette.

Step 2. Choose **Edit/New**. A thumbnail is added to the palette, representing the new light source.

Step 3. Double-click the new light source, or select it and choose **Edit/Edit** to open the *Modify Light Source* dialog box. In this dialog box, you can see the default attributes assigned to all new light sources.

Step 4. Modify the light source attributes as needed to achieve the lighting effects you want (see "Modifying Light Source Definitions" on page 603).

To Delete a Light Source

Step 1. Open the Light Source palette.

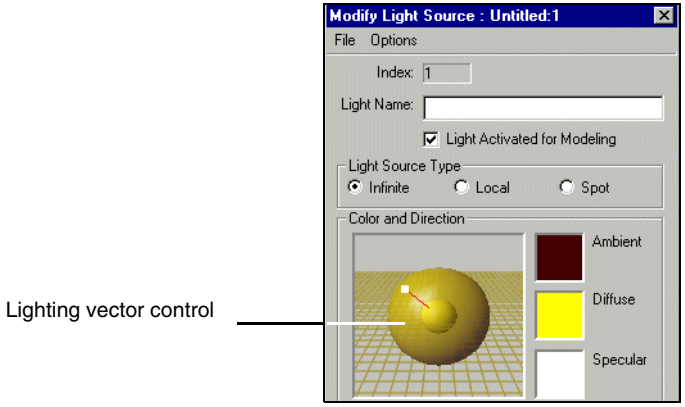
Step 2. Choose **Edit/Delete**.

Modifying Light Source Definitions

All newly-created light sources have a default definition (Infinite, modeling light). You change a light source's definition through the *Modify Light Source* window. To open the *Modify Light Source* window, open the Light Source palette and choose **Edit/Edit** or double-click a thumbnail in the Light Source palette.

The *Modify Light Source* window automatically updates the current light source information when a new light source is selected in the Light Source palette. Choosing **File/Revert** restores the last light source definition saved in the Light Source palette. A light source definition is saved when you close the *Modify Light Source* window or load a different light source into the modification window. To restore the original default light source definition, choose **File/Default**.

The *Modify Light Source* window contains a graphic representation of the light source, called a lighting vector control. The lighting vector control shows changes in the light source as you modify it.



The **Options** menu in the *Modify Light Source* window determines how changes to light sources will be put into effect in Lit lighting mode (see "Database Shading" on page 620). **Options/Dynamic DB Update** updates the Graphics view as soon as a light source parameter changes. Otherwise, the Graphics view updates only when you select **Options/Update Database** or click within the database window.

To modify a light source definition:

Step 1. Select the light source to be edited.

Step 2. Choose **Edit/Edit** in the Light Source palette, or double-click the light source definition. The *Modify Light Source* window opens.

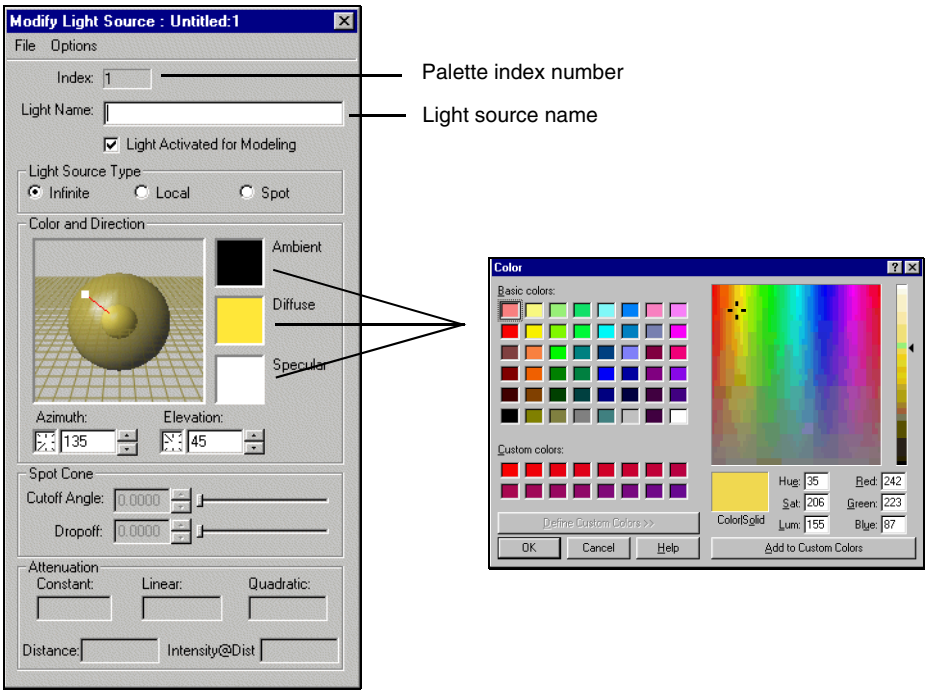
Step 3. Set or clear **Light Activated for Modeling** as needed.

When **Light Activated for Modeling** is set it is automatically enabled in the database. If **Light Activated for Modeling** remains set and you enable the same light source definition in a light source node, you will have two light sources.

Step 4. Set **Light Source Type**, if necessary.

Step 5. Change the **Ambient**, **Diffuse**, or **Specular** color by clicking the color swatch next to the corresponding label. The **Color Editor** opens.

Step 6. Select a new color, using the intensity slider to lighten or darken the color, and then click **OK**. Note how the lighting vector control updates immediately.



Step 7. You can achieve the greatest flexibility in lighting by defining white light sources and applying material properties to the different faces. Material properties control the color and the degree of glare or highlight reflected off the faces to which they are applied when a light source shines on them. For more information about material properties, see "Applying Material Properties" on page 394.

Step 8. Set the **Direction** parameters as needed.

Step 9. Set the **Spot Cone** parameters if **Light Source Type** is Spot. You can set **Attenuation** parameters if **Light Source Type** is Spot or Local.

Step 10. When you are satisfied with the change, choose **File/Close** to close the *Modify Light Source* window.

Modify Light Source Parameters

Modify Light Source parameters define the light source's type, color, brightness, and glare.

Light Name

Assigns an optional name to the light source definition.

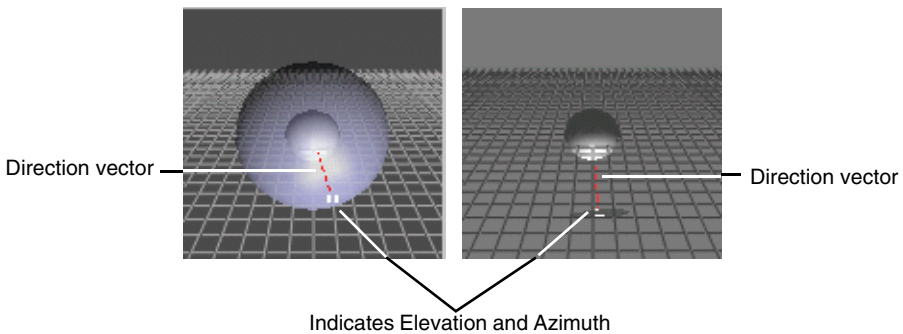
Index

Displays the palette index number of this light source definition.

Lighting Vector Control

Provides a graphic interface for changing the light source definition. The effect of this control on each type of light is described below.

The position of the dot relative to the sphere in the display represents the light source **Azimuth** and **Elevation**. When editing an infinite light source, the red line in the display represents the direction vector of the light, pointing from the dot towards the center of the sphere. The direction vector displays as a solid red line when it is in front of the sphere, and a broken line when it is behind the sphere.



When editing a local light source, the graphic control has no effect. There is no single direction vector associated with a local light source.

When editing a spot light source, the red line defines the axis of the spot light's cone of effect, shining from the dot towards the center of the sphere.

Database transformations override the defined position of light sources in the palette.

Light Activated for Modeling

Turns on a modeling light (one that illuminates the entire database) using this light source definition (see “Modeling Lights” on page 611).

A light source definition turned on as a modeling light can also be referenced by individual light source nodes.

Light Source Type

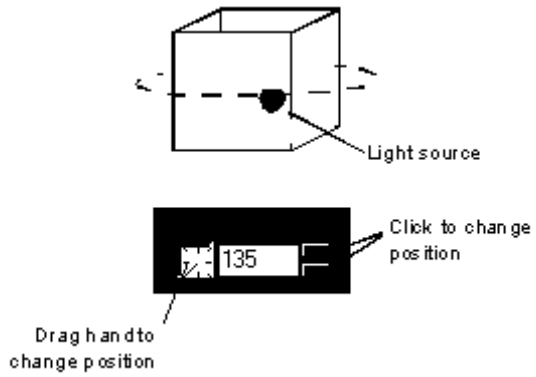
Determines the type of light source.

Infinite light sources shine along a direction vector and produce general overall lighting. When an infinite light source illuminates a portion of a database, the same vector is applied to every affected vertex. Infinite light sources do not have a point of origin in the database. They require the least computation to display.

Local light sources are located at a point in space and shine in all directions. When a local light source illuminates a portion of a database, a distinct vector is calculated from the light source position to each affected vertex. Local light sources produce a more natural lighting effect than infinite light sources, especially when lighting objects that have sharp angles between faces. However, they place a greater computational burden on the computer than infinite light sources.

Azimuth

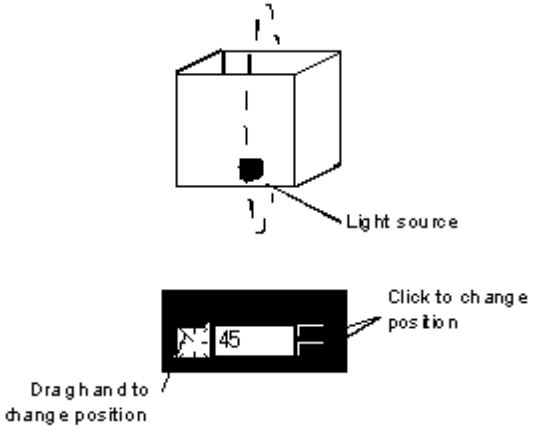
Indicates the rotation of the light source along a great circle around the horizontal axis. To change position, click the arrows to the right of the current position, or change the clock controls.



Elevation

Indicates the rotation of the light source around the vertical axis. An elevation of 0 negates the **Azimuth** control, because it places the light source on the vertical axis. To

change position, click the arrows to the right of the current position, or change the clock controls.



Ambient

Controls the intensity of the constant illumination added to the database by a light source.

Diffuse

Controls the color of the light that a light source adds to the database.

Specular

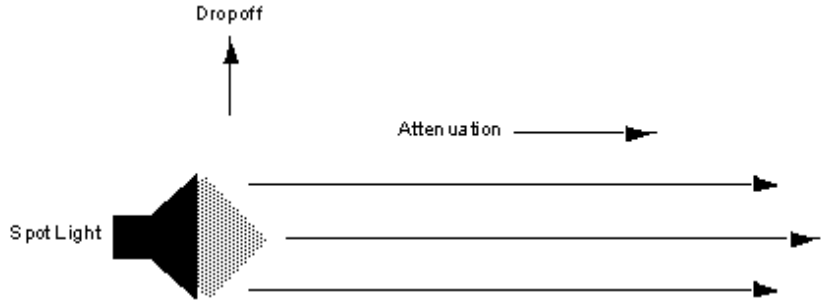
Affects the color of the highlight or glare produced on objects in the database by the light source.

Spot Cone Cutoff Angle

Controls the angle between the axis of the cone defining a spot light and the sharp cutoff that occurs along the edge of the cone (spot light only).

Spot Cone Dropoff Exponent

Controls the dropoff that occurs within the cone of effect defined for a spot light. Increasing this value makes the light source more focused (spot light only).



Attenuation

Controls the decrease in light as the distance from a local or spot light source increases. This parameter has no effect on infinite lights.

The attenuation factor is defined by the equation:

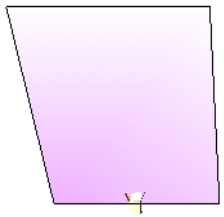
$$\text{Attenuation factor} = \frac{1}{\text{Constant} + \text{Linear}(d) + \text{Quadratic}(d^2)}$$

where d is the distance from the light source position to a vertex and **Constant**, **Linear**, and **Quadratic** are the terms you supply in the *Modify Light Source* window. If you enter a **Distance** for attenuation effects, Creator calculates values for the **Constant**, **Linear**, and **Quadratic** terms. You can also enter an **Intensity** value for a distance, which is a number between **0** and **1**. If you enter **1**, which is full intensity, you will not see attenuation effects.

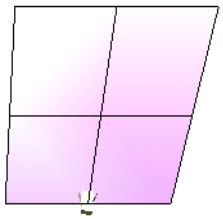
Light Source Tips

This section contains suggestions and information to keep in mind when creating and modifying light sources.

- Set the **Ambient** value fairly low; a high value causes the surface to appear washed-out.
- When shining a spot light on a large surface (such as a wall), the face must be split into two ore more smaller faces in order to achieve the proper effect; light sources interact with vertex normals and vertex colors.



Shading with face not split



Shading with face split into smaller polygons

- Set the **Ambient** color to black or a dark color to minimize cumulative ambient color effects.
- Setting the **Specular** color to white allows face and material colors to show.
- Using bright modeling lights can wash out the face and material color applied to your model.

Enabling Light Sources

Once a light source has been created and defined it must be enabled, either as a modeling light or as a light source node. Only eight light sources can be enabled in a database at any one time.

Modeling Lights

Modeling lights are light sources that are activated by directly enabling the light source in the Light Source palette. Modeling lights illuminate everything in the database. They are attached to, and move with, the eyepoint. They have no graphical representation in the database, and cannot be positioned.

Modeling lights are used for preview purposes only and do not export to the realtime system with the database.

To enable a light source as a modeling light, set Light Activated for Modeling in the *Edit Light Source* dialog box.

Light Source Nodes

Light source nodes function as group nodes in the hierarchy. They can be attached at any point in the hierarchy where a group node is legal. Light source nodes allow you to vary the position or direction of the light source and restrict their effect to specific portions of the hierarchy. They are exported to the realtime system as part of the database.

Unlike modeling lights, you can activate the following functions with light source nodes:

- Assign a unique position and direction vector to the light source using **Attributes/ Place Light Source** or the **Rotate** tool in the **Maneuver** toolbox.
- Restrict the lighting effect produced by the node to its descendants in the hierarchy.
- Turn the light source on or off dynamically. For example, different light sources can be enabled for corresponding levels of detail, or for the children of a switch node.

To create a light source node:

Step 1. Select the appropriate light source as the **Current Light Source**.

Set a group node as the database parent.



Step 2. Open the **Create** toolbox.



Step 3. Click **Create Light Source**. The current light source is automatically assigned to the new node, and the node is placed at the database origin.

Placing and Viewing Light Source Nodes

Modeling lights are attached to the eyepoint. Since they move with the eyepoint, they cannot be positioned. Infinite, Local, and Spot light sources that are enabled through light source nodes are initially located at the database origin and can be repositioned using **Attributes/Place Light Source** or the **Rotate** tool.

You can scale light sources. However, this also affects the amount of light produced, so you should use caution when doing this.

Infinite light sources are vectors that have direction but no position. Local light sources have a specific position from which light radiates in all directions. Spot light sources have both position and direction.

The following light sources types have no graphical representation and cannot be viewed in the Graphics view:

- Infinite light source nodes
- Disabled light source nodes
- Modeling lights

The display of enabled light source nodes in the Graphics view and their size are determined by the State preferences **Draw Light Sources** and **Light Source Size** (see "Setting Lighting Preferences" on page 618). Local lights are represented by closed boxes. Spot lights are represented by boxes that open in the direction of the lighting vector.

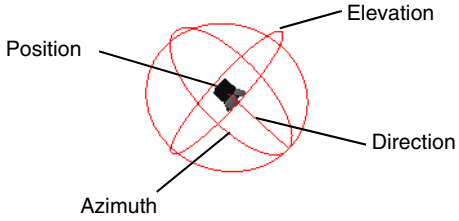
To position a light source:

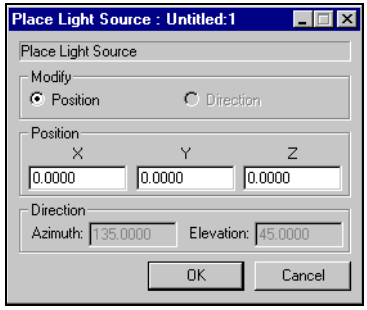


Step 1. Set the modeling mode to **Light Source**.

Step 2. Select the light source in the Graphics view, or select the light source node in the Hierarchy view.

Step 3. Choose **Attributes/Place Light Source**. The *Place Light Source* dialog box opens and displays a graphical rotation guide in the Graphics view that shows the orientation of the light.





Step 4. Set the light source's **Position**. This is its coordinate location in the database. To change the position, edit the value of *X*, *Y*, and *Z*, or drag the red control point in the Graphics view with the mouse. **Position** has meaning only for local and spot light sources.

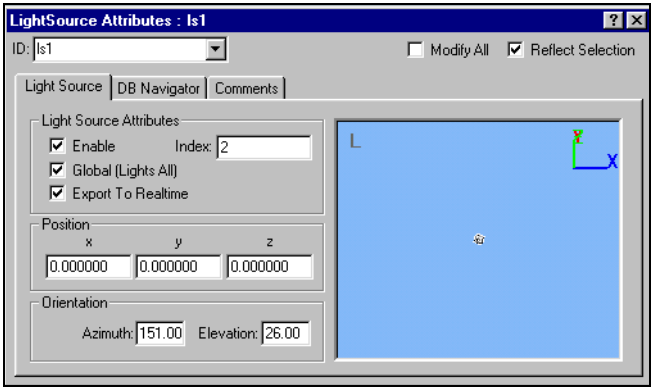
Step 5. Set the **Direction** for the light source, if the light source is a local or spot light, by turning the graphical rotation guide.

Step 6. Set the **Elevation**, or rotation around the vertical axis, by entering a new value in the **Elevation** text field or rotating the graphical rotation guide in the Graphics view with the mouse. **Elevation** has meaning only for infinite and spot light sources.

Step 7. Set the **Azimuth**, or rotation around the horizontal axis, by entering a new value in the **Azimuth** text field, or rotating the graphical rotation guide in the Graphics view with the mouse. **Azimuth** has meaning only for infinite and spot light sources.

Modifying Light Source Nodes

You can view and modify a light source node through its attributes window. The *Light Source Attributes* window contains all of the attributes assigned to it.



To view and modify light source node attributes:



Step 1. Set the modeling mode to Light Source.

Step 2. Select the light source in the Graphics view and choose **Attributes/Modify Attributes** (Ctrl+=), or double-click the light source node in the Hierarchy view. The *Light Source Attributes* window opens, displaying a graphic representation of the light source, its Light Source palette index number, and its current attributes.

Step 3. Be sure **Reflect Selection** is set. Make the desired changes to the attributes (see Light Source Attributes) and then close the window.

Light Source Attributes

Light Source Attributes window has three panels: Light Source, DB Navigator, and Comments. The Light Source panel and the general attribute window characteristics are described here. See “OpenFlight Nodes and Attributes” on page 825 for information about the DB Navigator and Comments panels.

ID

Displays the identifier (name) of the light source node.

Modify All

Applies the changes you make to the open attributes window to every light source node that is currently selected. See “Modify All” on page 827 for a more complete description.

Reflect Current Selection

Causes the current attribute window to display the attributes of the selected node rather than opening a new attribute window. See “Reflect Selection” on page 827 for a more complete description.

Index

Indicates the Light Source palette index of the light assigned to the node.

Enable

Turns the lighting effect of the light source node on or off.

Global (Lights All)

Applies the lighting effect of the light source node to the entire database. If this is turned off, the light source affects only the children of the light source node in the hierarchy.

Export to Realtime

Flags the node for use by the realtime system.

Position

Indicates the location of the light source in database coordinates. This attribute has meaning only for local and spot light sources. It defaults to (0,0,0). Assign light position manually, or select a light source node and choose **Attributes/Place Light Source.**

Orientation

Modifies the direction of the light. This attribute has meaning only for infinite and spot light sources. By default, the direction is (0, -1, 0) for infinite lights, and (0, +1, 0) for spot lights. Orientation is defined by **Elevation** (rotation about the z axis) and **Azimuth**

(rotation about the x axis). Set the **Light Direction** numerically, or select a light source node and choose **Attributes/Place Light Source**.

Saving/Loading Light Source Palettes

A light source palette can be saved to a file and used with other databases. You can develop custom Light Source palettes and save them in files that can be loaded by other databases. This makes it easy for similar databases to use similar light sources.

If you want your custom Light Source palette to load automatically whenever Creator launches, name the file mg1.lts and save it in the appropriate directory. Creator searches for the file mg1.lts, first in the folder specified by the MPI_CREATOR_RESOURCEPATH environment variable (see "Environment Variables" on page 43), and then in the working directory. The working directory for Creator for Windows is the directory where the Creator is located, and for Creator for IRIX it is the directory where Creator was launchedCreator application. If mg1.lts is not found in either of these directories, Creator loads a standard Light Source palette that it builds from source definitions.

To Save a Light Source Palette to Disk

Step 1. Open the Light Source palette.

Step 2. Choose **File/Save Palette**. The *Write Light Source File* dialog box opens.

Step 3. Enter a file name in the **File Name** field, and click **Save**.

To Load a Saved Light Source Palette

Step 1. Open the Light Source palette.

Step 2. Choose *File/Load Palette*.

Step 3. Select the name of the Light source palette to load from the *Load Palette* dialog box, and click **Open**.

To Save a Default Light Source Palette

Step 1. Open the Light Source palette.

Step 2. Choose **File/Save Palette**. The *Write Light Source File* dialog box opens.

Step 3. Set Save In to the folder specified by the MPI_CREATOR_RESOURCEPATH environment variable, if it was set. Otherwise, set it to the MultiGen-Paradigm\Creator directory (Windows), or to your working directory (IRIX).

Step 4. In the **File Name** field, type mg1, and click **Save**. The file will be saved with a .lts suffix.

Step 5. Restart Creator to activate the default palette.

To Copy a Light Source

Step 1. Choose **File/Load Palette**.

Step 2. Select the palette that contains the light source to be copied, and then click **Open**.

Step 3. Select the light source and choose **Edit/Copy**.

Step 4. Choose **File/Load Palette** and load the target Light Source palette.

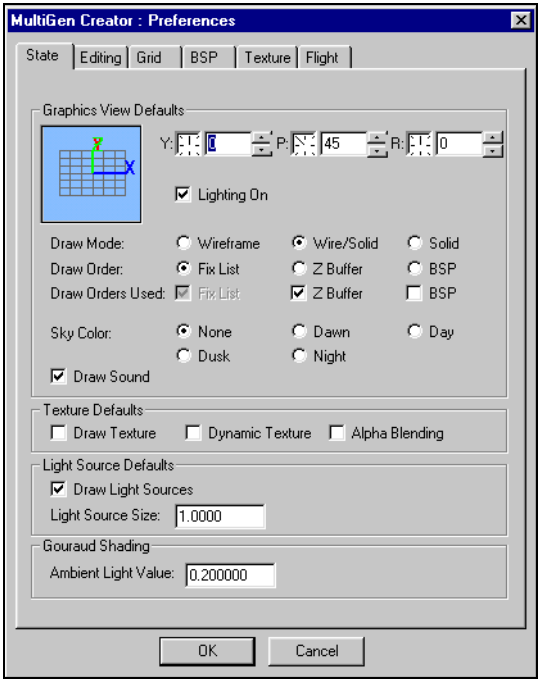
Step 5. Choose **Edit/New** to create a new light source.

Step 6. Choose **Edit/Paste** to copy the existing light source definition into the new light source.

Setting Lighting Preferences

Creator's preferences control lighting in the database. They enable you to see light sources in the Graphics view and determine Creator's default lighting condition.

State preferences set whether lighting is initially turned on or off, whether light sources will be visible in the Graphics view, and the size of the light source.



State Preferences for Lighting

The following State preferences affect lighting.

Lighting On



Lighting On, in Graphics View Defaults, turns lighting on in the Graphics view of the current database and any new database. When lighting is on an L appears in the upper left corner of the Graphics view. Lighting can also be turned on and off using the **Lights On/Off** control in the View controls, or simply by pressing the L key.

Draw Light Sources

The Light Source Default **Draw Light Sources** displays markers that represent enabled light source nodes in the Graphics view. Local lights are represented by closed boxes.

Spot lights are represented by boxes that open in the direction of the lighting vector. Infinite lights have no representation in the Graphics view.



Light Source Defaults: Light Source Size

Light Source Size, in Light Source Defaults, determines the size (in database units) of each light source box.

Database Shading

Shading creates lighting effects in a database. You can shade the entire database for an overall lighting effect, or shade different objects in the database individually to create unusual and interesting effects. Shading is performed using **Calculate Shading** in the **Attributes** menu.

You can shade a database without placing light source nodes if modeling lights have been enabled (see "Modeling Lights" on page 611). However, since modeling lights are only used for general lighting within Creator, the shading effect is not transferred to the realtime system. When modeling lights are enabled, shading is general, as if a light is illuminating the entire database. The result is soft shadows. Shading a database by placing light source nodes will have different effects, depending on the type of light source and the shading attribute used.

Creator has four shading modes:

- Flat
- Lit
- Gouraud
- Lit Gouraud

Flat shading shows faces in their assigned colors and disregards any material or lighting properties assigned.Vertex colors are used in static (Gouraud) lighting mode.

Vertex normals are used in dynamic (Lit) lighting mode. Vertex colors and normals are computed by calculating the effect each enabled light source has on the faces it illuminates. See "Calculate Shading Parameters" on page 622 for more information about shading modes.

After a database has been shaded, you can create more dramatic shading effects by modifying the shade attributes of different faces or by modifying vertex normals.

Shading Preferences

The Editing preference **Shade All When Nothing Selected** determines how shading is applied when **Attributes/Calculate Shading** is performed (see "Editing Preferences" on page 53). If **Shade All When Nothing is Selected** is enabled and nothing in the database is selected when you choose Calculate Shading, Creator shades everything in the database. When this preference is disabled **Calculate Shading** has no effect when nothing in the database is selected.

The State preference **Ambient Light Value** specifies a background intensity affecting the amount of ambient light applied to surfaces when Gouraud shading is applied (see "Shading Preferences" on page 621). Ambient light is the light present in the environment from reflective surfaces and ambient light sources. The amount of ambient light affecting a surface is equal to the sum of the **Ambient Light Value** plus the ambient component of each enabled light source. The total ambient light of each shaded vertex is computed by multiplying this sum by the ambient material reflectance of the face. It is a good idea to set ambient light fairly low, as a high value causes the surface to appear washed-out.

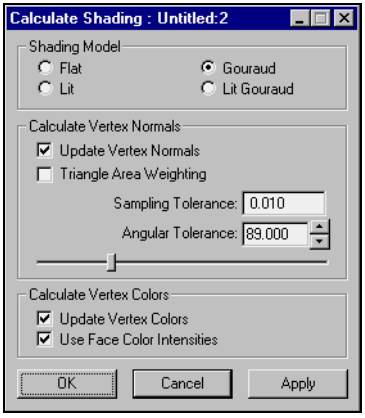
The Flight preference **Intensities for RGB Matching** (up to 128 shades) determines the number of color intensities that can be assigned to shaded vertices. When **Calculate Shading** is performed, Creator searches the Color palette for the closest match to each computed color. Restricting this value accelerates the **Calculate Shading** process. If the value of this field is one, vertex colors are restricted to the colors displayed in the Color palette. Each additional increment permits the next darker intensity level in the palette to be searched during shading. Recall that the Color palette contains 128 shades of each color that appears in the palette (only the brightest shade is displayed).

Shading the Database

To calculate shading on a database:

Step 1. If you want to shade everything in the database, choose **Info/Preferences**, click the Editing tab and make sure **Shade All When Nothing** Selected is set. Otherwise, select the faces you want to shade.

Step 2. Choose **Attributes/Calculate Shading** (Ctrl+G) to open the *Calculate Shading* dialog box.



Step 3. Set the **Calculate Shading** parameters, click **OK** to perform the shading and close the Calculate Shading dialog box, or click **Cancel** to cancel shading. See "Calculate Shading Parameters" on page 622 following this procedure for descriptions of each of the parameters.

Step 4. Click **Lights On/Off** in the View controls to view shading. You can also toggle lighting on and off by pressing the L key. An L appears in the upper-left corner of the Graphics view when lighting is on.



Calculate Shading Parameters

This section describes each of the parameters found in the *Calculate Shading* dialog box.

Shading Model

Sets the shading attribute of each selected face. If the Edit preference **Shade All if Nothing Selected** is set, the shading attribute will be applied to the entire database.

- **Flat**
  Displays faces in their assigned colors. No shading. Vertex normals are directed away from each face at a 90 degree angle.
- **Lit**
  Displays faces using the vertex normals generated by the **Calculate Shading** function, as well as applied material attributes, face color, and the effects of any light sources.
- **Gouraud**
  Displays faces using vertex colors generated by the **Calculate Shading** function. Interpolates color from vertex to vertex across a polygon's face.
- **Lit Gouraud**
  Displays faces using both Lit and Gouraud shading.

Update Vertex Normals

Calculates and saves vertex normals for every shaded face. Vertex normals are used when Lit shading mode is used.

Triangle Area Weighting

Weights the shading according to the area of the triangle defined by the two edges of each face that share a vertex. Larger faces contribute more to shading information than smaller ones.

Sampling Tolerance

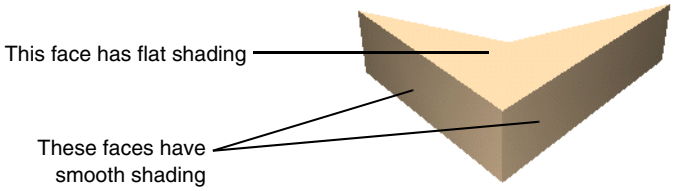
Determines whether two vertices are considered to share a coordinate location. If the distance between the vertices is less than **Sampling Tolerance** (in database units), the vertex coordinates are considered equal. You may need to use a small tolerance to shade small surfaces.

Angular Tolerance

Determines whether adjoining faces are assigned smooth or flat shading.

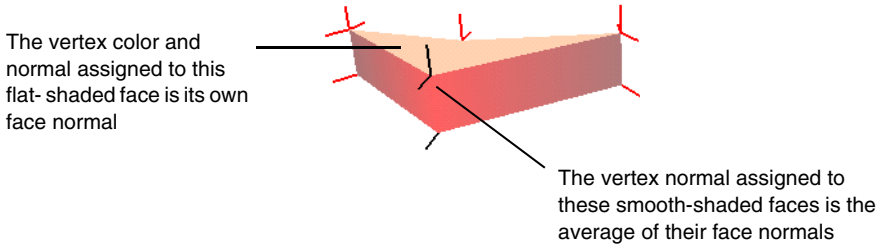
Smooth shading results when an averaged color (for Gouraud shading) or an averaged normal (for dynamic shading) is assigned to vertices that share a single coordinate location. This occurs if the angle between the normals of adjoining faces is smaller than the value of the **Angular Tolerance** parameter.

Flat shading results when different colors and normals are assigned to vertices that share a single coordinate location. This occurs if the angle between the normals of adjoining faces is larger than the value of the **Angular Tolerance** parameter.



Smooth shading is desirable for surfaces over which the normals change gradually, such as a sphere. Flat shading is more appropriate for surfaces that contain abrupt angles, such as a cube. The larger the value of **Angular Tolerance**, the more likely it is that a given pair of faces will be smoothly shaded.

In the example below, three adjoining faces share one vertex location. The angle between the normal of the top face and the normal of the side faces is greater than the **Angular Tolerance** of 89°. The angle between the normals of the two side faces is less than 89°. Therefore, the top vertex is assigned the color or normal determined by the top face alone, but an averaged vertex color or normal is assigned to both the vertices in the side faces.



Update Vertex Colors

Calculates and saves vertex colors for every shaded face. Vertex colors are used for Gouraud lighting. If specific colors have been assigned to vertices, this option should be disabled when **Calculate Shading** is performed. Update Vertex Colors applies the shaded face color to every vertex.

Use Face Color Intensities

Forces Gouraud colors to be selected from the same color band (in the color palette) as the original face color. This prevents shaded vertices from becoming excessively bright. It also overrides the Color palette search, which has the effect of speeding up Gouraud shading and permitting any intensity of the original color to be assigned to the vertex color. (See "Color Matching" on page 414.)

If you do not want to save vertex normals with a database, clear the Save Vertex Normals option in the database Header Attributes window.

Modifying Shade Characteristics

Once **Calculate Shading** has been performed, you can add drama or special effects to your shaded models by modifying the shade attributes of certain faces, adding vertex colors, or modifying vertex normals.

Shade Attributes

The Shade attribute of each face determines lighting on a face-by-face basis. The choices are: **Flat**, **Lit**, **Gouraud**, and **Lit Gouraud** (see "Calculate Shading Parameters" on page 622 for descriptions of the shading modes). If **Lit Gouraud** is selected, faces are drawn in realtime using both vertex colors and vertex normals (provided both have been calculated). This attribute is sometimes used to produce special shading effects when exported to other applications.

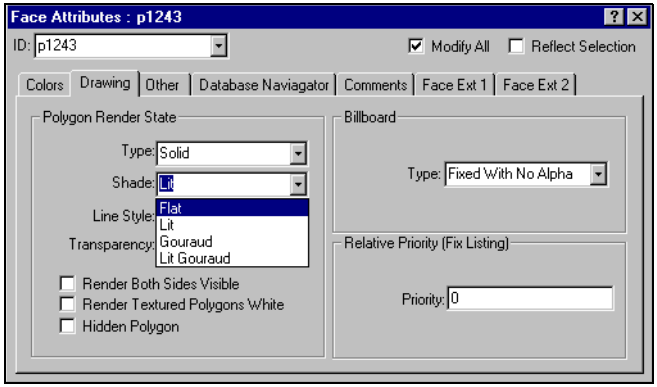
To change a face's shade attribute:



Step 1. Set the modeling mode to Face.

Step 2. Select the face or faces.

Step 3. Choose **Attributes/Modify Attribute**. The *Face Attributes* window opens.



Step 4. Choose a shading mode from the **Shade** list box. If multiple faces have been selected, set **Modify All**.

Vertex Normals

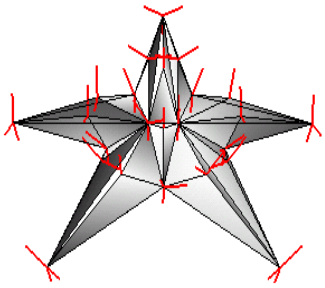
When **Calculate Shading** is performed, vertex normals extend out from a vertex at a 90 degree angle to the face. You can change the angle of the vertex normals to create different lighting effects.

Viewing Vertex Normals



To view vertex normals when the Graphics view is in wireframe or wireframe-solid mode, click **Draw Vertex Normals** in the View panel.

To change the display length of normals (in database units), hold down Ctrl and click **Draw Vertex Normals**.



Displaying Vertex Normals

Modify Vertex Normals Tool



The **Modify Vertex Normals** tool in the **Modify Vertex** toolbox changes the vector assigned to one or more selected vertices or light points (see "Shading the Database" on page 622 and "Creating Light Points or Strings" on page 631). This allows you to rotate the vertex normals that control dynamic shading and light point direction.

The **Modify Vertex Normals** tool window displays a graphical rotation guide that shows the orientation of the first vector on the select list. You can change the vector by dragging the guide with the left mouse button in the Graphics view, or by turning the dials on the clocks in the *Modify Vertex Normals* dialog box.

To Modify Shading on a Face



Step 1. Choose **Draw Vertex Normals** in the View panel.

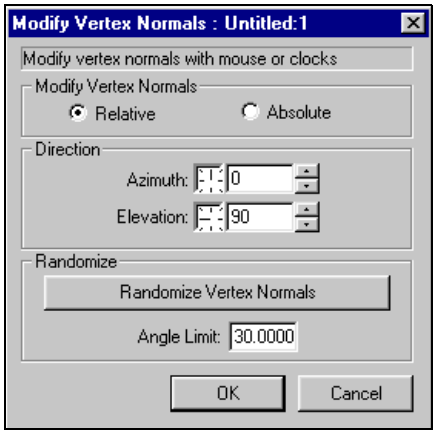
Step 2. If you have not already done so, calculate shading to assign normals (see "Shading the Database" on page 622 for instructions).

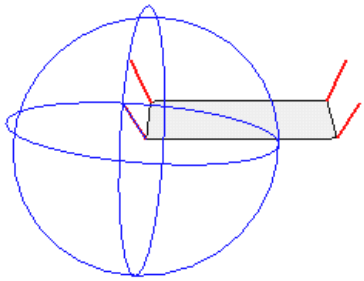


Step 3. Set the modeling mode to **Face**.

Step 4. Select the face and click **Modify Vertex Normals**.

Step 5. Use the dials in the *Modify Vertex Normals* dialog box, or drag the rotation guide in the Graphics view to change the direction of the normals associated with the selected face. Click **OK**.





To Modify Light Point Direction or Individual Vertex Normals



Step 1. Choose **Draw Vertex Normals** in the View panel.

Step 2. If you have not already done so, calculate shading to assign normals (see "Shading the Database" on page 622 for instructions).

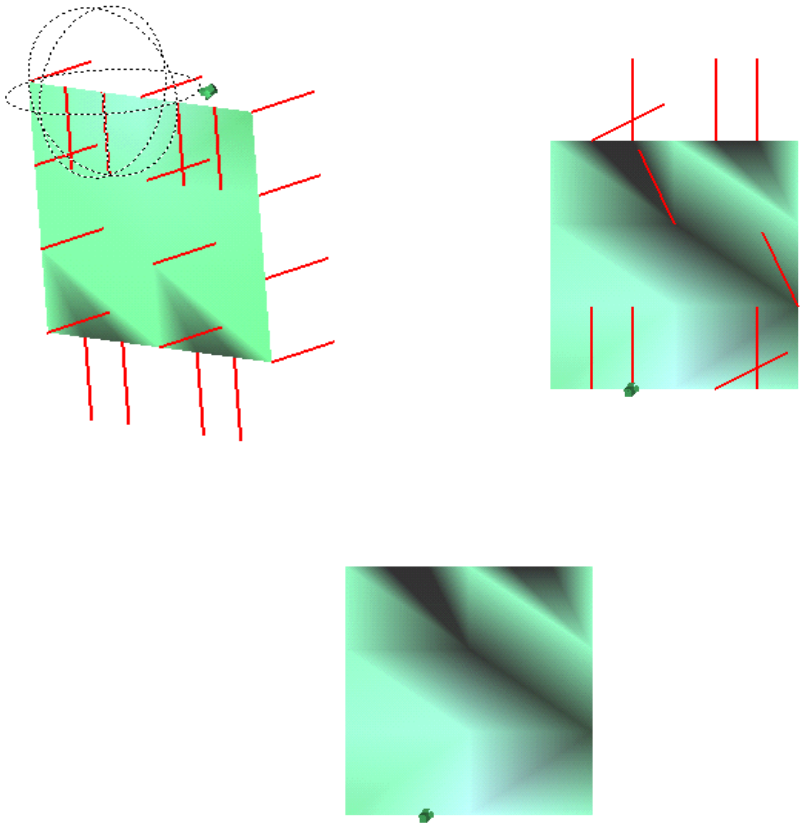


Step 3. Set the modeling mode to **Vertex**.

Step 4. Select the vertices or light points to be modified and click **Modify Vertex Normals**.



Step 5. Use the dials in the *Modify Vertex Normals* dialog box, or drag the rotation guide in the Graphics view to change the direction of the normals. Click **OK**.



Modify Vertex Normals Parameters



Modify Vertex Normals in the **Modify Vertex** toolbox has the following parameters:

Make Vectors Equal

Assigns the vector contained in the rotation guide to all selected vertices. When this option is set, all selected vertices receive the same vertex normal.

Change Vector Delta

Applies the change you define with the rotation guide to each selected vertex. When this option is set, the relative distance between vertex normals remains constant.

Azimuth

Rotates the vector through the great circle defined by the z axis.

Elevation

Pitches the vector around the current position of the x axis, as defined by the **Azimuth** control.

Randomize Vectors

Alters each direction vector by a random amount that is no greater than the value **Angle Limit**. Randomizing the vectors of a surface gives it a rougher (shaded) appearance.

Angle Limit

Defines the amount of variation that can be introduced within a set of vectors that is being randomized.

17 Light Points and Light Strings

The **Light Point** tool creates sets of non-radiant light points in the database. *Light points* are individual vertices that represent distantly-viewed points of light such as stars, runway lights, or city lights. *Light strings* are light points that are arranged along a predefined path.

You can choose the type of light points you want to create, and define their arrangement, color, and other properties at the time you create them. You can save a set of attributes as a Light Point palette entry to be applied to other light points in the OpenFlight file. You can save the palette entries in a separate file to share with other databases or projects.

Light points can be automatically created when generating terrain with features. (see“Light Point/String Attributes” on page 634).

Like all vertices, light points can be adjusted and moved independently after you create them. Unlike light sources, light points do not illuminate the visual scene or affect shading.

Creating Light Points or Strings

The **Create Light Point** tool opens a dialog box containing many parameters that define the light point characteristics, including whether the light points will be arranged as points or strings. Most of the characteristics will be visible only in the realtime system. Once you define how you want the light points to look and behave in realtime, you click in the Graphics view to place the light points. All light points created before you close the tool are placed in a light point node in the database hierarchy.

To create light points or light strings:



Step 1. Set a group node, such as a **Light Point System** node, or object node in the database as a parent. Set the **Light Point System** group node if you want to be able to control light points at runtime. See “Light Point System Mode Tool” on page 660 for more information.



Step 2. If you want to place light points randomly, turn off **Grid Snap** in the View panel.



Step 3. Open the **Create** toolbox.

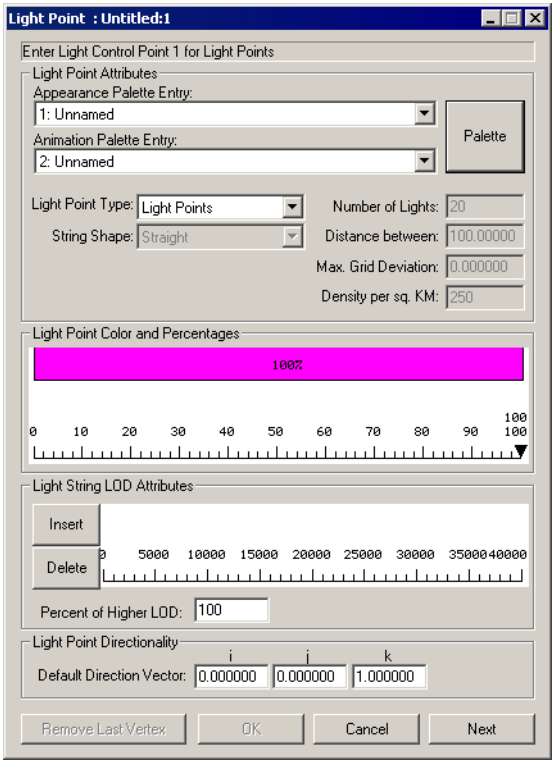


Step 4. Click **Create Light Point**. The *Light Point* dialog box opens.

Step 5. This dialog box has many attributes that can be applied to light points. At the top of the dialog box, you can specify a Light Point Appearance palette entry or Animation Palette entry to use, or you can open the *Light Point Palette* dialog box to create new palette entries. For help, see the following topics:

"Light Point Attributes" on page 634

"Light Point Palette" on page 640



Step 6. Choose the appropriate **Light Point Type**. For example, choose **Light Points** to create and place individual light points, choose **String w/Delta** to create strings of evenly-placed light points, or choose **Random Rectangle** to create a rectangular area of randomly-scattered light points. See "Light Point Type" on page 635 for descriptions of all available light point types.

If the **Light Point Type** is **String w/Delta**, **String w/Count** or **String w/Delta and Count**, choose a **String Shape**. For these light point types, the appearance of the string is controlled by both settings. See "String Shape" on page 636 for more information.

Step 7. Set the other parameters. For help, see the following topics:

"Light Point Attributes" on page 634

"Light Point Color and Percentages Parameters" on page 637

"Light Point LOD Attributes" on page 638

"Light Point Directionality" on page 639

Step 8. Click in the Graphics view to create individual light points, or to set the control points for the light points or strings you want to create.

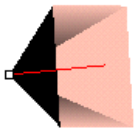
Step 9. Click **OK** in the *Light Point* dialog box, or **Next** to continue creating light strings.



Step 10. Choose **View/Sky Color** and change the sky color to make the light strings visible, if necessary. You can also set **Draw Lights Large** in the View panel.

Step 11. Set **Draw Vertex Normals** in the View panel to view the angular cones that represent uni- and bi-directional light points in Creator.

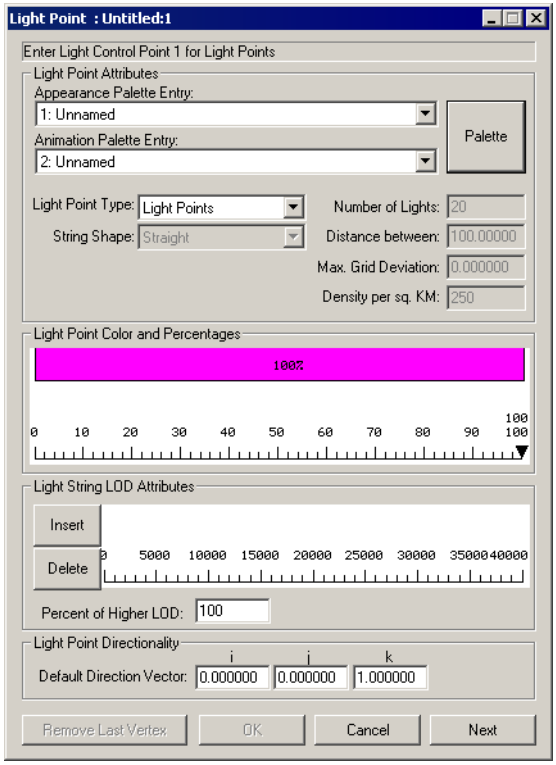




Light Point/String Attributes

Light Point attributes define the type of lights to be created. The *Light Point* dialog box, which opens when you click the **Create Light Point** tool in the **Create** toolbox, contains attributes that control the appearance and animation characteristics, as well as other information such as placement, normals, and colors.

Light Point Dialog Box



Light Point Attributes

Light Point attributes allow you to assign previously defined Light Point Appearance and Animation entries, or you can click **Palette** to create new palette entries using the *Light Point Palette* dialog box.

Appearance Palette Entry

This list allows you to assign an Appearance palette entry to the light point or light string.

Animation Palette Entry

This list allows you to assign an Animation palette entry to the light point or light string.

Light Point Type

Defines the type of light points to be created; whether they are individually placed, a set of randomly-placed points in a defined area, or a string of light points.

- **Light Points** – Places one light point at each coordinate entered with the mouse.
- **String w/Delta** – Creates a line of light points between each pair of control points. The first light is placed at the first control point. Additional lights are placed along the line between the first and second control points at constant intervals determined by the **Distance Between** parameter. When you select **String w/Delta**, you also select a **String Shape** to control whether the light string shape is straight, curved, or a spiral.
- **String w/Count** – Creates a line of light points between the control points. Straight strings have two control points; curved strings have three control points; and spiral strings have four control points. One light is placed at each control point, and the specified **Number of Lights** is distributed evenly along a line between the control points. When you select **String w/Count**, you also select a **String Shape** to control whether the light string shape is straight, curved, or a spiral.
- **String w/Delta and Count** – Places one light point at the first control point and inserts additional lights along the line determined by the first and second control points. **Distance Between** controls the length of the interval between light points. **Number of Lights** controls the number of additional lights in the string. When you select **String w/Delta and Count**, you also select a **String Shape** to control whether the light string shape is straight, curved, or a spiral.
- **Random Rectangle** – Randomly distributes the specified **Number of Lights** within the rectangular area defined by the two control points placed at opposite corners. To produce an elevation, offset the tracking plane before entering the second point.

- **Random Polygon** – Randomly distributes light points at a constant density within an irregular outline defined by the control points. The density is specified by **Density per sq. KM**.
- **Random Circle** – Randomly distributes the specified **Number of Lights** within the circular area defined by two control points - the first at the center and second along the edge. To produce an elevation, offset the tracking plane before entering the second point.
- **Light Grid** – Randomly distributes light points along the tracking plane grid within an irregular outline of control points to simulate a grid of city lights. Density is determined by the **Distance Between**. Randomization is determined by the **Max Grid Deviation**.

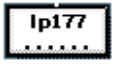
  Using **Light Grid**, it is possible to create a large number of light points which may not be necessary for the intended use of your database.

String Shape

Defines the type of light point string to be created when the selected light type is a string. You can only select from this list when **String Type** is **String w/Delta**, **String w/ Count** or **String w/Delta and Count**.

- **Straight** – Creates a straight light point string between a pair of control points. A translate matrix is added to the light point node.
- **Curved** – Creates a curved light string between three control points. The first two control points you enter define the endpoints, and the third point defines the edge of a circle. The light string will be placed along an arc of this circle. A rotation matrix is added to the light point node.
- **Spiral** – Creates a flat spiral light string between four control points. The first three control points define the center of the circle (as in Curved). The distance between the third and fourth control point determines the scaling factor. A rotate-on-scale matrix is added to the light point node.

  Note: String-type light point nodes have a dotted line symbol indicating that there is a single vertex with matrix and replication count.



Number of Lights

Controls the number of light points created when **Light Type** is String w/Count, String w/Delta and Count, Random Rectangle, or Random Circle.

Distance Between

Controls the spacing between the light points attached to each face when **Light Type** is String w/Delta, String w/Delta and Count, or Light Grid.

Max. Grid Deviation

Controls the limits for randomizing light points used when **Light Type** is Light Grid. This defines the largest allowable distance from the trackplane grid.

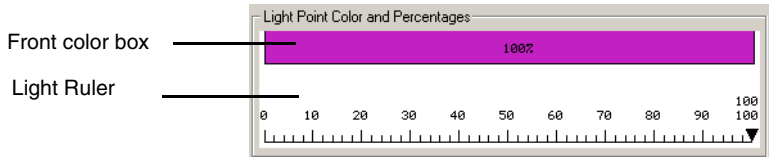
Density per sq. KM

Controls the density of light points used when **Light Type** is Random Polygon.

Light Point Color and Percentages Parameters

Color parameters assign front-facing color(s) to light points when they are created. In Creator, color is displayed for uni-directional and bi-directional light points in an angular cone only when **Draw Vertex Normal** is on. To select a front-facing color, double-click the Front color box to open the Color palette. Select a Primary Color and click the Front color box once.

Light Point Dialog Box Color Parameters



Light Ruler

Determines the number of front-facing colors and the distribution of color (as a percentage) in a random distribution of light points (non String-type). Up to eight colors can be defined. For String-type and individual light points that use only one

vertex, only the color band to the far left is used when multiple colors are defined on the Light Ruler.

Set the color distribution by dragging the triangular sliders to the left to define the desired percentages.

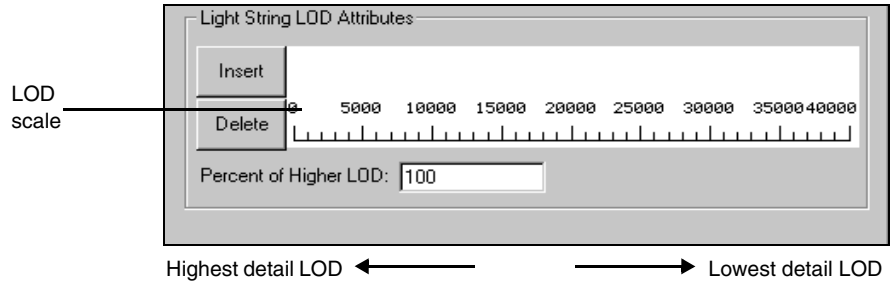
Light color can be added to the ruler. Click once to apply the current color, or double-click to open the Color palette window, select a new **Primary Color**, and click to apply it.

Light Point LOD Attributes

When you assign LODs to light points, a percentage of light points are turned off when viewed at specified distances. Multiple LODs reduce the effect of light points growing in intensity when they appear closer together as they recede from view.

The lower LODs have fewer light points than the higher LODs. For light strings, the number of light points in the lower LODs are reduced by half. Every other light point disappears from view.

Light Point Dialog Box LOD Parameters



Note: LODs are not allowed as subfaces. If an LOD is created with a parent set to face, the highest detail LOD is used for that face.

LOD Scale

Sets the number of LODs and the switching distance of each LOD. Each interval between triangular sliders (or between a slider and the end of the scale) corresponds to

one LOD, with the lowest detail LOD on the right. Each triangular slider specifies the switch out attribute of the LOD to its right and the switch in attribute of the LOD to its left.

The lowest detail LOD switches in at the right most triangular slider (by default); the highest detail LOD switches out at 0.

To change a switching distance, drag a slider left or right, or select and edit the switching distance text displayed above the slider. Dragging a slider past the right end of the scale increases the range of available switching distances displayed.

To insert an LOD, click **Insert**. If a slider is selected when this button is clicked, the LOD on its left is split in two. If nothing is selected, the highest LOD is split in two.

To delete an LOD, select a slider and click **Delete**. If nothing is selected when this button is clicked, the highest LOD is deleted.

If there are no sliders on the ruler, LOD nodes will not be created, and the lights will always be on.

Percent of Higher LOD

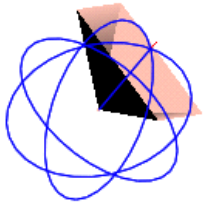
Sets the percentage of light points to be retained from the next higher LOD. The next lower LOD retains this percentage and the rest are randomly removed. The percentage value for light strings is fixed at 50% because strings are represented by replicating a single vertex.

Light Point Directionality

The direction vector is assigned to the light point vertex and defines the viewing direction.

Default Direction Vector

Defines the i, j, k vectors assigned to the light point vertices. The vector direction defaults to positive z (i, j, k = 0, 0, 1). The direction vector becomes the vertex normal.

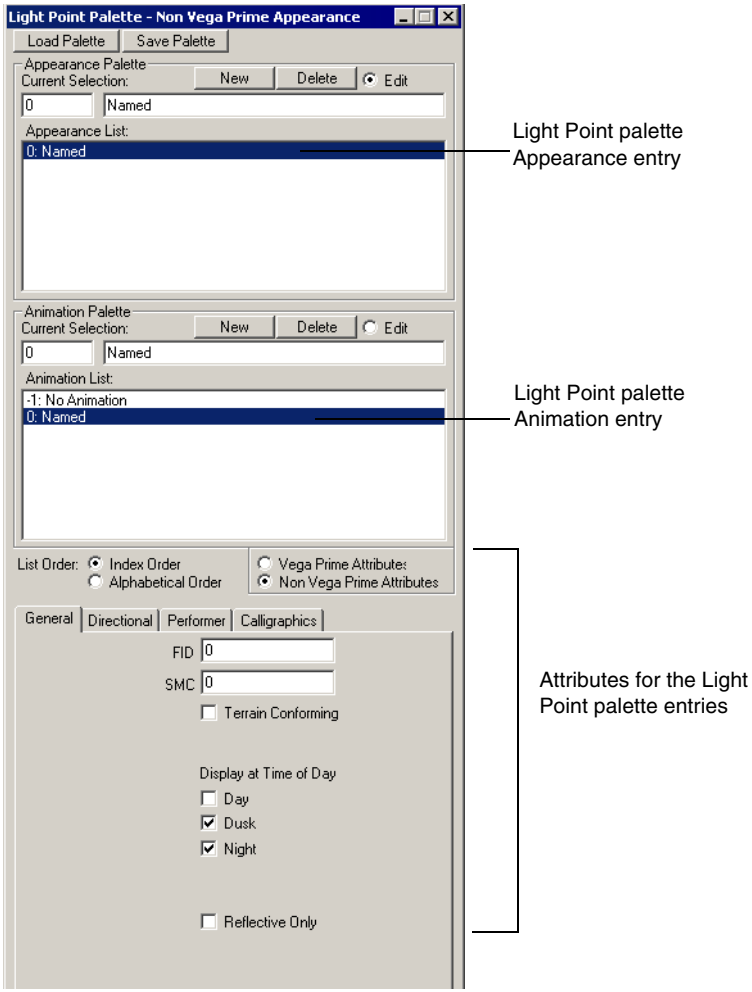


Light Point Palette

Light Point palette entries are sets of attributes that are saved with the OpenFlight file and used whenever they are needed for your light points or light strings. You create your light point attributes using the *Light Point Palette* dialog box.

You define Vega Prime or non-Vega Prime appearance attributes and animation attributes. If you will be using both Performer and Vega Prime realtime systems, you

can define Vega Prime *and* non-Vega Prime attributes for an entry. When you are done creating all of your entries, you can then save them in a palette file.



Using the Light Point Palette

The *Light Point Palette* dialog box allows you to create and define Light Point palettes that contain multiple appearance and animation attributes. You can assign appearance and animation entries to one or more light point or light point strings in the *Light Point* dialog box.

Light Point palette attributes are saved in the OpenFlight file, but you can also save Light Point palettes in ASCII file format. The default file is named mgprop.dat. The palette file can be loaded and used in other OpenFlight files.

You can save, load, and modify pre-existing Light Point palettes from within the *Light Point Palette* dialog box. Modifications to the parameters are automatically applied to all light points associated with the palette.

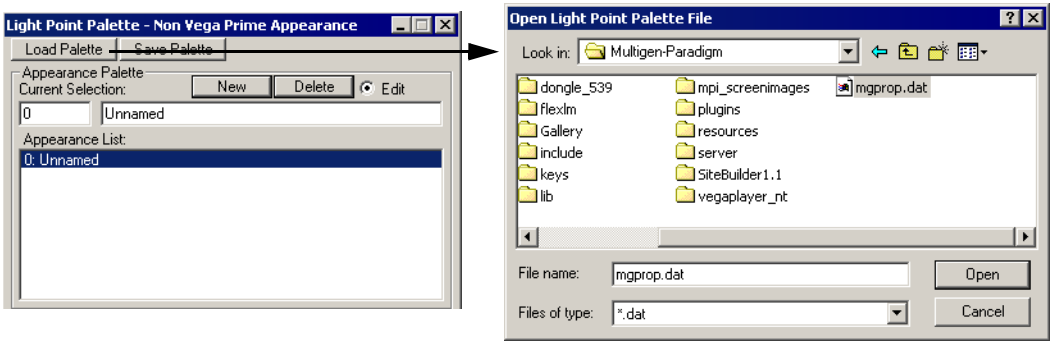
Note: If you delete a palette entry in the *Light Point Palette* dialog box, question marks will appear in place of the associated light points in the Graphics view.

To Load a Light Point Palette File

The following steps load a Light Point palette from a palette file in the *Light Point Palette* dialog box.

Step 1. Choose **Palettes/Light Point** to open the *Light Point Palette* dialog box. You can also click **Palette** in the *Light Point* dialog box.

Step 2. Click **Load Palette**. The *Open Light Point Palette File* dialog box displays.



Step 3. Select the name of the file to load and click **Open**.

Step 4. The Light Point palette entries are now available in the *Light Point Palette* dialog box.

To Create a New Light Point Palette Entry

The following steps create a new Light Point palette entry for the currently loaded Light Point palette file.

Step 1. Choose **Palettes/Light Point** to open the *Light Point Palette* dialog box. You can also click **Palette** in the *Light Point* dialog box.

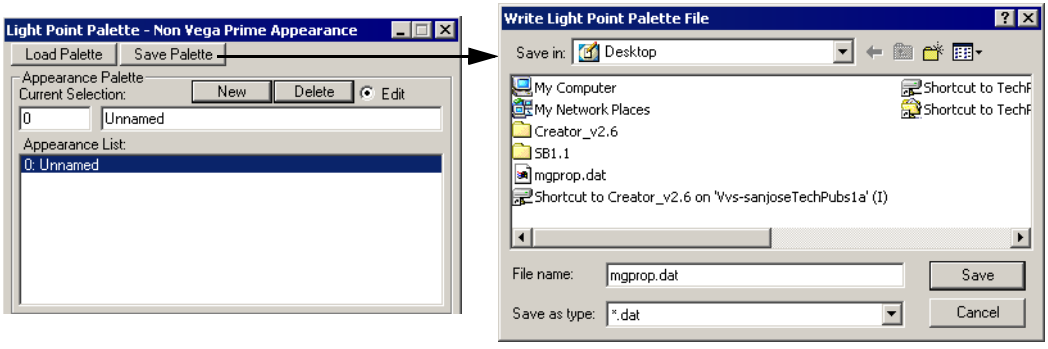
Step 2. Click **New** for the Appearance palette or the Animation palette. A new entry appears in the list.

Step 3. Change the default name (**Unnamed**) for the new Light Point palette entry in **Current Selection**. You can also change the default entry number that is automatically generated.

Note: You can select **Index List Order** to display the entries sequentially by entry number, or **Alphabetical List Order** to display the entries alphabetically.

Step 4. Select either **Vega Prime Attributes** or **Non Vega Prime Attributes** to specify palette attributes. You can also specify both Vega Prime and non-Vega Prime attributes for an entry for use on both realtime systems.

Step 5. After specifying all the attributes, click **Save Palette** to update the current palette file, or to save the palette entry in a new palette file in the *Write Light Point Palette File* dialog box.



To Edit a Light Point Palette Entry

The following steps edit a Light Point Appearance or Animation palette entry.

Step 1. Choose **Palettes/Light Point** to open the *Light Point Palette* dialog box. You can also click **Palette** in the *Light Point* dialog box, or in the *Light Point Attributes* dialog box.

Step 2. Select **Edit** for the Appearance palette or the Animation palette. The attributes will display for the selected entry.

Step 3. Edit the attributes for the entry.

Step 4. After editing the attributes, click **Save Palette** to update the current palette file, or to save the palette entry in a new palette file in the *Write Light Point Palette File* dialog box.

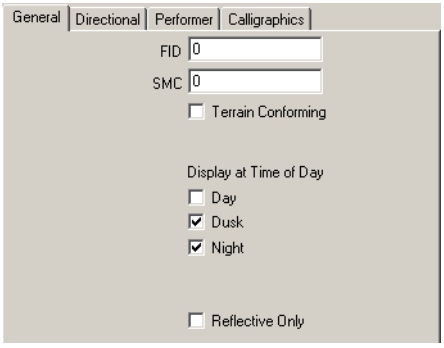
Light Point Palette Attributes

You can define appearance and animation attributes for both Vega Prime and non-Vega Prime realtime systems. Appearance attributes include the type of light points to create, the method of display, terrain settings, color, display, direction, and SGI Performer information. Animation attributes define animation characteristics, such as the type, period, and animation vector, for example.

Light Point palette attributes are saved in the OpenFlight file. The attributes can also be saved in palette files that can be used with other OpenFlight files.

General Attributes

Click an entry in the Appearance list, and then select **Non Vega Prime Attributes** to display these attributes in the General panel.



FID

Feature Identifier, used with the **SMC**, is defined by the user to tag data for realtime control. The FID is the Feature Identifier of the projected feature on terrain.

SMC

Surface Material Code, which is used with the FID , is defined by the user to tag data for realtime control. The SMC is the Surface Material Code of the projected feature on terrain.

Terrain Conforming

Drapes all light points on the surface of a terrain. This setting creates a draped Adaptive attribute attached to the light point node that allows the light points to conform to the CAT terrain. Light points move up and down with the CAT terrain as it morphs between levels of detail.

This feature can only be viewed with Performer.

Display at Time of Day

Specifies the time(s) of day at which the light points are displayed.

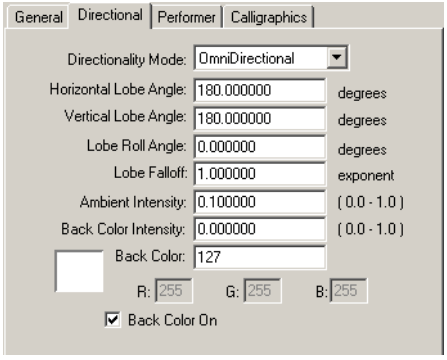
Reflective Only

Modifies the intensity of the light points based on the vertex normal and eyepoint position.

Directional Attributes

Direction attributes define how light points are displayed based on viewing direction.

Click an entry in the Appearance list, and then select **Non Vega Prime Attributes** to display these attributes in the Directional panel.



Directionality Mode

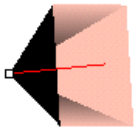
Controls the directions from which light points are visible. Choose one of the following:

OmniDirectional

Light points can be viewed from all directions. Displays the light point's front color, which is defined by the vertex color.

UniDirectional

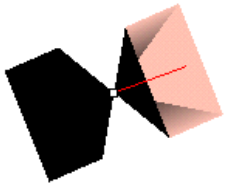
Light points can be viewed only from the front. Displays the light point's front color, which is defined by the vertex color.



BiDirectional

Light points can be viewed from both front and back. Displays the front-facing color on the front side of the **Direction Vector/Light Lobe**, displays the **Back Color** when viewed from the back side, and displays ambient intensity outside the lobes.

The front color is defined by the vertex color. The back color for non-Vega Prime light points is defined in the Directional panel in the *Light Point Palette* dialog box. The back color for Vega Prime light points is defined in the *Vertex Attributes* dialog box.



Horizontal Lobe Angle

Specifies the angle that defines the horizontal sweep of the light lobe.

Vertical Lobe Angle

Specifies the angle that defines the vertical sweep of the light lobe.

Lobe Roll Angle

Defines the rotation of the lobe shape (defined by **Horizontal** and **Vertical Lobe Angles**) about the vertex normal.

Lobe Falloff

Defines the change in intensity from the center of the light lobe to the edge. Recommended values are greater than or equal to 0. A value of 1 creates a linear falloff.

Ambient Intensity

Defines the intensity of the light outside of the light lobe. A value of 0 indicates the light is off and 1 indicates the light is at full intensity.

Back Color Intensity

Defines the intensity of the back color. A value of 0 indicates the light is off and 1 indicates the light is at full intensity.

Back Color

Determines the back color of the current set of Bidirectional light points. To select the back color, double-click the **Back** color box to open the Color palette. Select a new primary color, and click the **Back** color box once.

Back Color On

Enables the specified back color to be used instead of the front color. If this is clear, the the front color is used for the back color.

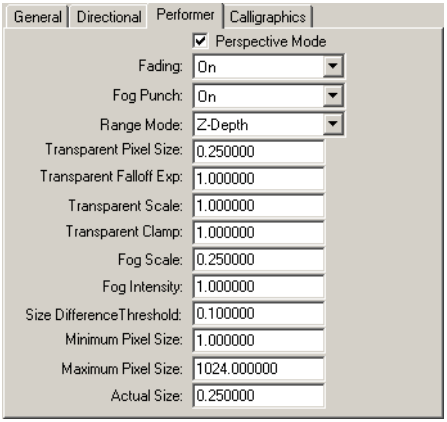
This effect can only be viewed in the runtime system.

SGI Performer Attributes

The SGI Performer attributes are used specifically by the OpenFlight Performer loader and Iris Performer. They are defined in the SGI Performer rendering system, but some may also be used in other runtime systems for light point display.

> **Note:** The effect of these attributes may not be visible in Creator.

Click an entry in the Appearance list, and then select **Non Vega Prime Attributes** to display these attributes in the Performer panel.



Perspective Mode

Uses perspective fading to attenuate distant light point intensity.

Fading

Uses transparency to attenuate distant light point intensity.

Fog Punch

Allows light points to shine through fog as a light would.

Range Mode

Selects the method for measuring the distance to the eyepoint.

- **Z-Depth** – Uses z depth to calculate the distance to the eyepoint. Z-depth range mode may be faster on SGI systems.
- **Slant Range** – Uses the slant range to calculate the distance to the eyepoint. Slant range is the distance from the eyepoint to the center of the level of detail.

Transparent Pixel Size

Sets the threshold diameter for computed light points at which the alpha component is decreased and the light point becomes more transparent.

Transparent Falloff Exp.

An exponential factor for light point Fading. Values should be greater than or equal to 0. Values greater than 0 and less than 1 flatten the falloff curve, and values greater than 1 sharpen it.

Transparent Scale

A scale factor for light point Fading. Values greater than 0 and less than 1 decreases the falloff rate, and values greater than 1 increase it.

Transparent Clamp

Defines the minimum Fading multiplier value.

Fog Scale

Scale factor that determines the degree to which light points can be viewed through fog. Values greater than 0 and less than 1 cause the light points to punch through fog more than other non-emissive geometry.

Fog Intensity

A factor used to increase or decrease light point alpha values.

Size Difference Threshold

Sets the threshold size at which the light point OpenGL point size is changed to optimize the rendering in OpenGL.

Minimum Pixel Size

Sets the minimum diameter of light points in pixels.

Maximum Pixel Size

Sets the maximum diameter of light points in pixels.

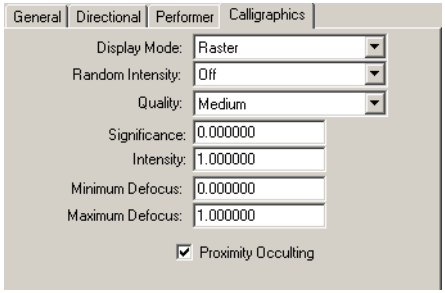
Actual Size

Sets the light point diameter, in database units.

Calligraphic Display Attributes

Calligraphic Display attributes describe controls commonly used by calligraphic display generators. (Requires SGI Performer version 2.2). Unlike Raster displays, Calligraphic displays can render light points at different qualities, depending on the amount of time spent rendering each point.

Click an entry in the Appearance list, and then select **Non Vega Prime Attributes** to display these attributes in the Calligraphics panel.



Display Mode

Display mode specifies the method used to display light points.

- **Raster** – Specifies that light points are to be drawn for use with a raster display. Raster information is organized into a regularly-spaced two-dimensional grid or bitmap.
- **Calligraphic** – Specifies that lights points are to be drawn for a calligraphic display, which produces images by creating the points directly. Light points are brighter on a calligraphic display than on a raster display.
- **Raster or Calligraphic** – Specifies that light points are to be drawn in calligraphic mode if the rendering load allows, otherwise they are to be drawn in raster mode.

Random Intensity Off / Low / Medium / High

Randomizes the light point intensity when the light points are loaded. Calligraphic generators may use several levels of randomization to break up light point uniformity.

Quality

Sets the amount of time the calligraphic gun is turned off between drawing light points, enabling it to seek the next position more accurately (high quality = long period of time, low quality = short period of time).

Significance

A number that assigns relative importance to light points, enabling the realtime software to decide which lights should become raster lights as the rendering load increases (used with Raster or Calligraphic mode).

Intensity

A scale factor that determines light point intensity.

Minimum Defocus

The minimum value for the calligraphic beam to defocus the light points. The smaller the value, the smaller the diameter of the calligraphic beam and the more defocused the light point becomes.

Maximum Defocus

The maximum value for the calligraphic beam to defocus the light points. Controls the size to which light points can grow in perspective as they approach the eyepoint. Value should be greater than or equal to 0 and less than 1.

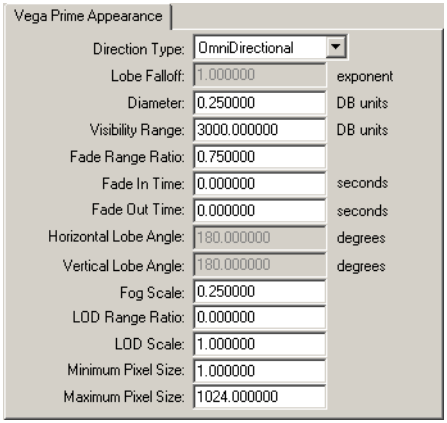
Proximity Occulting

Reduces the intensity of light points when many light points must be rendered in the same place (runway lights viewed from a distance, for example), which can damage some displays. Also called *debunching* or *debauching* on other rendering systems. This option is set by default.

Vega Prime Appearance Attributes

These Appearance attributes are used by the Vega Prime realtime system.

Click an entry in the Appearance list, and then select **Vega Prime Attributes** to display these attributes in the Vega Prime Appearance panel.



Direction Type

The Direction type defines how light points are displayed based on viewing direction. Choose either **BiDirectional**, **OmniDirectional**, or **UniDirectional**. For more information about these Direction types, see "Directional Attributes" on page 646.

Lobe Falloff

Defines the change in intensity from the center of the light lobe to the edge. Recommended values are greater than or equal to 0. A value of 1 creates a linear falloff.

Diameter

Sets the light point diameter, in database units.

Visibility Range

The distance from the eyepoint that the light point is visible, in database units.

Fade Range Ratio

The percentage of the total range (0-1) at which light points start to fade and become transparent.

Fade In Time

The time in seconds that the light point will fade in.

Fade Out Time

The time in seconds that the light point will fade out.

Horizontal Lobe Angle

Specifies the angle that defines the horizontal sweep of the light lobe.

Vertical Lobe Angle

Specifies the angle that defines the vertical sweep of the light lobe.

Fog Scale

Scale factor that determines the degree to which light points can be viewed through fog. Values greater than 0 and less than 1 cause the light points to punch through fog more than other non-emissive geometry.

LOD Range Ratio

The percentage of the total range (0-1) at which light point LODs are active.

LOD Scale

Sets the number of LODs.

Minimum Pixel Size

Sets the minimum diameter of light points in pixels.

Maximum Pixel Size

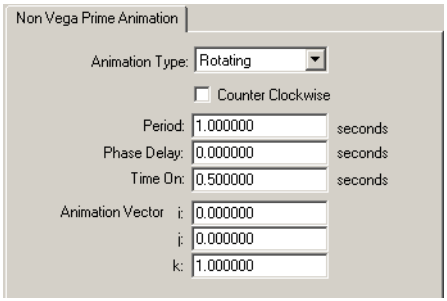
Sets the maximum diameter of light points in pixels.

Animation Attributes

Animation attributes randomize and synchronize the animation of multiple light points for non-Vega Prime realtime systems.

Click an entry in the Animation list, and then select **Non Vega Prime Attributes** to display these attributes in the Non Vega Prime Animation panel.

Non Vega Prime Animation Attributes



Animation Type

Specifies one of three animation types: **No Animation**, **Flashing**, or **Rotating**.

- **No Animation** – Leaves the lights set to shine constantly.
- **Flashing** – Sets the light points to flash periodically based on values for Period, Phase Delay, and Time On.
- **Rotating** – Sets the light points to rotate periodically based on Period and Phase Delay.

Counter Clockwise

Sets the light point's direction of rotation to counter-clockwise.

Period

Specifies the time for a full cycle of animation for **Flashing** lights or the time for one revolution of **Rotating** directional lights.

Phase Delay

Specifies the time from start of **Period** to beginning of **Time On** when used with Flashing lights or the delay before starting a **Rotating** light point.

Time On

Specifies the length of time that a Flashing light point is displayed.

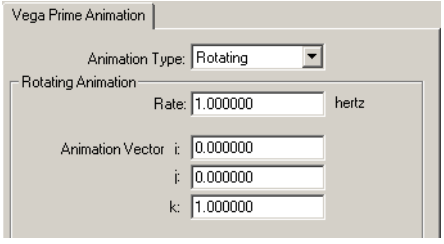
Animation Vector

Sets the axis around which the light points will revolve. This information is exported to the realtime system and cannot be viewed within Creator.

Vega Prime Animation Attributes

Vega Prime Animation attributes randomize and synchronize the animation of multiple light points for Vega Prime realtime systems.

Vega Prime Animation Attributes



Click an entry in the Animation list, and then select **Vega Prime Attributes** to display these attributes in the Vega Prime Animation panel.

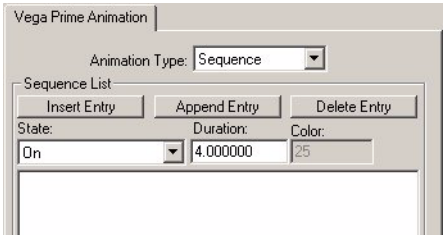
Animation Type

Specifies one of four animation types: **Sequence**, **Rotating**, **Strobe,** or **Morse Code**.

- **Sequence** – Sets the light points to flash on and off, and change colors based on a specified sequence of entries.
- **Rotating** – Sets the light points to rotate periodically.
- **Strobe** – Animates a light string to simulate an airport runway approach light system in which all light points flash on and off in sequence based on a rate. The rate is the frequency for all the lights to flash once.
- **Morse Code** – Sets a morse code string as a light point signal based on Timing, Word Rate, and Character Rate values.

Creating a Sequence Animation

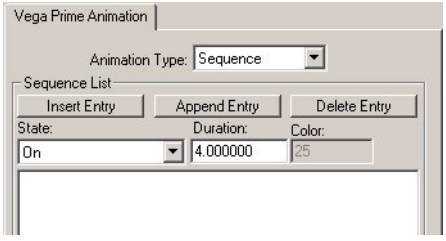
A sequence animation renders light points according to attributes that you define in a sequential list of entries. You specify an entry for each on, off, or color change state.



To Create a Sequence Animation

The following steps create a sequence animation.

Step 1. In the Vega Prime Animation panel, choose the **Sequence** Animation Type.



Step 2. Click **Insert Entry** to insert entries at the bottom of the **Sequence List**.

Step 3. Choose either an **On**, **Off**, or **Color Change** state.

Step 4. Enter the duration in seconds for each state.

Step 5. If you chose the **Color Change** state, enter the color index value. You can find the index value for a color in the **Color Palette**.

Step 6. Click **Append Entry** to append entries after the selected entry.

Step 7. Click **Delete Entry** to delete a selected entry.

Step 8. Click **Save Palette** to save all of your sequence of animation entries in a palette file.

Sequence Animation Attributes

When you choose the **Sequence** Animation Type in the *Light Point Palette* dialog box, these attributes appear for creating a list of entries.

State

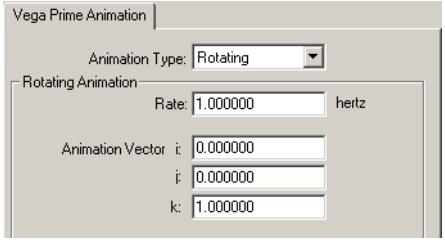
Renders a light point on, off, or a specified color.

Duration

Specifies the duration of time of the animation state, in seconds.

Rotating Animation Attributes

When you choose the **Rotating** Animation Type in the *Light Point Palette* dialog box, these attributes appear.



Rate

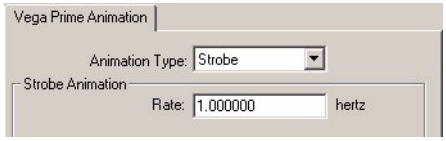
Specifies the rate that the light points will rotate, in hertz.

Animation Vector

Sets the axis around which the light points will revolve. This information is exported to the realtime system and cannot be viewed within Creator.

Strobe Animation Attributes

When you choose the **Strobe** Animation Type in the *Light Point Palette* dialog box, these attributes appear.

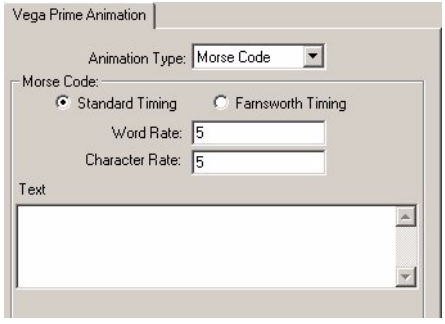


Rate

Specifies the rate that all the light points will display once, in hertz.

Morse Code Animation Attributes

When you choose the **Morse Code** Animation Type in the *Light Point Palette* dialog box, these attributes appear.



Standard Timing

Standard timing for the word rate. Standard timing is slower than Farnsworth timing.

Timing methods specify the duration of the dot, dash, intra-character, inter-character, and inter-word spacing in a morse code string.

Farnsworth Timing

Uses fast characters and slow inter-character and inter-word spacing to maintain the word rate.

Timing methods specify the duration of the dot, dash, intra-character, inter-character, and inter-word spacing in a morse code string.

Word Rate

The rate that words display, in seconds.

Character Rate

The rate that characters display, in seconds. This attribute is only necessary for the Farnsworth timing method.

Text

The string of characters in the message that will display in morse code.

Light Point Tools

These tools are associated with light points: The **Create Light Point** tool, **Light Point Mode** and **Light Point System Mode** tools, and the **Draw Lights Large** and **Draw Vertex Normals** view controls.

Create Light Point Tool



You create light points using the **Create Light Point** tool in the **Create** toolbox. You can create light points using the Create Light Point tool regardless of the modeling mode (see "Creating Light Points or Strings" on page 631).

Light Point Mode Tool



Select **Light Point Mode** in the **Mode** toolbox to select light points for modifying, to display light point attributes, or when using the **Create** tool to create light point nodes (instead of using the **Create Light Point** tool).

Light Point System Mode Tool



Select **Light Point System Mode** in the **Mode** toolbox to select light points for modifying, to display light point system attributes, or when using the **Create** tool to create light

point nodes under the light point system parent node (instead of using the **Create Light Point** tool). Use the light point system node when you want to be able to control the light points during runtime. See "Modifying Light Point System Attributes" on page 662 for information about modifying a light point system's attributesl

Draw Lights Large Tool



To easily select light points for modifying, turn on **Draw Lights Large** in the View panel and then click the light point(s). This control enlarges each light point in the database.

Draw Vertex Normals Tool



The **Draw Vertex Normals** control in the View panel displays the angular cones that represent uni- and bi-directional light points in Creator.

Modifying Light Point Attributes

You can modify some light point attributes in the Display panel in the *Light Point Attributes* dialog box, including the string type if the selected light point node is a light string. (You cannot change a light point to a light string, or change a string to an individual light point.) You can also change the Light Point palette entry for a selected light point. When you modify light point attributes, the changes apply to all light points in the node.

The DB Navigator panel in the *Light Point Attributes* dialog box displays the light points as they appear in the Graphics view or the Hierarchy view. The Comments panel is used to add modeling comments to the database.

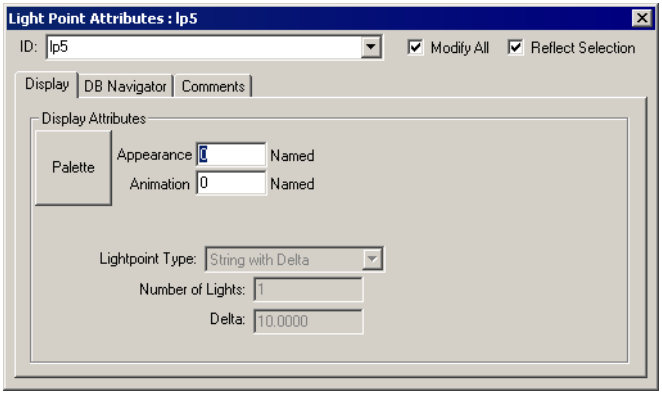


Step 1. Click **Draw Lights Large** in the View panel.

Step 2. In the Hierarchy view, select the light point node that you want to modify, or in the Graphics view, set the modeling mode to **Light Point** and select a light point, light string, or light polygon that you want to modify.



Step 3. Choose **Attributes/Modify Attributes** to open the *Light Point Attributes* dialog box.



Step 4. Click **Palette** to open the *Light Point Palette* dialog box.

Step 5. Change the light point attributes as desired, and then close the *Light Point Attributes* window.

Step 6. To change the light point's Appearance or Animation palette entry, change the palette entry numbers in the **Appearance** and **Animation** fields. When you press the Tab or Enter key, a message next to the number identifies the name of the entry, or indicates that the entry number does not exist.

There are certain characteristics that you cannot modify in the light point attributes. You must use the color tools to modify the vertex color. You can use the **Modify Vertex** tool to change the position of individual light points, to change the shape of a light string, or to move points in a light rectangle, light circle, or light grid. You can also use the **Translate** tool (in Vertex mode) to move all the light points in a node.

Modifying Light Point System Attributes

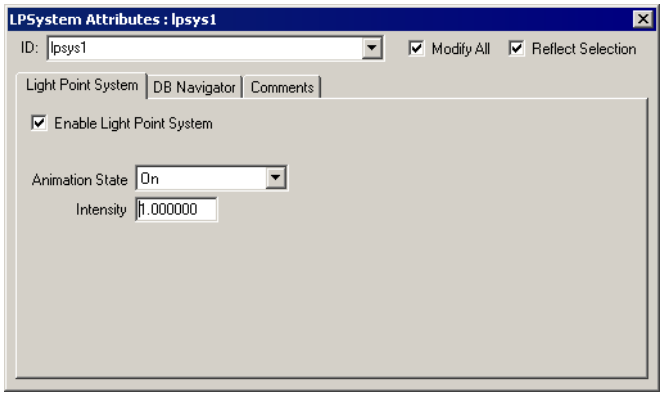
A light point system node is a group node that Vega Prime uses to control light points at runtime. You can modify a light point system's attributes in the Light Point System panel in the *LPSystem Attributes* dialog box. For the light point system, you can disable the light points and change their animation state and intensity.

Step 1. In the Hierarchy view, select the light point system node that you want to modify, or in the Graphics view, set the modeling mode to **Light Point**



System and select the light points, light strings, or light polygons that you want to modify.

Step 2. Choose **Attributes/Modify Attributes** to open the *Light Point System Attributes* dialog box.



Step 3. Clear **Enable Light Point System** to disable the light points in the light point system (the light points are enabled by default).

Step 4. You can choose to turn off the animation state, or change the animation state to **Random**.

Step 5. You can change the intensity to a value between 0 and 1.

Modifying Light Point Colors

Light Points can be visible from all directions (omnidirectional), from two directions only (bidirectional), or from a single direction (unidirectional), depending on the directionality parameter you set in the *Light Point Palette* window (see "Directional Attributes" on page 646). All types of light points display a Front Color; Bidirectional light points display a Front Color from one direction and a Back Color from the other direction. The following topics describe how to change these colors, and how to set the current Primary and Alternate colors to match light point colors.

To Change the Light Point Front Color

Step 1. In Vertex mode, select a light point or light point string. You can select Light Point mode to change all of the light points in the database.

Step 2. Open the Color palette and select a new color.

Step 3. Click **Insert Color** (Ctrl+R) in the **Properties** toolbox, or choose **Edit/Insert Color**.



To Change the Light Point Back Color

Step 1. In Vertex mode, select a light point or light point string.

Step 2. Open the Color palette, and select an **Alternate Color**.

Note: For non-Vega Prime light points, you can alternatively open the *Light Point Attributes* dialog box, and change the back color in the Light Point palette. See "Directional Attributes" on page 646 for more information. The back color for Vega Prime light points is stored in the vertex attributes.



Step 3. Hold down Alt and click **Insert Color** in the **Properties** toolbox.

Note: Using Alt and the **Insert Color** keyboard shortcut (Alt+Ctrlt+R) does not have any effect.

To Change Both Front and Back Light Point Colors

Step 1. In Vertex mode, select a light point or light point string.

Step 2. Open the Color palette and select an **Alternate Color**.



Step 3. Hold down Alt+Shift and click **Insert Color** in the **Properties** toolbox.

Note: Using Alt+Shift and the **Insert Color** keyboard shortcut (Alt+Shift+Ctrlt+R) does not have any effect.

Modifying Light Point/String Placement

You can use modeling tools to modify the placement of light points and strings. To change the position of individual points, whether they are random light points or points on a string, you can use the **Modify Vertex** tool. To move a group of light points, you use the **Translate** tool.

Light point polygons and grids (**Random Rectangle**, **Random Circle**, or **Light Grid**) behave differently from true light strings (**String w/Delta** or **String w/count**) when they are modified. You can move individual light points in light polygons and grids. When you move a light point in a light string, the other light points also move, because light strings are constructed with replicated light points using a matrix.

Modifying Individual Light Points, Polygons, and Grids

You can change the position of an individual light point, or of any light point that is part of a light point node that has a **String Type** of **Random Rectangle**, **Random Circle**, or **Light Grid**, with the **Modify Vertex** tool. These light points are each represented by a vertex under the Light Point node. To do this:

Step 1. In Vertex mode, select the light point to be modified.

Step 2. Choose **Modify Vertex** and click a new position for the light point.

You can also move an entire **Random Rectangle**, **Random Circle**, or **Light Grid**, with the **Modify Vertex** tool. To do this:



Step 1. Click **Draw Lights Large** in the View panel.

Step 2. In the Hierarchy view, select the light point node you want to modify.

You can alternatively set the modeling mode to **Light Point**, and select a light point, or select one of the light points in a string or light polygon you want to modify in the Graphics view.





Step 3. Open the **Maneuver** toolbox and click **Translate**. The *Translate* dialog box opens and a **From Point** reference point appears on the selected item. The **From Point** defines the origin of the translation. You can change it by selecting a different position anywhere in the Graphics view.

Note: The **From Point** does not have to be located on the selected item. The vector between **From Point** and **To Point** determines the distance and direction of the translation.

If you want to change the location of the **From Point**, click **Reset From Point** to change it back to its default position at the center of the selected light node.

Step 4. Select **To Point**, if it is not already selected, and then click the new position. The light points move so that the reference point selected in the previous step is placed at the new position. You can change the location by selecting a different position.

Step 5. Click **OK**.

Modifying Light Strings

You can change the shape of a light string by changing the position of individual vertices with the **Modify Vertex** tool. The control points are recalculated, and the light strings are regenerated. To do this:

Step 1. In Vertex mode, select a light point on the string.

Step 2. Click **Modify Vertex**, click a new position for the light point in the Graphics view, and then click **OK** in the *Modify Vertex* dialog box.

The effect of the change on the entire string depends upon the **String Type**, the **String Shape**, and the position of the modified light point in the light string:

- In a string with a **String Type** of **String w/Delta** a specified distance (the **Distance between** setting) is always preserved. This means that when moving a vertex changes the size of the string, the number of light points in the string will increase or decrease accordingly.
- In a string with a **String Type** of **String w/Count**, the number of lights in the string remains fixed. If modifying a vertex changes the length of the string, the distance between the light points increases or decreases to achieve even spacing along the new length.
- If the **String Type** is **String w/Delta and Count**, the number of lights in the string remains fixed, and the distance between the lights is preserved. Modifying the

position of a vertex may change the shape or position of the string, but it will not change the number of lights or the distance between them.

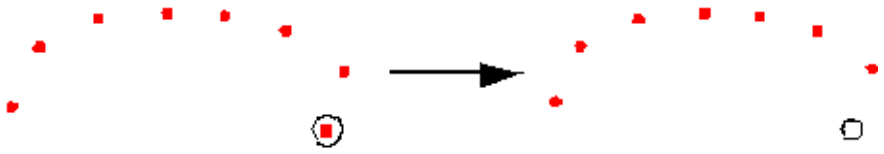
- If the **String Shape** is **Straight**, and you move a vertex other than the first one in the string, the string re-aligns along the vector defined by the first point in the string and the modified vertex. If you move the first point, the light string aligns itself along the vector defined by the modified point and the end point of the string. The number of lights in the modified string, and the distance between the lights depend on the **String Type** (see above).
- If the **String Shape** is **Curved**, the light string was built by creating a circle from three specified control points, the two end points and a point in the middle that defined a third point on the circle. The light point was built along the arc of this circle between the first two control points. If you move a middle point in a curved string, the end points remain fixed, a new circle is formed using these points and the new middle point as control points, and the modified light string is built to follow the arc of the circle between the end points. If you move either the first or last end point, the point at the other end of the string remains fixed, and a new curved light string is built using these end points and the original third control point (which is invisible and is no longer located on the perimeter of the circle). The number of lights in the modified string, and the distance between the lights, depend on the **String Type** (see above).
- If the **String Shape** is **Spiral**, the light string was built using the two end points and two additional control points, which are now invisible. For this reason, it is very difficult to modify a spiral string and achieve predictable results. You must depend on trial-and-error methods. The number of lights in the modified string, and the distance between the lights, will depend on the **String Type** (see above).

Deleting Light Points

You can select and delete individual light points when the **String Type** is Light Point, Random Rectangle, Random Polygon, or Light Grid. To delete a light point, set the mode to Vertex, select a light point, and press Delete.

When the **String Type** is String w/Delta, String w/Count, or String w/Delta and Count, the results differ depending on which point is deleted.

If either end point is deleted, the string will be shortened by one light point (the second or second to last light point becomes the new end point).



If the second or second to last point is deleted, the string will be split into one string and one random light point.



If the deleted point is more than two points from either end, the light string will be split into two strings at the deleted point.



18 Audio Modeling

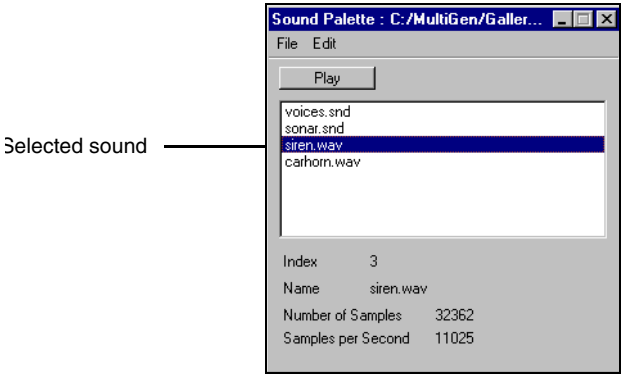
Sound files cannot be created in Creator, but any type of sound file can be loaded and assigned to sound emitters (sound nodes) in a database using the Creator Sound palette.

With Windows and a sound card installed, you can play `.wav`, `.avi`, `.mid`, and `.rmi` files using the Windows default sound player utility `mplay32.exe`. To edit sounds on Windows, you can use the Windows sound recorder utility, `sndrec32.exe`. For IRIX systems, the default utility is the SGI Sound Editor.

The Sound Palette

Every Creator database can have its own Sound palette file (for example, `sound.spl`) containing a list of the sound files available for use in the database. This Sound palette is created the first time you save a Sound palette with a new database.

To open the Sound palette, choose **Palettes/Sound** from the Database window.



Sound Palette Parameters

The following information appears at the bottom of the Sound palette when you select one of the sound files in the palette:

Index

Indicates the sound palette index of the selected file. The index is the list position of the sound in the palette. A sound file is associated with a sound node by assigning its sound palette index to the **Wave Index** attribute of the sound node.

Name

Identifies the name of the selected file.

Number of Samples

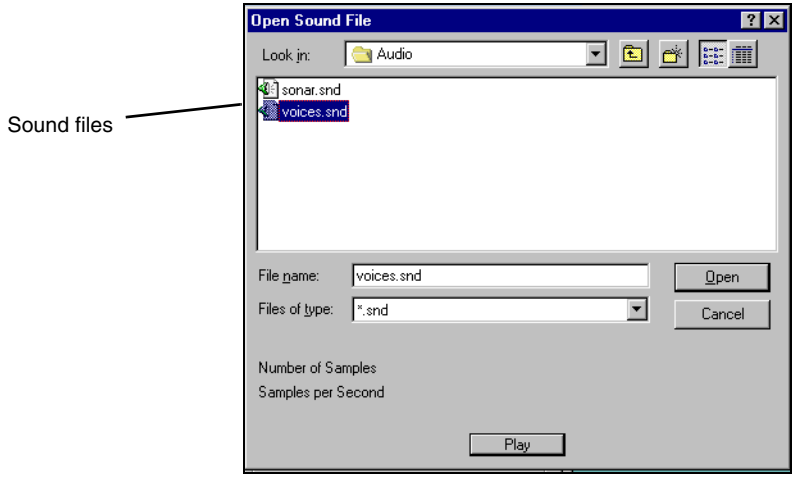
Displays the number of samples present in the selected file.

Samples per Second

Indicates the rate at which samples are played in the selected file.

Loading Sounds

To load a new sound file into the palette, choose **File/Read Sound** and select a sound file in the *Open Sound File* dialog box.



Editing Sounds

On Windows systems with a sound card installed, you can use the default Windows sound recorder utility, `sndrec32.exe`. On IRIX systems, the SGI Sound Editor is the default editing utility. It supports mono, stereo, and 4-channel sound files of 8, 16, or 24-bit audio; however, you can specify any installed third party sound editor by setting the SOUNDEDITOR environment variable (see "MPI_CREATOR_NOSPLASH" on page 44).

To edit the sound file selected in the Sound palette, choose **Edit/Edit Sound** from the **Sound palette** menu to open the sound editing utility. Edit the waveform and save the file. (In Creator, the term *waveform* refers to the data stored in a sound file).

Deleting Sounds

To delete the selected sound file from the Sound palette, choose **Edit/Delete Sound**. The indices of the remaining sounds are not affected.

To delete all sound files not currently assigned to sound nodes in the database, choose **Edit/Remove Unused** from the palette.

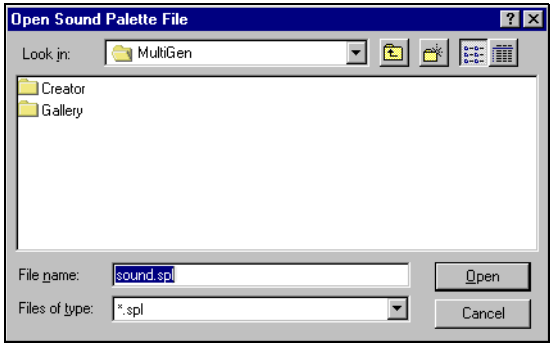
To clear all sound files from the Sound palette, choose **Edit /Delete All**.

Loading and Saving Sound Palettes

A sound palette file (`.spl`) is a text file containing the index, directory path, and file name of each sound file loaded into the Sound palette. Saving and loading custom sound palette files lets you organize sounds to fit a variety of applications.

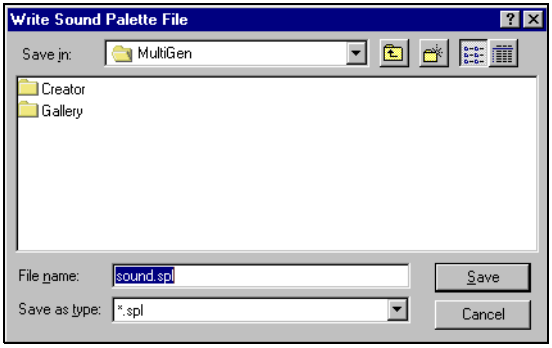
To Load a Sound Palette

To load a sound palette file, choose **File/Load Palette** and select a sound palette file name from the *Open Sound Palette File* dialog box.



To Save a Sound Palette

To save a sound palette file, choose **File/Save Palette** and enter the new file name in the *Write Sound Palette File* dialog box.



Sound Nodes

Sound nodes function as group nodes in the database hierarchy. They are attached to groups, levels of detail, switch nodes, DOF, etc., and can have any node type as a child.

To Create Sound Nodes

Step 1. Set a group or group type node as the database parent.



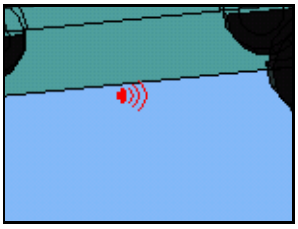
Step 2. Open the **Create** toolbox.

Step 3. Click **Create Sound**. The waveform selected in the Sound palette is assigned to the node. A small sound icon appears in the Graphics view at the coordinate position of the node.



To Hide or Display Sound Nodes

The State preference **Draw Sound** controls whether sound nodes are displayed in the Graphics view. When sound node display is enabled, an icon appears in the graphics view at the sound node's location. The following figure shows the sound node indicator.



You can toggle the display of sound nodes on and off using the **Draw Sound** control in the View panel. When the State preference **Draw Sound** is set, clicking this control turns off sound node display in the Graphics view. Conversely, when the **Draw Sound** preference is cleared, clicking the **Draw Sound** control turns the display on.



Creator's default setting is to display sound nodes in the Graphics view. To change this:

Step 1. Choose **Info/Preferences**.

Step 2. Click the **State** tab.

Step 3. Clear the **Draw Sound** checkbox, and click **OK** to save the change. The current database and all new databases will have sound node display turned off.

To Select Sound Nodes

To select a sound node, click it in the Hierarchy view. To select a sound node in the Graphics view, choose **Sound** modeling mode from the Toolbar, or click **Sound** in the **Mode** toolbox, and then select the sound node in the Graphics view. Alternatively, you can select the sound node in the Graphics view in any modeling mode.

To Insert Sounds

Once a sound node has been created, you can assign a new sound file to it at any time. To assign a sound file selected in the Sound palette to one or more sound nodes:

Step 1. Select the node and choose **Sound/Insert Waveform**.

Step 2. Click **Yes** in the *Verify* dialog box to insert the sound into all sound nodes that are descendants of the selected node. Click **No** to assign the sound to the selected sound node only.



To Remove Sounds

To remove a sound file from one or more sound nodes:

Step 1. Select the node and choose **Sound/Remove Waveform**.

Step 2. Click **Yes** in the *Verify* dialog box to remove the sound from all sound nodes that are descendants of the selected node.

Click **No** to remove the sound from the selected sound node only.

To Position Sound Nodes

The sound node position is the coordinate location from which the sound is emitted. When a sound node is created, it is automatically positioned at the database origin.

To reposition a sound node:

Step 1. Select the sound node icon in the Graphics view.

Step 2. Choose **Sound/Position Sound** in the Database window to open the *Position Sound* dialog box.



Step 3. Select the current position and then the desired position for the sound node in the Graphics view. You can use the grid to position the sound node. See "The Tracking Plane and Grid" on page 303.

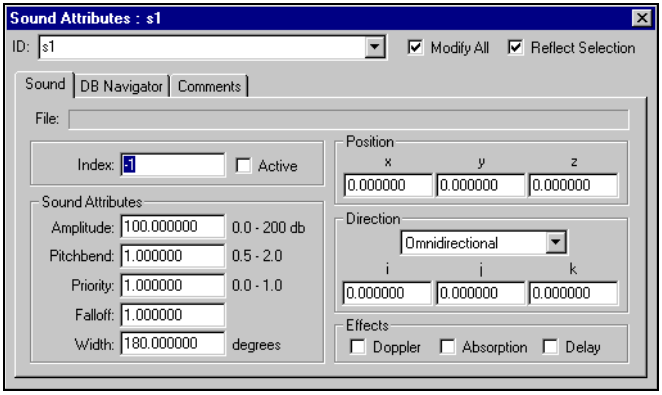
Step 4. Click **OK**.

To Play Sounds in the Database

To play (activate) a sound, select a sound node and choose **Sound/Play Selected**. Only sound nodes displayed in the Graphics view of the database can be activated in this manner.

Sound Attributes

Sound attributes control the physical properties of a sound. To view or edit sound attributes, double-click a sound node in any view of the database or select the node and choose **Attributes/Modify Attributes** (Ctrl+=). The *Sound Attributes* window opens.



The following sound attributes are supported in Creator:

ID

Indicates the alphanumeric identifier of the sound node.

File

Indicates the name of the sound file assigned to the node.

Index

Displays the sound palette index of the sound file assigned to the node. If no sound is assigned to the node, the wave index = -1.

Active

Indicates a sound is to be played when the database is loaded.

Amplitude

Controls the volume of a sound.

Pitchbend

Indicates the ratio of values used to lower or raise the pitch of a sound relative to its base pitch. A value less than 1.0 lowers the pitch; a value greater than 1.0 raises the pitch.

Priority

Determines which sounds will play when there are more active sound nodes than the realtime software can play simultaneously.

Falloff

Indicates the rate at which amplitude falls off when approaching the edge of the sound cone, with maximum amplitude at the center of the cone.

Width

Defines, in degrees, the half angle of the cone of a directional sound.

Position

Designates the *x*, *y*, and *z* coordinate location of the sound node (that is, the position from which the sound is emitted).

Note: The following sound attributes are used by the realtime software. The effects produced by these attributes cannot be previewed in Creator.

Direction

Defines the direction in which the sound is emitted (a normalized vector for the sound).

- **Omnidirectional** emits sound equally in all directions.
- **Directional** concentrates the incidence of a sound in a specific direction. If the observer is behind a sound in the database scene, the sound is inaudible.
- **Bidirectional** emits sound in opposite directions from the same point.

Doppler

Controls the effect of sound wave compression or rarefaction based on the relative motion of the sound and the observer. If **Doppler** is set, sounds rise slightly in pitch as an observer moves closer to the sound and fall slightly in pitch as an observer moves away from the sound. If **Doppler** is cleared, the pitch of the sound does not change as a function of motion.

Absorption

Attenuates sound to simulate the effect of passing through an object or traveling through the atmosphere. When **Absorption** is cleared, no attenuation is performed.

Delay

Calculates the time lag that occurs as a sound travels from the point of emission to the observer. If **Delay** is set, the sound is not emitted until this time has elapsed. Otherwise, the sound is emitted instantly.

Exercise: Adding Sound to the Database

In the following exercise you will load a sound file, insert a sound node, and turn off display of the sound emitter icon. Before you begin, you may want to print this topic to make it easier to follow the instructions as you work with Creator.

Step 1. Choose **Palettes/Sound** or click the **Sound Palette** button in the Toolbar to open the *Sound Palette* window.



Step 2. In the Graphics view, create a simple object.

Step 3. Choose **File/Read Sound** to open the *Open Sound File* dialog box.

Step 4. In the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory, select the sound file `ouch.wav` and click **Open**.

Step 5. Choose **File/Save Palette** to save the Sound palette. Use the default file name (`sound.spl`) or change the file name.

Step 6. Click **Save** in the *Write Sound Palette File* dialog box to save the Sound palette. Close the Sound palette.

Step 7. In the Database window, choose **Info/Preferences** and select the **Draw Sound** State preference.

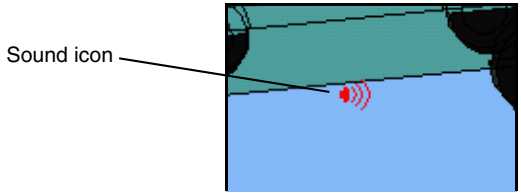
Step 8. Set the database parent.



Step 9. Open the **Create** toolbox.

Step 10. Click **Create Sound**. The waveform selected in the Sound palette is assigned to the node. A small sound icon appears in the Graphics view at the coordinate position of the node.





Note: The default position (the coordinate from which the sound is emitted) of the new sound node is the database origin.

Step 11. Select the sound node icon in the Graphics view.

Step 12. Choose **Sound/Position Sound** from the Database window to open the *Position Sound* dialog box.



Step 13. Select the desired position for the sound node in the Graphics view and click **Done**.

19 Realtime Considerations

Building models that look good is important to everyone. But building models that perform well is of critical importance to realtime modelers. Because your realtime system must be able to redraw everything in a visual scene very quickly, typically as many as 30 times per second, you may find yourself paying as much attention to realtime optimizations as to the obvious appearance of the model. This extra effort is worth it, however. Better realtime performance means you can use more models and produce a richer visual scene.

Realtime Overview

The first step in optimizing your models is understanding your realtime system. Find out the answers to questions such as:

- What is the frame rate?
- How many polygons are drawn per frame?
- What sort of texture memory is available?
- How does the realtime system cull (ignore) unnecessary polygons?
- Is collision detection important?

Once you know what your realtime system requires, you can focus on the features that are important to you. This chapter covers realtime considerations such as draw order, database traversal, viewing volume and collision detection (bounding volumes).

Levels of detail (LODs), lighting, Binary Separating Planes (BSPs), Degrees of Freedom (DOFs), and database hierarchy also influence realtime performance and are discussed in depth in other areas of the documentation. These other factors should be understood when you are designing and optimizing a database for realtime performance.

Frame Rate

Realtime simulators render each scene many times per second. Each rendering is called a *frame*, and the number of frames rendered per second (fps) is called the *frame rate*.

Frame rates in simulations are similar to frame rates in movies and video; the illusion of motion is best at about 30 frames per second. Speeds in excess of 30 fps do not gain anything in terms of simulating motion for the human eye, but may be required to correct visual problems such as double-imaging and flicker. Speeds slower than 10 fps may be frustrating for users and begin to bend the definition of "realtime."

Note: The term *Hertz* (Hz) refers to cycles per second, and is frequently used to indicate frame rate. An update rate of 30 Hz is equivalent to 30 frames per second.

Obviously, anything you can do to optimize your database for speed is worthwhile. If you achieve too much speed you can always trade it for complexity, detail, or texture in your database.

Bounding Volumes and Collision Detection

Collision detection is also an important realtime consideration. It is used by the realtime system when testing whether an object intersects with other objects in order to simulate real world behavior. Balls bounce, cars collide, planes touch down on airport runways, all in a manner reflecting their real world counterparts.

In order for the realtime system to react with the appropriate behavior, it must know when the objects intersect. If the program tested every polygon for collision with every other polygon in the simulation, it would never achieve realtime speeds. As a shortcut, the program tests the *bounding volumes* of objects in the simulation. It is up to you to decide which objects need bounding volumes and the shape of each bounding volume. This is easily achieved in Creator. The full discussion begins with "Bounding Volumes" on page 694.

Draw Order

When viewing a three-dimensional model on the computer screen, your eye is tricked into believing that you are seeing solid objects that show depth. This is especially true when viewing a database with several objects where some objects appear closer than others. If a closer object is between your eyepoint and a distant object, it should obscure the view of the distant object, just as it does in the real world. To achieve this, the closer object is drawn *after* the distant object is drawn. The order in which objects are drawn is called is the *draw order*.

Draw order has a profound effect on the appearance of your models. Used correctly, the draw order can enhance the realism of a scene. Used incorrectly, the draw order can detract from realism, and may slow rendering in the realtime system.

Creator has three types of draw order that you can use in the Graphics view of a Database window. *Fixed List*, *Z-buffer*, and *BSP* (available only if the BSP option is installed in your software).

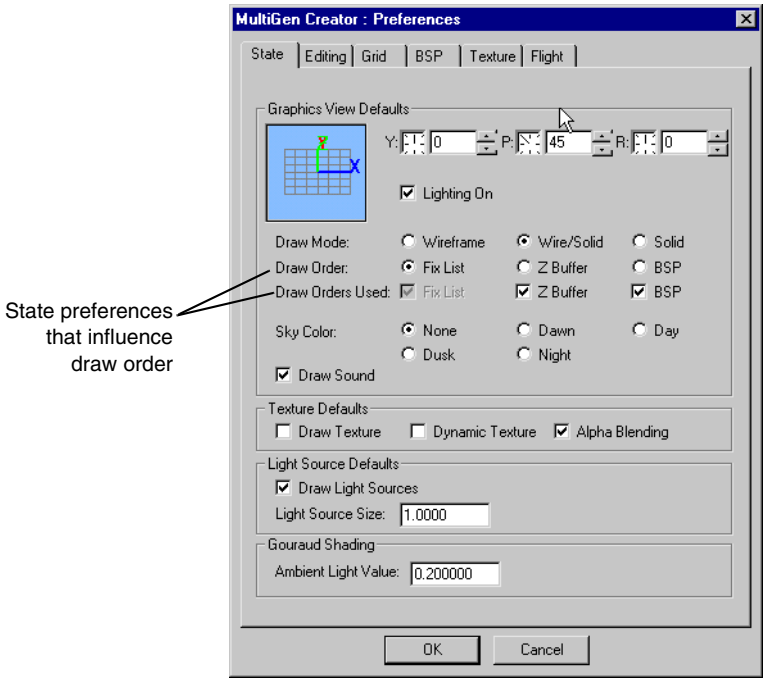
Most realtime systems are based on either Z-buffer or BSPs, but do not offer you a choice between the two. Before you proceed, make sure you know what your realtime draw order will be.

Choosing a Draw Order

Creator draw order does not control the way your realtime system displays a database. Choose a draw order that is convenient to work with, but remember that the ultimate goal is to optimize your models for realtime.

To Set Available Draw Orders

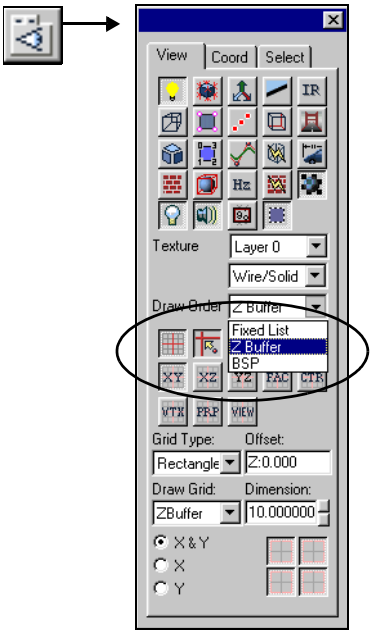
The **Draw Order** and **Draw Orders Used** State preferences control the default draw order behavior in Creator. To edit State preferences, choose **Info/Preferences** (Ctrl+Shift+P) and click the **State** tab.



- **Draw Order** sets the default draw order in the Graphics view.
- **Draw Orders Used** assigns draw orders to the keyboard shortcut Z. Clicking Z in the Graphics view toggles the display between the draw orders that are enabled by this preference.

To Change the Draw Order

After you set the available draw orders in the *State Preferences* window, press the keyboard shortcut Z to switch among the available draw orders in sequence, or open the View panel and choose the draw order you want from the **Draw Order** list.



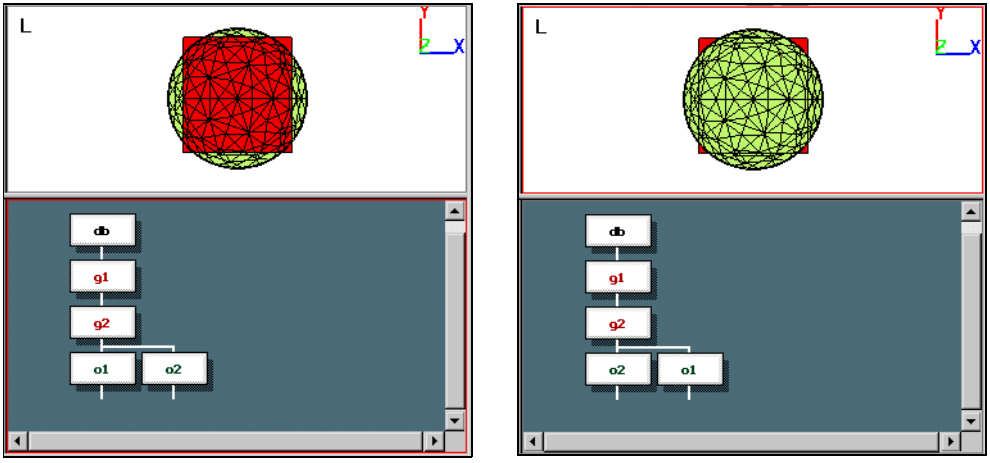
Note: The Graphics view must have focus before you can use either method. To give the Graphics view focus without deselecting in the Hierarchy view, right-click the Graphics view pane.

In Z-buffer draw order, a **Z** appears in the upper-left corner of the Graphics view. In BSP draw order, a **P** appears in the upper-left corner.

Fixed List Drawing

Fixed List is the fastest drawing method, but may not always render a scene correctly. Fixed List relies on the database hierarchy and traversal order to decide which object to draw first (see "Database Traversal Order" on page 688). Each node is processed as it is reached during the database traversal. Thus, each node is drawn on top of (in front of) its left sibling and under (behind) its right sibling.

The following figure shows an example of a Fixed List database. The rectangle (**o2**) appears in front of the sphere (**o1**) if **o2** is on the right in the hierarchy. The rectangle appears behind the sphere if **o2** is on the left in the hierarchy.



Because Fixed List drawing does not take eyepoint into account, it can only be used when you define a hierarchy that is correct when viewed from every potential eyepoint. This can be quite hard (or impossible) to do, so Fixed List drawing is normally used in conjunction with Binary Separating Planes.

Binary Separating Plane (BSP) Drawing

Binary Separating Plane (BSP) draw order uses planes that are modeled into the database to determine drawing priority. Nodes on the same side of a BSP as the eyepoint are drawn on top of nodes on the opposite side of the plane. Nodes that are not separated by planes are drawn using Fixed Listing. For more information, see “Combining BSPs and Fixed Listing” on page 908.

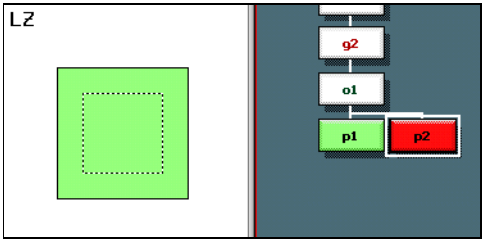
Z-buffer Drawing

Z-buffer order draws a scene very accurately, but at a higher performance cost than Fixed List or BSP. In Z-buffer draw order, the visual system:

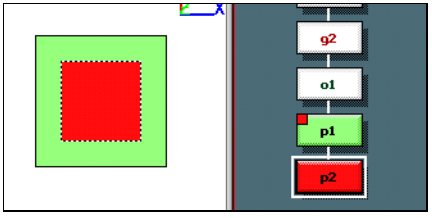
- Computes the distance from the eyepoint to each pixel on a visible surface. This distance is referred to as *pixel depth*.
- Draws the visual scene. Each pixel is drawn only if it is closer to the eyepoint than the last pixel drawn in that position on the screen (if it is farther away, the pixel is discarded). This ensures that the closest surface always draws on top.

Coplanar Faces and Subfaces

Z-buffer cannot resolve coplanar faces. If you model one face on top of another (for example, to add a window to the wall of a house) both faces are the same distance from the eyepoint and the Z-buffer system cannot tell which face to draw on top.



The usual method of indicating draw order for coplanar faces is to attach the face that should appear in front as a subface of the face that should appear in back, as shown below. The Z-buffer guarantees that all subfaces are drawn on top of parent faces.



Pixel Depth and Clipping Planes

Pixel depth is the distance from the eyepoint to each pixel in a visual scene. However, the default pixel depth (before Z-buffer computation is performed) is determined by the distance between the near and far clipping planes. If the distance between clipping planes is too large, you may see drawing errors such as cracks between faces. To correct this problem, adjust the near and far clipping planes to fit the scale of the database as nearly as possible (see “The Viewing Volume” on page 689).

The near and far clipping planes you define for Creator do not affect your realtime system. You may also need to adjust clipping planes on your realtime system to optimize the view of a Z-buffered database.

Database Traversal Order

The *traversal order* is the order in which the program processes the hierarchy. Creator reads the hierarchy from top to bottom and then from left to right. Nodes that are higher in the hierarchy are processed before nodes that are lower in the hierarchy. Similarly, nodes to the left in the hierarchy are drawn before nodes to the right. Each node is processed relative to its neighbors as follows:

1. The node's left sibling is processed.
2. The node is processed.
3. The node's children are processed, from left to right.
4. The node's right sibling is processed.

Some functions, such as Fixed List drawing and the rendering of face transparency, use the order of nodes in the database traversal to determine the final appearance of the Graphics view. This order can be changed using *relative priority*.

Relative Priority

Relative priority is an attribute of groups, objects, and faces that allows you to set a numerical value that controls a node's position in the hierarchy relative to its siblings. The hierarchy is periodically sorted so that nodes with lower **Relative Priority** are moved to the left of siblings with higher **Relative Priority**. (Siblings with equal **Relative Priority** are left alone.) By default, a node has a priority of zero, so newly created nodes sort to the left.

Because the order in which nodes are processed during a database traversal depends entirely on the hierarchy, you can use relative priority to control the order of processing among siblings.

To prevent yourself from accidently shifting nodes too far to the left or right (based on priority), choose **Info/Preferences/Flight** and set the **Check Relative Priority when Shifting Nodes Left or Right** checkbox. When this checkbox is set, the relative priority of a node

is always checked before you are allowed to move it. See “Flight Preferences” on page 60.

The Viewing Volume

The *viewing volume* defines the area of the database that is visible from the eyepoint. Although the viewing volume you set in Creator does not carry over into the realtime system, it is important to know how to adjust the viewing volume in both Creator and the realtime system, because it directly affects the number of polygons the realtime must process for each frame. Polygons inside the viewing volume are processed; polygons outside the viewing volume are culled (ignored).

The viewing volume is a cone, defined by the *field of view*, that extends from the *eyepoint* in the direction of the *viewing vector*. The amount of geometry visible within this cone is further restricted by the *near clipping plane* and the *far clipping plane*. The *eyepoint scale* controls the magnification of the visible scene relative to the eyepoint.

Changing the Clipping Planes

You may want to adjust the viewing volume to show only part of the database or to optimize the speed of MultiGen. When viewing large databases or when using Z-buffering, the computational requirements of the database may slow the speed of the display in the Graphics view. Either narrowing the field of view or moving the clipping planes can increase the viewing performance.

To experiment with clipping planes, try the following:

Step 1. Open the file `airfield.flt` from the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory.

Note: If you did not install the exercise files when you installed Creator, you can open this file directly from the Creator installation CD.



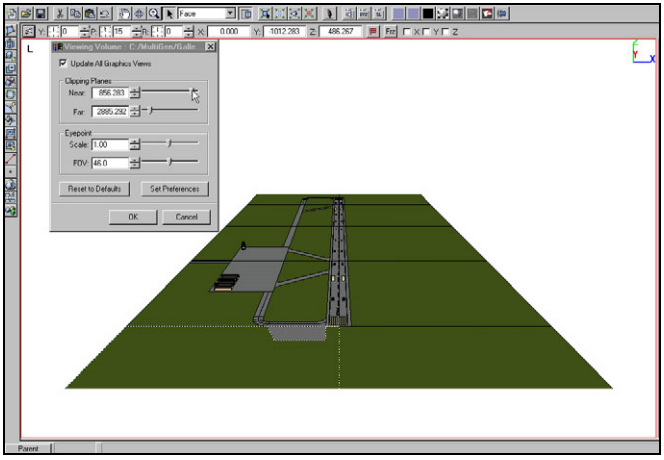
Step 2. Open the View panel and turn off the grid display, if it is on.

Step 3. Adjust the Eyepoint controls to 0,15,0.



Step 4. Zoom in until you are closer to the airfield, but far enough away to see the near and far end of the field.

Step 5. Choose **View/Viewing Volume**.



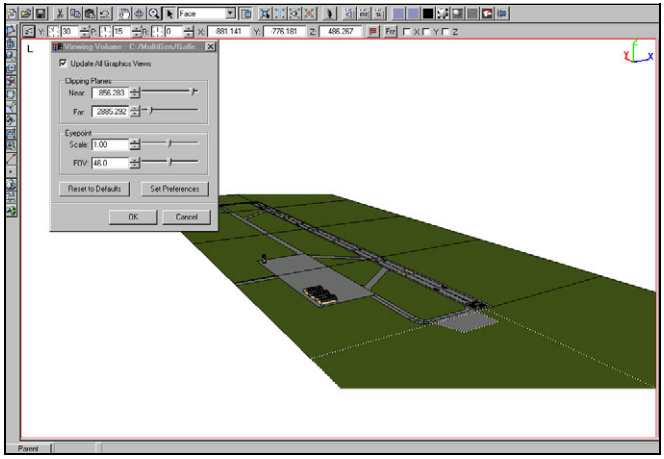
Step 6. In the *Viewing Volume* dialog box, *slowly* drag the **Near** slider to the left and right.The near edge of the airfield will get shorter and longer.

Step 7. Adjust the slider until the edge of the airfield is close to the edge of the runway. Click the up and down arrow buttons next to the **Near** field to make fine adjustments. If you change the viewing volume too much, click **Reset to Defaults**. Then try adjusting the slider again.

Step 8. *Slowly* drag the **Far** slider to the left and right.

Step 9. Move the far end of the airfield until it is close to the far end of the runway.

Step 10. Change the Eyepoint controls to 30, 15, 0. Adjust the near and far clipping planes as you did in the previous steps.



The airfield is now clipped at an angle. Your eyepoint's field of view relative to the database determines the angle at which the viewing volume's near clipping plane cuts the end of the airfield.

Step 11. Close the file.

Adjusting the Eyepoint Scale

The eyepoint scale affects the apparent size of geometry within the viewing volume. Scale is provided for modeling convenience, and does not affect the geometry.

To experiment with the eyepoint scale, try the following:

Step 1. Open the `airfield.flt` file from the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory.

Note: If you did not install the exercise files when you installed Creator, you can open this file directly from the Creator installation CD.

Step 2. Choose **View/Viewing Volume**.

Step 3. In the *Viewing Volume* dialog box, type 90 in the **FOV** field and press the Tab key. This decreases the field of view by half.

Step 4. Type 2 in the **Scale** field and press the Tab key. This increases the eyepoint scale by a factor of two.

Note that the apparent size of the database is approximately the same as it was when you opened the file. In the Graphics view, doubling the scale has negated the effect of changing the field of view. But note that scale does not change the size of the geometry, only its display. Before you export a model to realtime, be sure to check its appearance with the scale factor set to one.

Step 5. Close the file.

Multiple Eyepoint Viewing Volumes

Each Graphics view can have its own viewing volume, or you can apply the settings to all the Graphics views.

To experiment with viewing volumes, try the following:

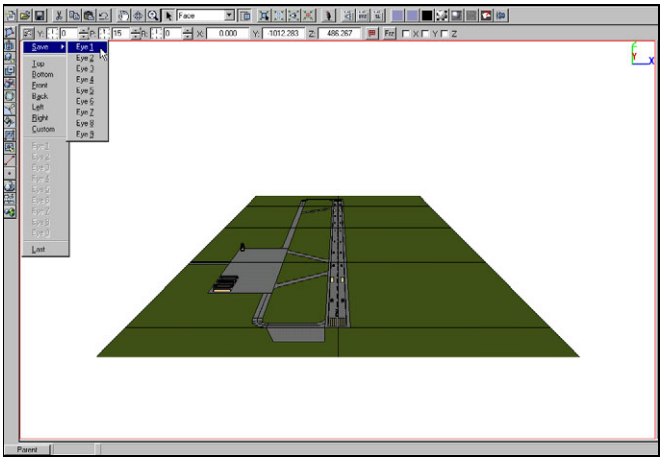
Step 1. Open the file `airfield.flt` in the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory.

Note: If you did not install the exercise files when you installed Creator, you can open this file directly from the Creator installation CD.

Step 2. Adjust the Eyepoint controls to 0, 15, 0.



Step 3. Click the **Eyepoint** menu, and then click **Save** and choose **Eye 1**.

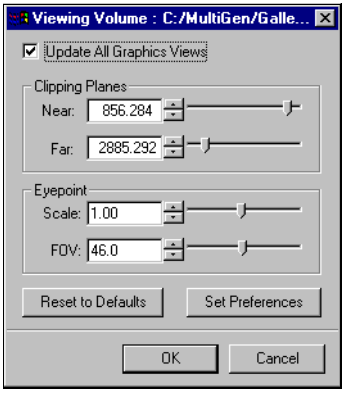


This adds the current eyepoint position to the **Eyepoint** menu as **Eye 1**. You can return to this eyepoint position later by choosing it from the **Eyepoint** menu.

Step 4. Adjust the Eyepoint controls to 90, 90, 0.

Step 5. Click the **Eyepoint** menu, then click **Save** and **Eye 2**.

Step 6. Choose **View/Viewing Volume**.



Step 7. In the *Viewing Volume* dialog box, drag the **Near** slider until the viewing volume is visibly decreased.

Step 8. Make sure the **Update All Graphics Views** checkbox is set.

Step 9. Choose the **Eyepoint** menu and then select **Eye 1**. The viewing volume of both Graphics views is now the same size although you are looking at the database from different angles.

To set different viewing volumes for each Graphics view, clear the **Update All Graphics Views** checkbox and change the viewing volume for each eyepoint. (Change focus between Graphics views by clicking or pressing Tab.)

Bounding Volumes

One of the tasks a realtime system must perform is *collision detection* (also known as *intersection testing*) which means testing models that move to see if they have come into contact with anything in the database. Collision detection allows models in a simulation to interact with one another.

Because testing all the faces in a database for points of intersection can be very time-consuming, many realtime systems approximate the surface of a model by wrapping it in a box, sphere, or other simplified shape, called a *bounding volume*. During a simulation, the realtime system periodically checks the bounding volumes for points of intersection. If two or more bounding volumes intersect, the realtime system responds as if the models had collided.

Some realtime systems use bounding volumes that you model in Creator, while others generate their own bounding volumes when a database is loaded. In Creator, bounding volumes are assigned to groups. For information on creating bounding volumes, see “Group Attributes: Bounding Volume Panel” on page 836.



You should turn on bounding volumes only for objects that logically need it; managing each bounding volume takes processing time and therefore slows the speed of the realtime system. Once you turn on a bounding volume, you can turn its display off and on using the **Draw Bounding Volume** control in the View panel.

Displaying Bounding Volumes

The following example demonstrates how to display bounding volumes in Creator:

Step 1. Open a new database.



Step 2. Open the **Geometry** toolbox.



Step 3. Click the **Sphere** tool and make a sphere.

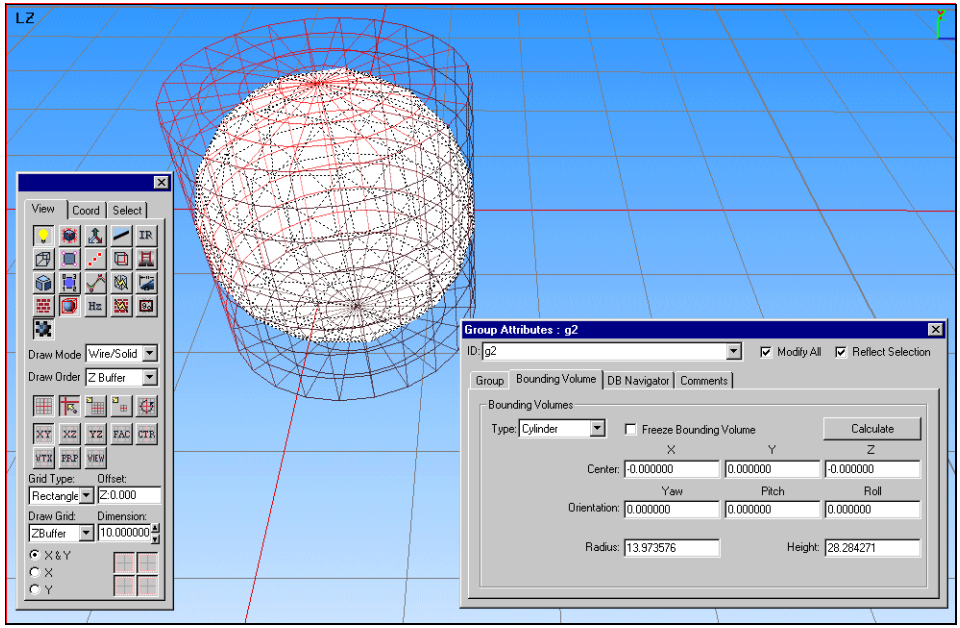
Step 4. In Group modeling mode, double-click the sphere to display the *Group Attributes* window.

Step 5. Click the **Bounding Volume** tab.

Step 6. Select **Cylinder** from the **Type** list. The bounding volume will not be visible until you turn on **Draw Bounding Volumes** in the next step.

Step 7. Open the View panel and click **Draw Bounding Volumes**. This turns on the display of the bounding volume in the Graphics view. By default, bounding volumes are not displayed.





Changing the Bounding Volume

The following example demonstrates how to change a bounding volume:



Step 1. Open a new database.

Step 2. Adjust the Eyepoint controls to 0, 90, 0 and zoom out until you can see at least 8 grid lines to the left of the origin.

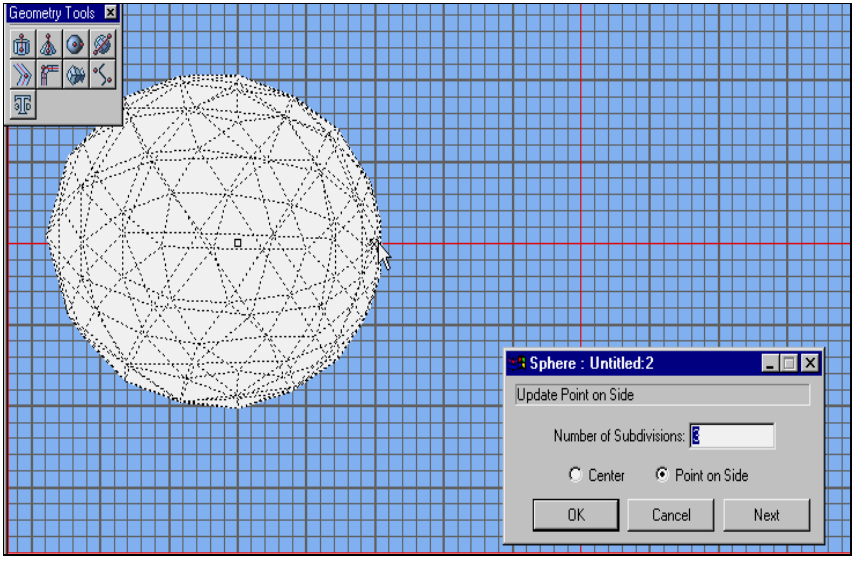
Step 3. If the grid is not visible, click **Grid On/Off** in the View panel.

Step 4. Open the **Geometry** toolbox.

Step 5. Click the **Sphere** tool.

Step 6. In the Graphics view, click a point five units to the left of the origin to set the center point for the sphere.

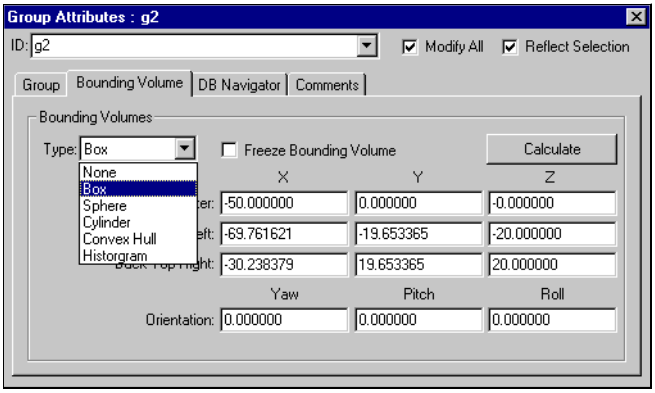
Step 7. Click a point three units to the left of the origin to set the side point for the sphere and then click **OK** in the *Sphere* dialog box.



Step 8. In the Hierarchy view, double-click the sphere group node.

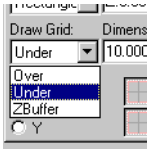
Step 9. In the *Group Attributes* window, click the **Bounding Volume** tab.

Step 10. Choose **Box** from the **Type** list.

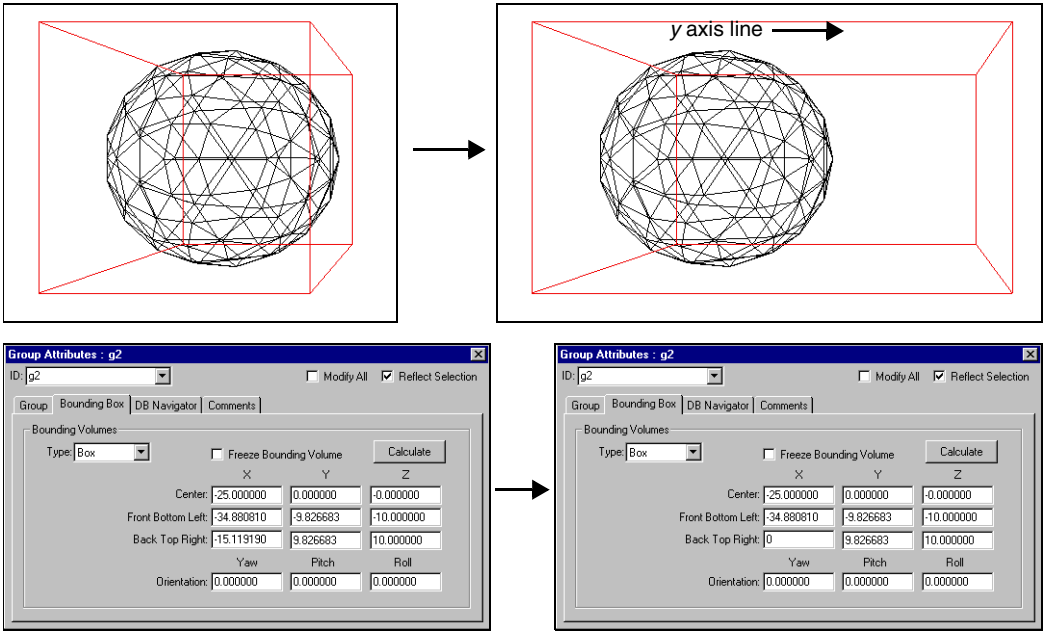


Step 11. Right-click the Graphics view to give it focus, and then turn on **Draw Bounding Volumes** in the View panel to display the bounding volume. If you have trouble seeing the bounding volume, select **Under** from the **Draw Grid** list in the View panel.





Step 12. In the *Group Attributes* window, change the first field in the **Back Top Right** row to 0 (zero) and press the Tab key. This moves one side of the bounding volume to the y-axis line at the database origin.



Step 13. Try moving other sides of the bounding volume by entering values in the other fields.

Step 14. To reset the bounding volume to its default settings, click **Calculate**.

Step 15. When you finish customizing the bounding volume, set the **Freeze Bounding Volume** checkbox.

Translating Bounding Volumes

Because you define bounding volume at the group level, several objects may be included in a single bounding volume. You should set up your database to logically reflect the need for collision detection. For example, a group may include the body, wings, and other parts of an airplane. If the bounding volume for that group is used for collision detection, the realtime system will detect collisions with any part of the airplane.

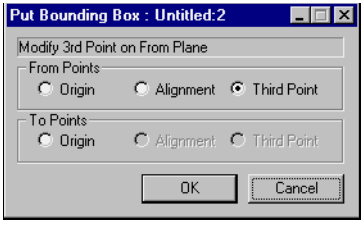
You can move the bounding volumes by choosing **Attributes/Put Bounding Volume**. This allows you to translate a bounding volume to another position relative to the object. This may be necessary if you want to increase or decrease the distance at which the collision takes place.

To Translate a Bounding Volume

When you translate a bounding box, you move three points *from* one position *to* another, and the positions of all the other points in the bounding box are calculated accordingly.

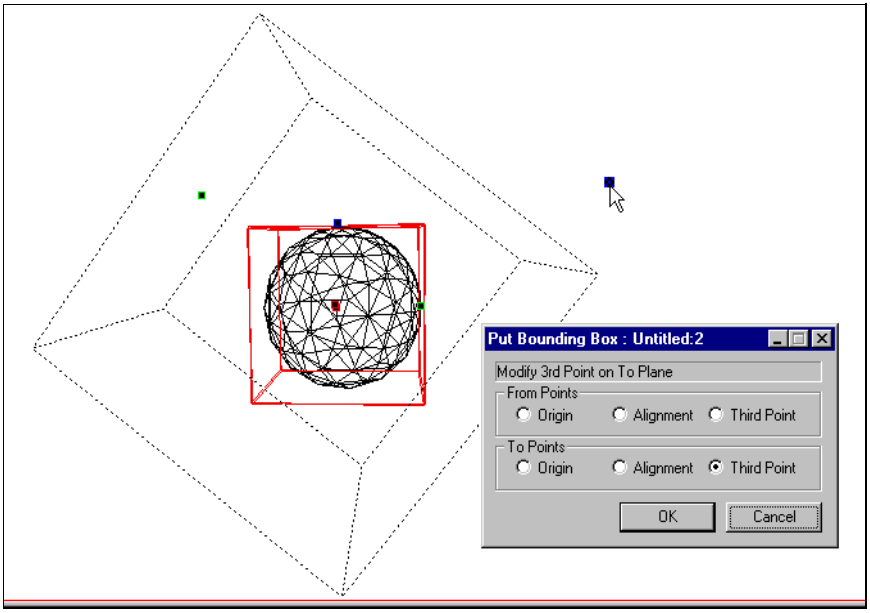
Step 1. Select the group node that contains the bounding volume you want to translate.

Step 2. Choose **Attributes/Put Bounding Volume**.



Step 3. In the Graphics view, click three **From Points** (the *origin*, *alignment*, and *third* points). The display in the *Put Bounding Box* dialog box changes accordingly. These are the points on the currently-selected object that define the bounding volume.

Step 4. Select the three **To Points**. These are the points where you want the bounding volume, relative to the geometry.



Step 5. Click **OK**.

Step 6. To reset the bounding volume to its default settings, click **Calculate** in the *Group Attributes* window.

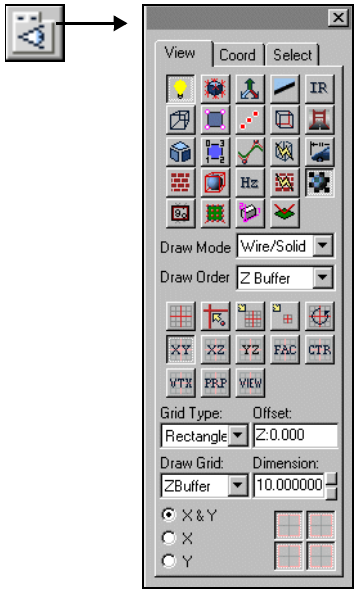
Exercise: Draw Order

This example demonstrates draw order in relationship to the database hierarchy. Before you begin, you may want to print this topic to make it easier to follow the instructions as you work with Creator.

Prepare for Modeling

Step 1. Choose **File/New** to create a new database file.

Step 2. Click the **View panel** button in the Toolbar to display the View panel. Drag the View panel to a convenient location on your screen.





Step 3. Click **Grid On/Off** in the View panel to display the grid.

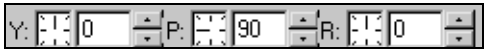
Step 4. Make sure **Fixed List** is selected in the **Draw Order** list.

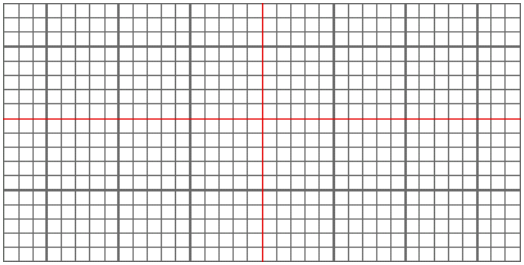
Step 5. Set the **Draw Mode** to **Solid**.

Step 6. Click the **Primary Color** button in the Toolbar to open the Color palette, and pick a primary color to use for your geometry.



Step 7. In the Graphics view, pitch the eyepoint up above the tracking plane by setting the Eyepoint clocks to **Y**aw = 0, **P**itch = 90, and **R**oll = 0.



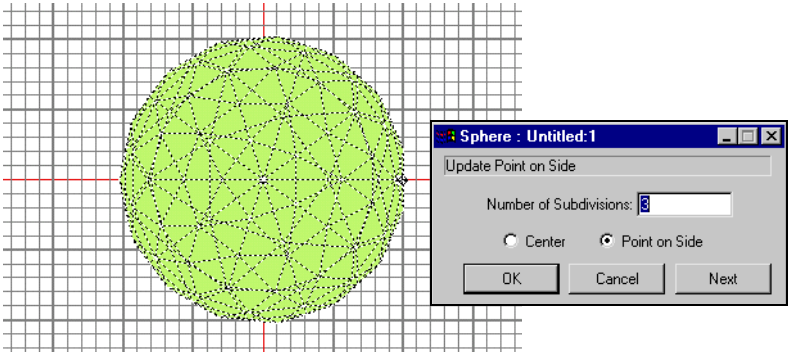


Create and View Geometry



Step 1. Open the **Geometry** toolbox.

Step 2. Click **Sphere** and create a sphere centered at the origin.



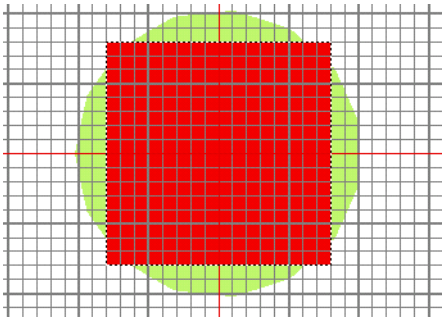
Step 3. Pick a new primary color in the Color palette.



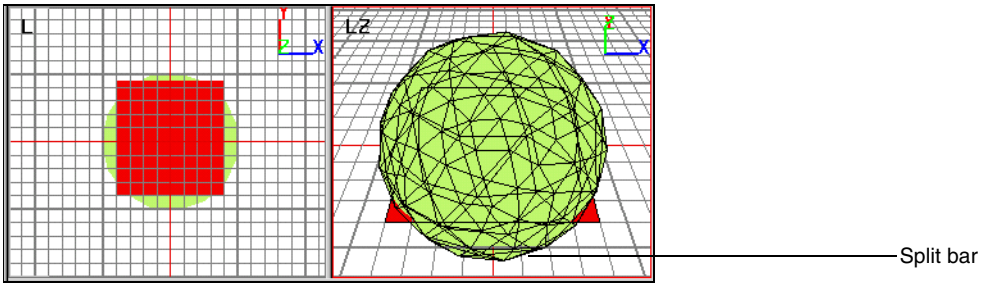
Step 4. Open the **Face** toolbox.



Step 5. Click **Rectangle**, and create a face that is centered at the origin and that partially covers the sphere. Notice that even though the sphere has height and the rectangle is flat, the rectangle appears in front of the sphere.

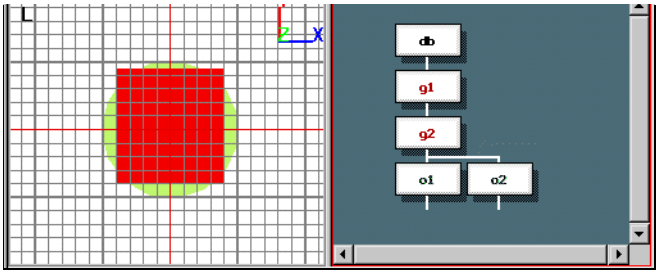


Step 6. Point to the Split bar at right side of the window and drag it left to split the Database window into side-by-side Graphics views.



Step 7. Click in the right Graphics view to make it active (the active view is outlined in red), and then press the keyboard shortcut S to change to a Hierarchy view.

Step 8. Middle-click in the Hierarchy view to display the two object nodes you created.

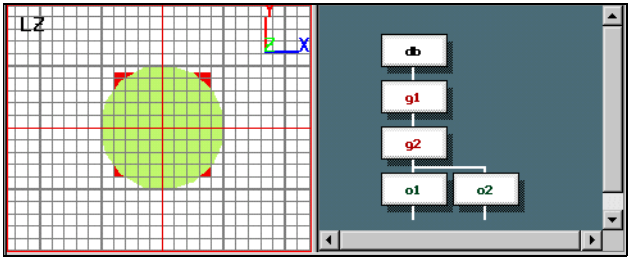


Step 9. Select the rectangle and observe that this is the object node on the right in the Hierarchy view.

In **Fixed List** draw order, objects are drawn from left to right, according to their position in the Hierarchy view. This is why the rectangle appears to be in front of the sphere in the Graphics view.

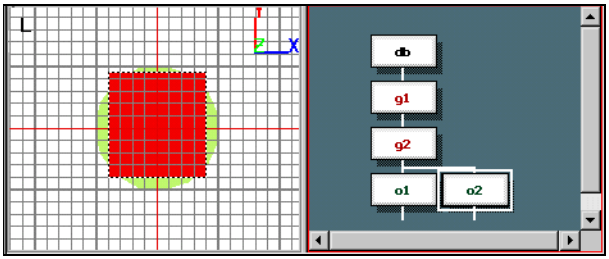
Step 10. In the View panel, select **Z-buffer** from the **Draw Order** list and observe that the sphere correctly covers the rectangle.

In Z-buffer draw order, pixels are drawn according to their distance from the eyepoint, giving a more accurate representation of the visual scene. However, Z-buffer drawing is slower than **Fixed List** drawing unless you have special Z-buffer hardware.



Step 11. In the View panel, select **Fixed List** from the **Draw Order** list.

Step 12. Select the object node for the rectangle in the Hierarchy view.



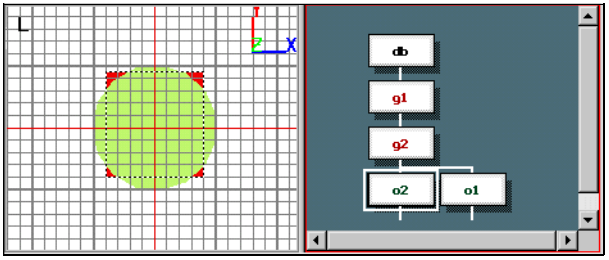


Step 13. Open the **Hierarchy** toolbox.



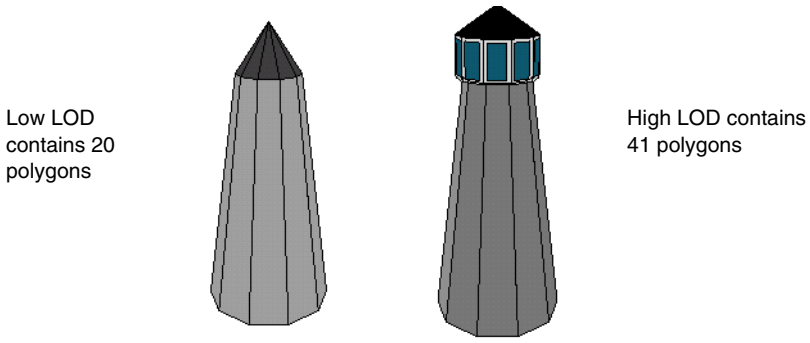
Step 14. Click **Shift Far Left** and observe the results in the Graphics view.

Because the sphere is now to the right of the rectangle in the Hierarchy view, it correctly appears to be on top in the Graphics view.



20 Levels of Detail

Levels of detail (LODs) are sets of models that represent the same object or terrain area with varying degrees of complexity. The realtime system selects one of the LODs to display, depending on the distance from the eyepoint to the LOD and on the number of polygons the realtime system can process.

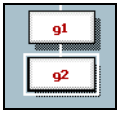


If the eyepoint is far away from an object, the object is displayed as a low LOD, containing relatively few polygons. As the eyepoint moves towards the object, the realtime system replaces it with increasingly complex LODs. LODs play an important part in optimizing a visual scene for drawing, because there is an effective limit on the number of polygons each realtime system can display. When you design a database that includes LODs you can concentrate the greatest visual detail in the area that is closest to the viewer.

Creating LODs

To create an LOD:

Step 1. Select a node in the database hierarchy to be the parent of the LOD.



Step 2. Click the **Parent** button at the bottom of the Database window to make the selected node the current parent.

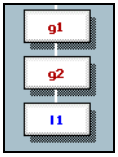




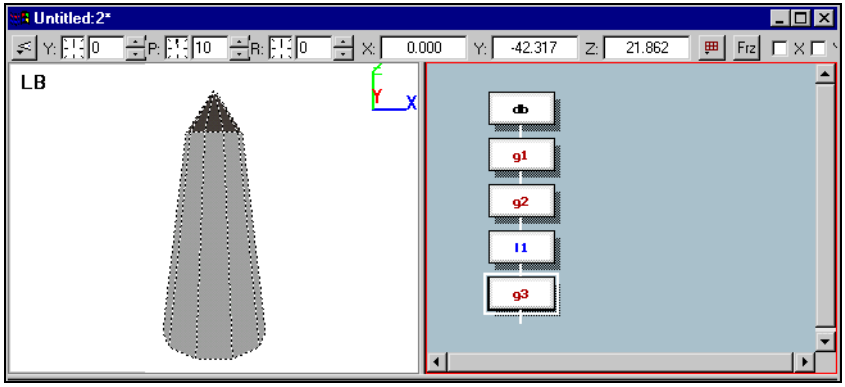
Step 3. Open the **Create** toolbox.

Step 4. Click **Create LOD**. A new LOD node appears in the hierarchy under the current parent.

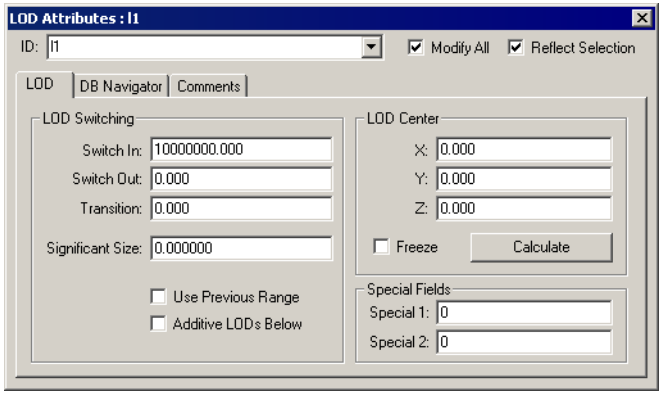




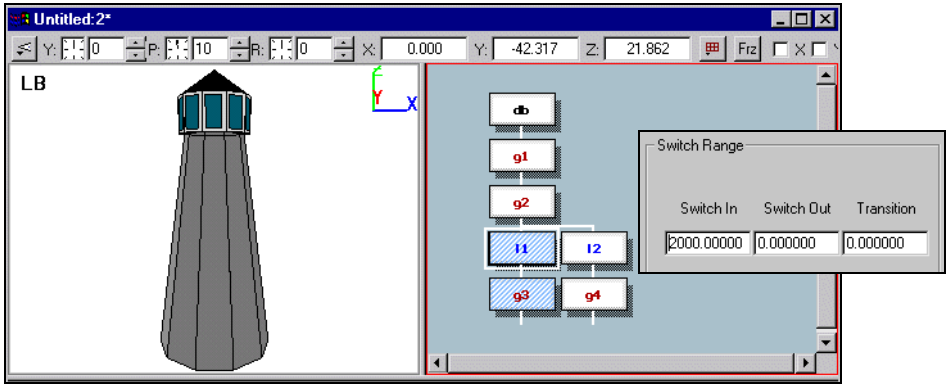
Step 5. Build or attach a model that defines one of the LODs. You may also want to build hierarchy that helps the realtime system choose an LOD for display (see “Level-of-Detail Hierarchy” on page 709).



Step 6. Double-click the LOD node to open the *LOD Attributes* window, and set the **Switch In** and **Switch Out** attributes to define the range in which the LOD is visible (see “Switch In and Switch Out” on page 713).



Step 7. Repeat this procedure for each additional LOD in the set.



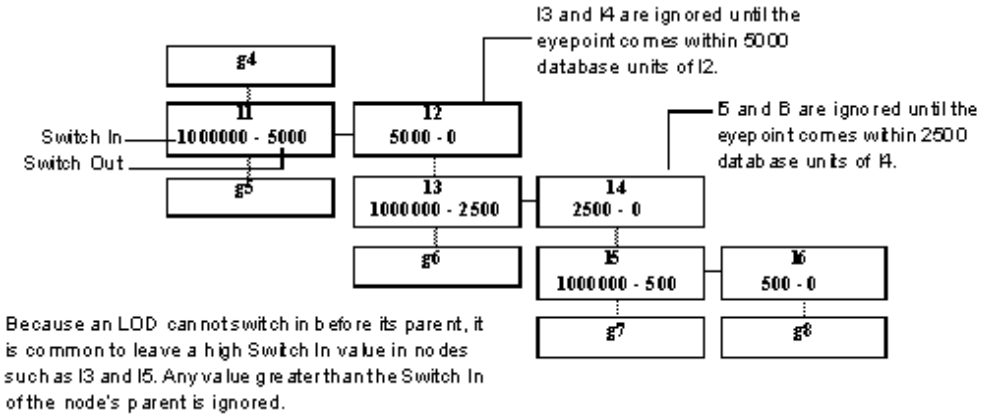
Adding a High Level of Detail

Level-of-Detail Hierarchy

OpenFlight specifies that when an LOD is *switched out* (not displayed), none of its descendants can be *switched in* (displayed). Because the realtime system does not process anything in the database hierarchy below a switched-out LOD, you can save drawing time by nesting LODs in the hierarchy.

Nested LODs

In the following figure, **g5**, **g6**, **g7**, and **g8** are models attached to nested LODs that all represent the same item.



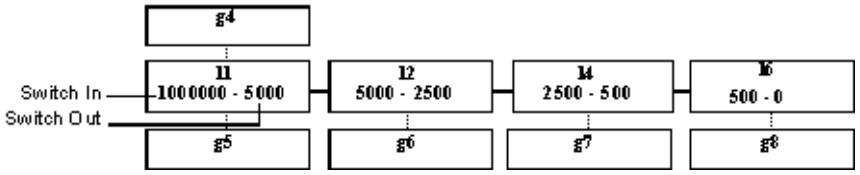
When the realtime system draws this database:

- The realtime system chooses to display **I1**, **I2**, or nothing, depending on the distance from the current eyepoint to the center of the LOD.
- If **I2** is switched out, its descendants are ignored.
- If **I2** switches in, the realtime system selects either **I3** or **I4** for display.
- If **I4** switches in, the realtime system selects either **I5** or **I6** for display.

Setting up this sort of hierarchy lets the realtime system *cull* (ignore) LODs that are not attached to the LOD node selected for display.

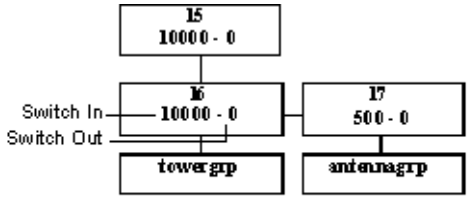
Independent LODs

Although LOD nodes are often nested for efficient display in realtime, they can also be placed individually in the database hierarchy. In the following figure, each LOD is selected for display solely on the basis of its own **Switch in** and **Switch Out** attributes. This means the realtime system must separately test each node for display.



Additive LODs

Additive LODs are pieces of a model that switch in and stay in, accumulating detail as the eyepoint approaches the center of the model. For example, a radio tower might contain two LODs: a tower body that switches in at a large distance from the eyepoint, and an antenna that switches in when the eyepoint draws very near.



Because l6 has a Switch Out value of zero, it stays visible when l7 switches in.

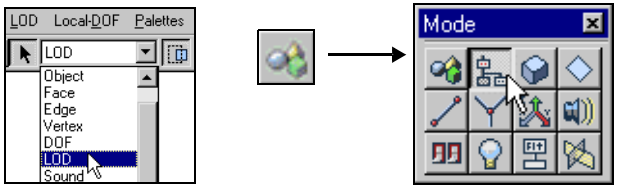
If you are using Binary Separating Planes (BSPs), be aware that Creator's automatic separation tools do not normally separate sibling LODs. To use additive LODs on a BSP system, create a parent LOD and set its **Additive LODs Below** attribute. This tells the software that some or all of the LOD's children might be visible at the same time, and therefore may need to be separated from each other. For more information on BSPs, see "Designing Separable Databases" on page 907.

Selecting LODs



To select an LOD in the hierarchy view, click it.

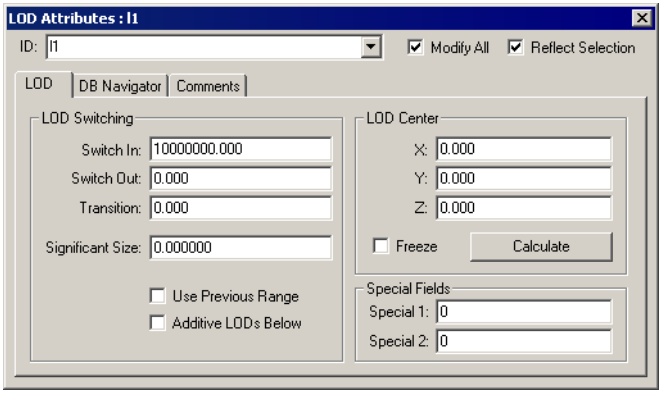
To select an LOD in the Graphics view, Creator must be in LOD modeling mode. To turn on LOD modeling mode, choose **LOD** from the **Mode Selection** list in the Toolbar, or openthe **Mode** toolbox and click LOD.



Displaying and Modifying LOD Attributes

To display LOD attributes, double-click an LOD or select an LOD and choose **Attributes/Modify Attributes** (Ctrl+=). The *LOD Attributes* window has three panels: **LOD**, **DB Navigator**, and **Comments**. The **DB Navigator** and **Comments** panels are common to the attributes windows for other node types, and are described in "OpenFlight Nodes and Attributes" on page 825.

The LOD panel in the *LOD Attributes* window controls how LODs are displayed during realtime. The following sections describe the attributes you set in this panel.

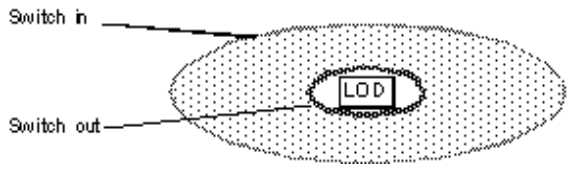


ID

Displays the alphanumeric identifier of the node.

Switch In and Switch Out

Define the range over which an LOD is displayed. **Switch In** must be greater than **Switch Out**.



Consider the case of an eyepoint moving towards the center of an LOD:

The realtime system calculates the distance from the eyepoint to the geometric center of the LOD. This distance is known as the *slant range*.

When the slant range becomes less than or equal to **Switch In**, the realtime system displays the LOD.

As the eyepoint continues to move towards the LOD, the slant range decreases. When the slant range becomes less than **Switch Out**, the realtime system turns off the LOD. At this point, it is common for a higher LOD to switch in.

Transition

Definesthe range in which one LOD morphs into another (see "How Morphing Works" on page 716).

Significant Size

A value assigned to an LOD for calculating switch ranges based on display settings such as resolution, screen size, and field-of-view.

Use Previous Range

Indicates an LOD is a replacement for its left sibling. Using this flag saves processing time, because the realtime system calculates the slant range (distance from the current eyepoint to the LOD) once and then selects one of the sibling LODs for display. If this flag is off, the realtime system recalculates slant range and tests each node separately.

Additive LODs Below

Indicates all LODs attached to this LOD node are additive (see "Additive LODs" on page 711). Because they may be displayed at the same time, additive LODs are often separated on systems that use binary separating planes.

LOD Center

Defines the center of an LOD for use in calculating slant range (the distance from the eyepoint to the LOD). Creator automatically recalculates this coordinate each time the database file is saved to disk. You can define the LOD center manually, as follows:

To set the center to the geometric center of the LOD, click **Calculate**.

To specify another coordinate as the center of the LOD, enter the coordinate and set the **Freeze** checkbox to disable automatic recalculation.

Special 1 and Special 2

Provide spare data fields that can be defined by the realtime system.

Switching LODs in Creator

You can use options in the **LOD** menu to switch in and out of LODs in Creator, giving you a preview of how a database looks as the eyepoint moves closer and farther away. These options do not affect how LODs are processed and displayed in the realtime system.

More Detail (Ctrl+M)

Switches out the displayed LOD and switches in the next higher LOD, regardless of the eyepoint position.

Most Detail (Ctrl+Shift+M)

Switches out the displayed LOD and switches in the highest LOD, regardless of the eyepoint position.

Less Detail (Ctrl+L)

Switches out the displayed LOD and switches in the next lower LOD, regardless of the eyepoint position.

Least Detail (Ctrl+Shift+L)

Switches out the displayed LOD and switches in the lowest LOD, regardless of the eyepoint position.

Set LOD Distance

Prompts you for a **Distance to LOD Center** and turns each LOD in the database on or off based on this distance. Each LOD is displayed if the distance you enter falls between its **Switch In** and **Switch Out** attribute values, regardless of the eyepoint.



Auto LOD

Turns on automatic LOD drawing mode. In this mode, Creator dynamically turns each LOD on or off as the eyepoint moves through the database. An LOD is displayed if the distance from the eyepoint position to the center of the LOD falls between its **Switch In** and **Switch Out** attribute values.

Eyepoint LOD

Sets the display of each LOD in the database on or off based on its distance from the current eyepoint position. An LOD is displayed if this distance falls between its **Switch In** and **Switch Out** attribute values. Note that this is done using the current eyepoint, but does not automatically update if you change the eyepoint. (For automatic LOD switching, choose **LOD/Auto LOD**.)

Morphing Between LODs

One problem that arises when using LODs is the visual discontinuity, or *popping*, that occurs as one LOD switches with another. *Morphing*, the visual merging of one LOD into another, can solve this problem.

In morphing, each vertex in an LOD is allocated a *morph vertex*—usually the closest vertex in the next lower LOD. When a higher LOD switches in, the realtime system first displays the morph vertices (located along the lower LOD) and then updates the visual scene in gradual steps until all the higher LOD vertices are displayed.

How Morphing Works

Morphing is controlled by the following LOD attributes:

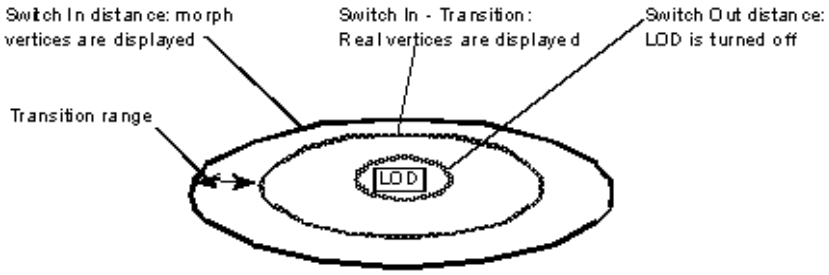
Center specifies the geometric center of the LOD.

Switch In is the distance from the **Center** to the eyepoint at which morphing begins. (**Switch Out** specifies the distance at which the eyepoint is too close to display an LOD, and is not specifically used in morphing.)

Transition is the distance over which the LOD morphs.

Creator displays morphing by dividing **Transition** by ten and moving each morph vertex to its real vertex location in ten equal steps. The shorter the transition distance, the faster the morph. Note, however, that your realtime system may display morphing differently.

When the eyepoint reaches the **Switch in** point, the LOD's morph vertices are displayed. As the eyepoint moves towards the Center, through the transition distance, the morph vertices gradually move towards the higher LOD vertex positions. After the eyepoint passes through the transition range, the higher LOD is displayed.



Working With Morph Vertices

You can choose to use morphing to reduce or eliminate LOD popping (see "Morphing Between LODs" on page 716). The following topics describe how to create and manage LOD morphing.

To Automatically Create Morph Vertices

Step 1. Select two *consecutive LODs*. To be consecutive, the **Switch In** attribute of the higher LOD must match the **Switch Out** attribute of the lower LOD.

Step 2. Choose **LOD/Find Morph Vertices**. For each vertex in the higher LOD, Creator creates a morph vertex by copying the closest vertex in the lower LOD.

Step 3. Open the attributes of the higher LOD and set the **Transition** range. This can be any value, but is typically between five and ten percent of the distance in which the LOD is visible (calculated by subtracting **Switch Out** from **Switch In**).

Step 4. Choose **LOD/More Detail** (Ctrl+M) and **LOD/Less Detail** (Ctrl+L), and verify that the two LODs morph smoothly back and forth.

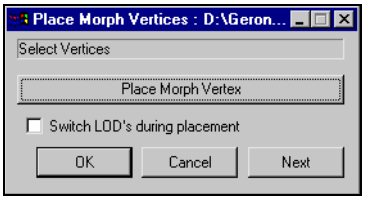
To Manually Create or Edit Morph Vertices

Step 1. Display the higher LOD in the Graphics view.

Step 2. Choose **Vertex** modeling mode in the Toolbar.

Step 3. Choose **LOD/Place Morph Vertices**.

Step 4. Set the **Switch LODs During Placement** checkbox in the *Place Morph Vertices* dialog box. This automatically switches the Graphics view back and forth between the higher and lower LOD as you edit morph vertices.



Step 5. Select one or more vertices in the higher LOD.

Step 6. Click **Place Morph Vertex** in the *Place Morph Vertices* dialog box. The Graphics view automatically switches to the lower LOD.

Step 7. Middle-click a vertex in the lower LOD to make it the morph vertex of all selected vertices in the higher LOD. A red construction edge appears showing the path the vertex will follow as it morphs.

Step 8. Click **OK** to finish or **Next** to place another morph vertex. If **Switch LODs During Placement** is set, the Graphics view automatically switches to the higher LOD when you click **Next**.

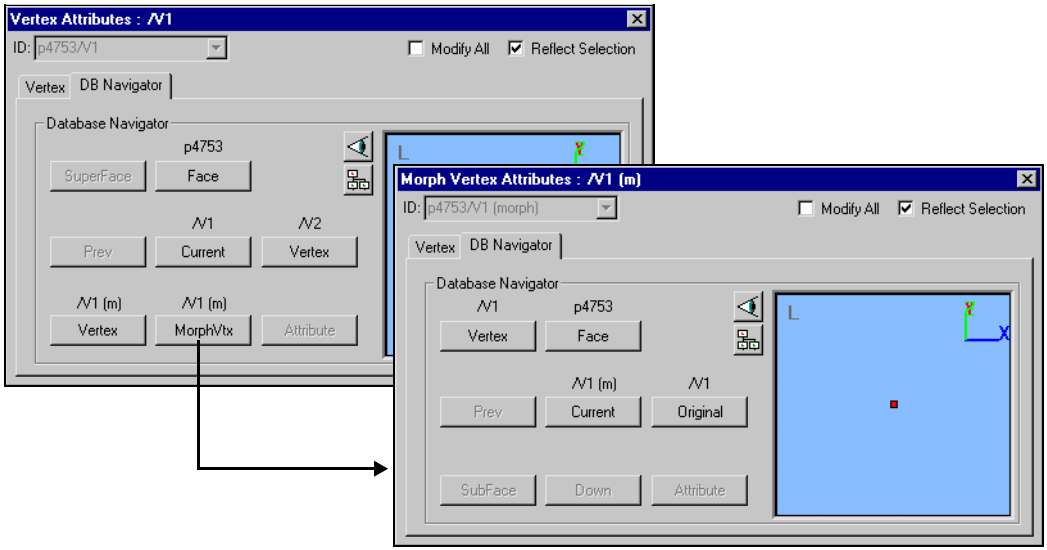
To View Morph Vertex Attributes

After morph vertices are assigned, each vertex in a higher LOD has a morph vertex as its child. Because the higher LOD has more vertices than the lower LOD, each lower LOD vertex usually becomes the morph vertex of several higher LOD vertices.

Step 1. Open the *Vertex Attributes* window of a vertex in the higher LOD.

Note: Vertices and morph vertices are not displayed in the Hierarchy view. To open vertex attributes, choose **Vertex** modeling mode from the Toolbar and double-click a vertex in the Graphics view.

Step 2. Click **Morph Vtx** in the DB Navigator panel.



Morph vertices have the same attributes as regular vertices (see “Vertex Attributes” on page 848).

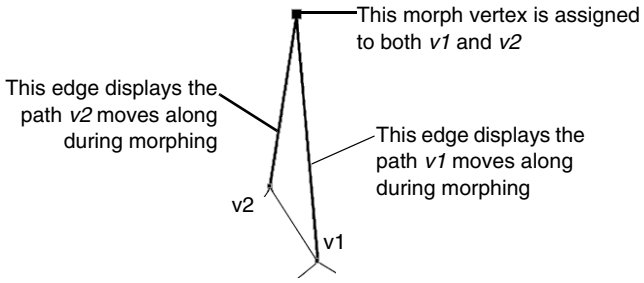
To Morph Between LODs

To view the effect of morphing after morph vertices are assigned, choose **LOD/More Detail** (Ctrl+M) or **LOD/Less Detail** (Ctrl+L).

You can also choose **LOD/Auto LOD** to turn on automatic LOD switching in Creator. In this mode, LODs morph according to distance from the LOD to the eyepoint as you change your eyepoint during editing.

To View a Morph Path

To view red construction edges showing the paths used for morphing, select one or more vertices and choose **LOD/Show Morph Vertices**.



To Delete Morph-Path Construction Edges

To delete morph-path construction edges, select them and choose **Edit/Clear Construction**.

You can delete all construction edges and vertices, including morph-path construction edges, by choosing **Edit/Clear Construction** with nothing selected. Be aware that all normal construction edges and vertices are also deleted when you do this.

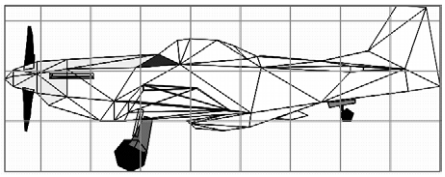
Because the edges and vertices displayed by **Show Morph Vertices** are constructions, they are not saved with the database. This means that editing or deleting them has no permanent effect.

To Remove Morph Vertices

To remove morph vertices, select an LOD and choose **LOD/Remove Morph Vertices**.

Generating an LOD Automatically

Generate LOD in the **LOD** menu automatically generates a lower LOD from a high-detail model. To to this, you will set up a three-dimensional grid around selected portions of the model, and then use the grid to define vertices in the lower LOD.



Each three-dimensional volume defined by the intersection of grid lines is referred to as one *rexel* (resolution element). The center of each rexel is its default *snap point*. To generate coordinates for the lower LOD, all vertices in a rexel that lie within a user-defined tolerance of its snap point are replaced by the snap point, and redundant vertices are culled.

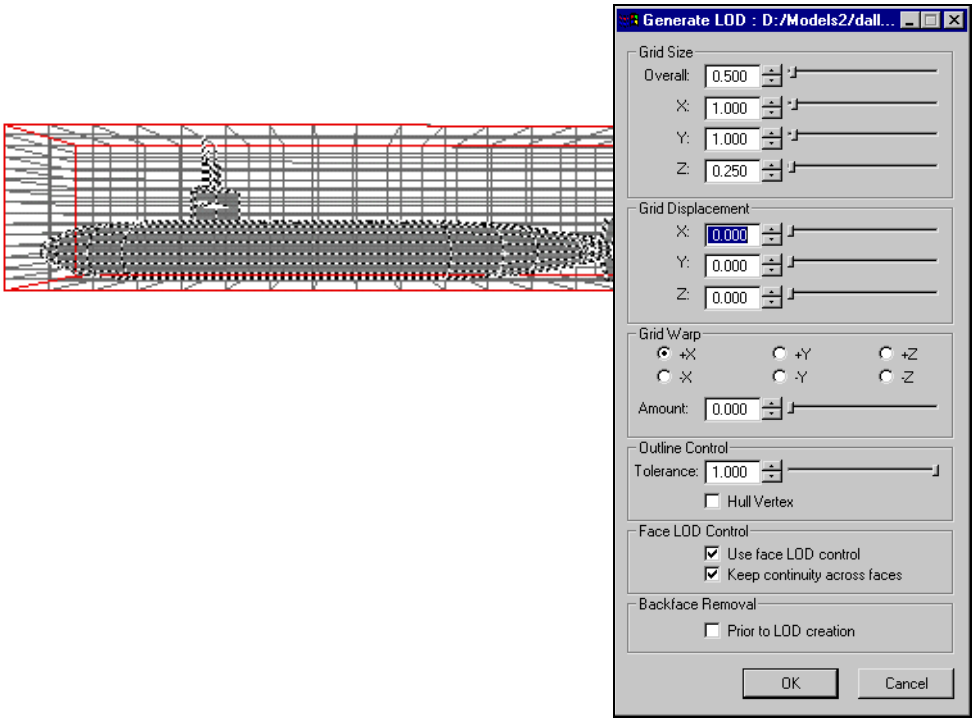
Although this discussion assumes you are processing an LOD node, you can also use **Generate LOD** with other kinds of nodes in the hierarchy, such as groups or objects. This lets you preserve important geometric details by processing pieces of a model separately.

To Generate an LOD

Step 1. Select an LOD node. (To process a model in pieces, select a group or object belonging to an LOD, and then repeat this procedure for each additional piece of the model.)

Step 2. Choose **LOD/Generate LOD** and set the parameters in the *Generate LOD* dialog box (see "Generate LOD Parameters" on page 723).

Step 3. Use **Grid Size**, **Grid Displacement**, and **Grid Warp** to position important features of the model as closely as possible to the snap points of the grid. If this is not possible, select and process each important piece of the model separately.



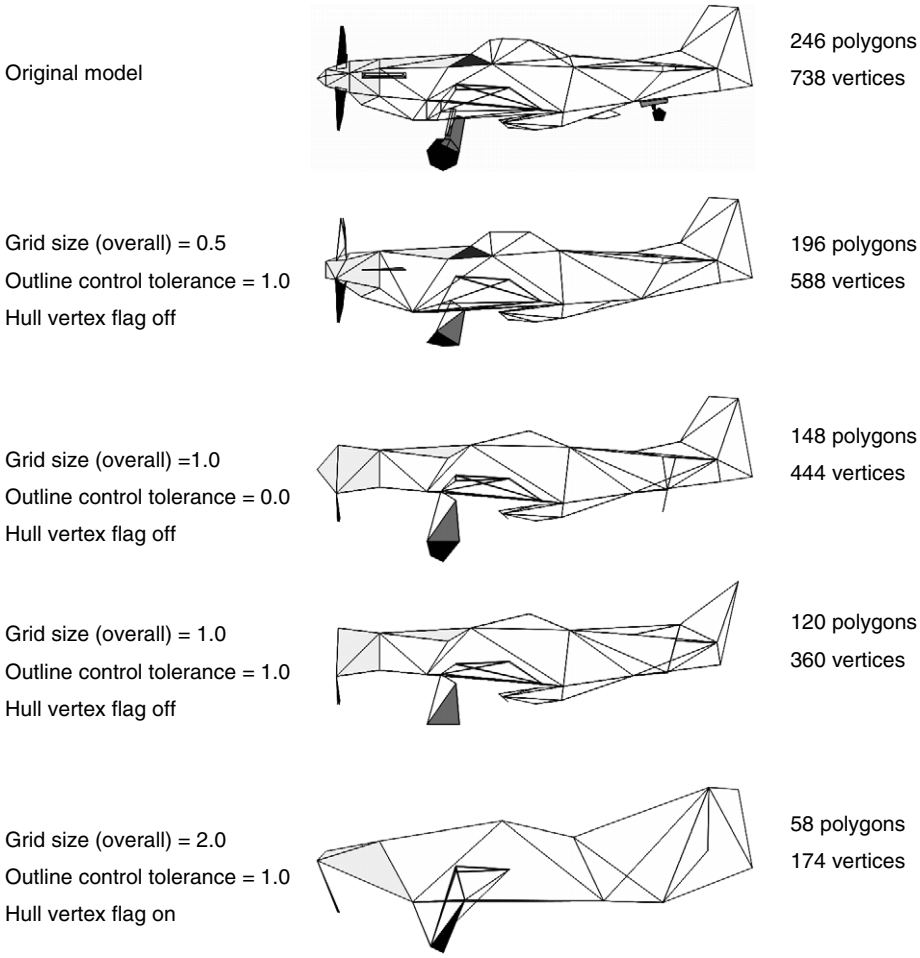
Step 4. Click **OK** in the *Generate LOD* dialog box. Creator processes the selected portion of the database and inserts the new geometry as closely as possible to the original node in the hierarchy.

Step 5. Evaluate the results of **Generate LOD**. Look carefully at important areas of the geometry. For example, if you are processing an airplane, was the propeller preserved or discarded appropriately? You may need to regenerate the model several times with different parameters to get the best results (see "Tips for Working With Generate LOD" on page 725).

Step 6. If the model was processed in pieces, open the Hierarchy view when you are done, create an LOD node, and attach the generated pieces to the new LOD node.

Generate LOD Parameters

The following figure shows the effect of varying Generate LOD parameters.



Varying Parameters While Generating a Level of Detail

Grid Size

Controls the spacing between grid lines, and so determines the complexity of the new LOD. A small **Grid Size** produces a model that more closely resembles the original geometry than a large **Grid Size**.

Grid Displacement

Controls the offset of the grid relative to the *convex hull* of the model. This value ranges from 0.0 to 1.0, with 1.0 equal to the distance between a consecutive pair of grid lines. Because the default snap point of each rexel is its center, you should position points of interest on a model as closely as possible to the center of each rexel.

Grid Warp

Distorts the grid in one or more dimensions. Use this to make the grid coarser around portions of the model where you need to preserve less detail.

Outline Control Tolerance

Controls the complexity of the new geometry. Generally, if the distance from a vertex to the corresponding rexel's snap point is less than the **Tolerance**, the vertex is replaced by the snap point. Vertices that lie outside of the **Tolerance** are more likely to be preserved. The range of this value is 0.0 to 1.0, where 1.0 is the distance between a pair of consecutive grid lines.

Hull Vertex

Preserves vertices lying along the convex hull of the model. This helps preserve the shape of the model by making hull vertices the snap points for the rexels in which they lie.

Use Face LOD Control

Preserves all vertices that are attached to faces in which the **LOD Control** face attribute is set. Use this attribute to preserve specific areas of interest in the original model.

Keep Continuity Across Faces

Matches the vertices of contiguous faces. This matches the edges of adjoining polygons and prevents discontinuity in the resulting model.

Backface Removal: Prior to LOD Creation

Discards backfacing polygons before generating an LOD.

Tips for Working With Generate LOD

It is usually necessary to run **Generate LOD** several times, experimenting with the grid and tolerance controls, to find a good compromise between the number of polygons and the aesthetic value of the new model.

Although the absolute limit is higher, there is a practical limit on this algorithm of about 1,000 vertices in the selected geometry (more or less, depending on the speed of your computer). If you are close to this limit, process the model in pieces. You can display the number of vertices in the selected geometry by choosing **Info/Statistics** (Ctrl+?).

This algorithm should provide you with a good start, but most automatically generated LODs require some manual correction of geometry and hierarchy. You may also need to edit the attributes of the new model.

Do not forget to set the switching distances in new LOD nodes.

Exercise: Working With LODs

In this exercise you will work with an existing file that contains two LODs, set up dynamic morphing between the LODs, and use the higher-detail model to generate a new LOD.

As you work through the exercise, keep in mind that it is a demonstration of LODs, and that detailed reference information was presented earlier in the chapter.

In this exercise, you will learn how to:

- View and edit LODs
- Control switching distance
- Set up a morph between two LODs
- Create a new LOD with **Generate LOD**

You can use the **Browse** buttons in the Help window to move from one part of the exercise to the next.



View Two LODs

Step 1. Choose **File/Open** (Ctrl+O) and select the example file `DallasEx.flt` in the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory.

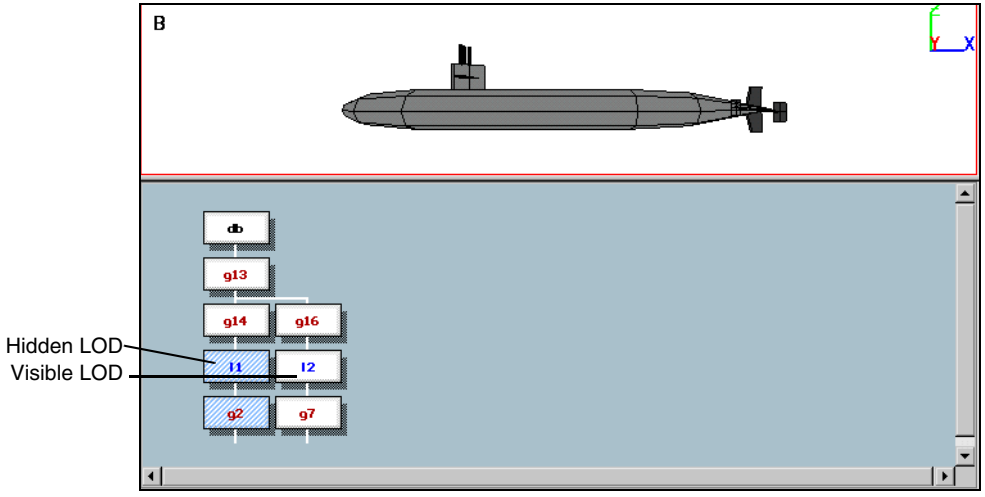
Note: If you did not install the exercise files when you installed Creator, you can open this file directly from the Creator installation CD.

Step 2. Choose **File/Save As** and save a working copy of the file.

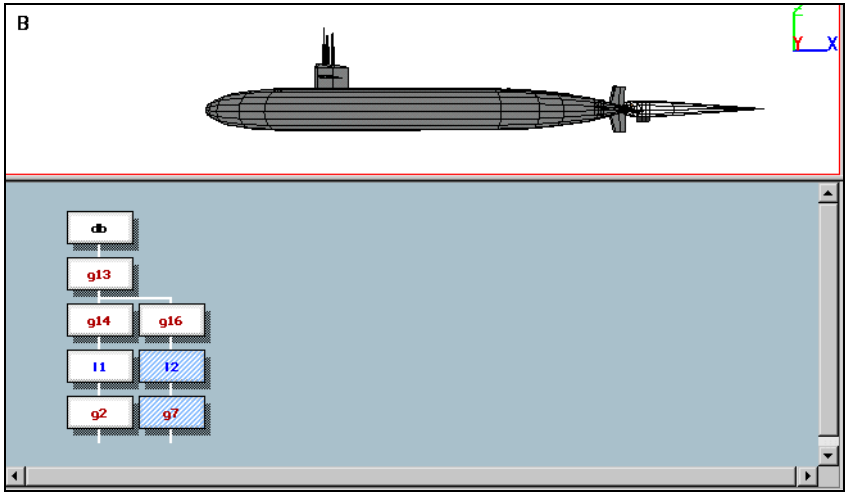
Step 3. Drag the bottom Split bar up to view both the geometry and the hierarchy of the database.

Step 4. Adjust the eyepoint by setting **Y**aw = 0, **P**itch = 0, and **R**oll = 0.

Step 5. Observe that the lower detail model (**l2**) is visible and the higher detail model (**l1**) is hidden. (Nodes filled with diagonal blue lines in the Hierarchy view are hidden in the Graphics view.)



Step 6. Choose **LOD/More Detail** (Ctrl+M) and **LOD/Less Detail** (Ctrl+L). Observe that the submarine switches back and forth between a simpler and more detailed model.

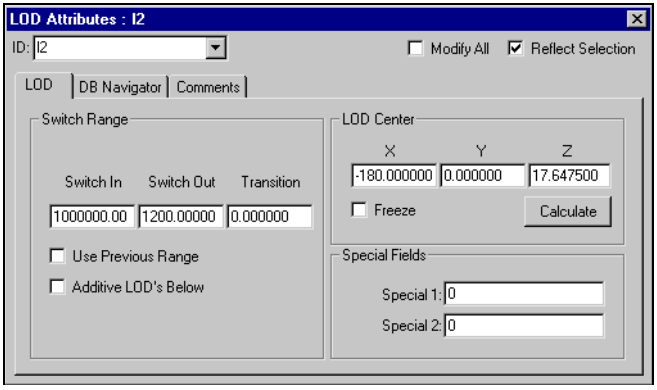


View LOD Switch Attributes

This exercise demonstrates LOD switch in and switch out distances, and how to set them in the *LOD Attributes* window. For more detailed information, see "Displaying and Modifying LOD Attributes" on page 712.

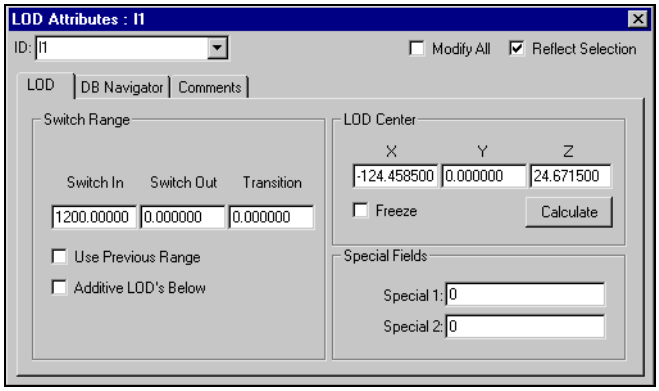
Step 1. Double-click **l2** to open its *LOD Attributes* window.

The **Switch In** attribute is 1,000,000. In realtime, as the eyepoint travels toward the LOD, the geometry controlled by this LOD first appears when the eyepoint reaches 1,000,000 meters (or whatever other system of units is being used by the realtime system) from center of the model. The **Switch Out** attribute is 1,200. When the eyepoint reaches this distance from the center of the model, it is turned off and the geometry in the next highest LOD appears.



Step 2. Double-click **l1** to open its *LOD Attributes* window.

The **Switch In** attribute is 1,200. This matches the **Switch Out** attribute of **l2**, indicating that this is a higher-detail replacement for **l2**.



Step 3. Close the *LOD Attributes* window.

Set Up Morphing Between LODs

Morphing is the gradual transition between one LOD and another. In morphing, an extra set of vertices is added to the higher LOD to allow the two LODs to switch smoothly back and forth.

For reference material, see “Morphing Between LODs” on page 716.

Step 1. Select both LODs (**I1** and **I2**).

Step 2. Choose **LOD/Find Morph Vertices**.

Step 3. Choose **LOD/More Detail** (Ctrl+M) and **LOD/Less Detail** (Ctrl+L). Observe that the low and high detail submarines now merge smoothly back and forth.

Step 4. Select the higher LOD (**I1**) and choose **LOD/Remove Morph Vertices** to restore the model to its original state.

Generate a Lower Detail Model

This example demonstrates the use of **Generate LOD** to produce a lower detail model from existing geometry. However, the result is not production quality. For information

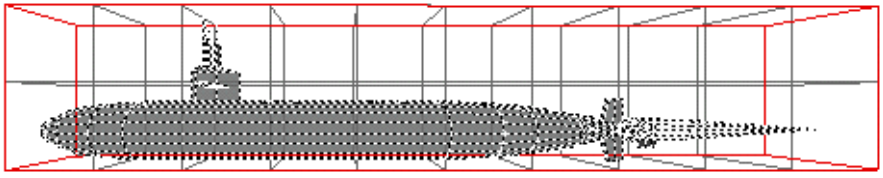
on optimizing **Generate LOD** for production, see "Generating an LOD Automatically" on page 720.

Step 1. Make sure the high LOD (**l1**) is visible in the Graphics view. You can do this by choosing **LOD/Most Detail** (Ctrl+Shift+M).

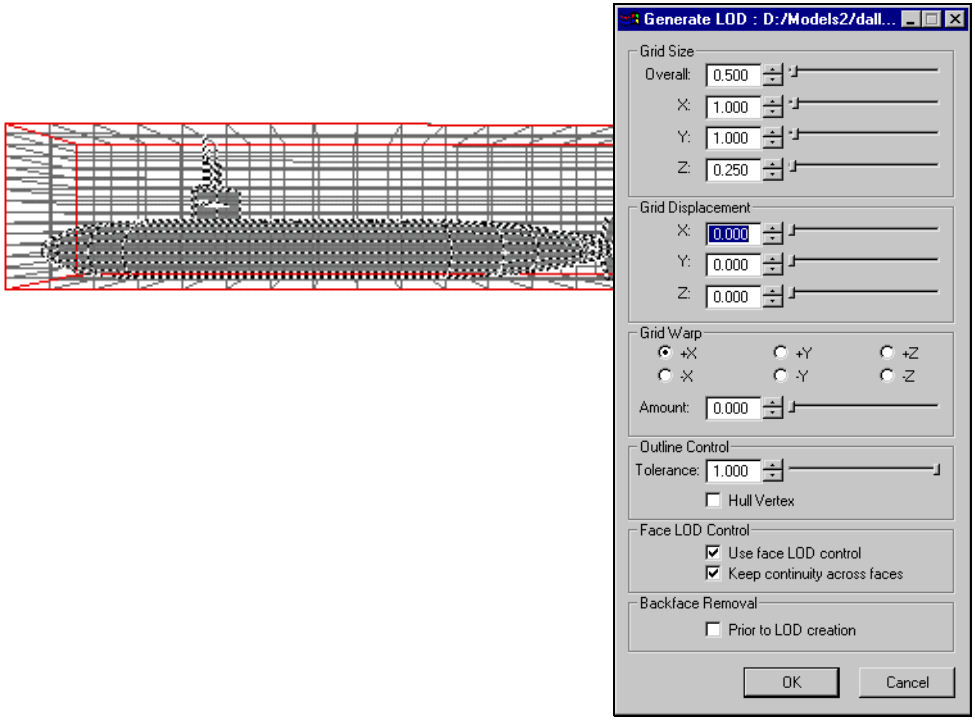
Step 2. Choose **LOD** modeling mode in the Toolbar.

Step 3. Select the high LOD (**l1**).

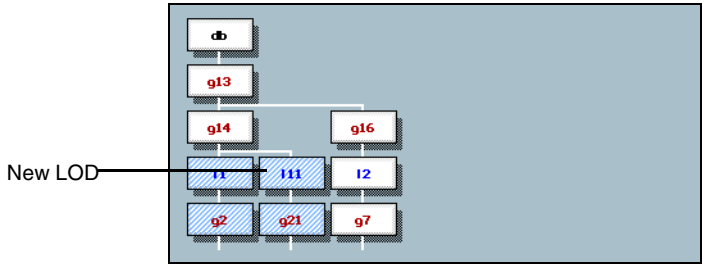
Step 4. Choose **LOD/Generate LOD**. Observe that a three-dimensional grid appears around the submarine in the Graphics view. This grid controls the amount of detail in the new model.



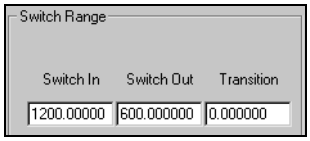
Step 5. In the *Generate LOD* dialog box, set **Grid Size: Overall** = 0.5 and **Grid Size: Z** = 0.25. Observe that the grid surrounding the model becomes much finer.



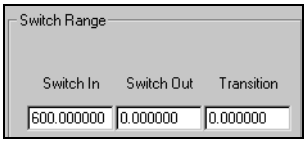
Step 6. Click **OK**. A new LOD node appears in the Hierarchy view.



Step 7. Double-click the new LOD node in the Hierarchy view to open its *LOD Attributes* window. Set **Switch In** = 1200, and set **Switch Out** = 600. (The software may have done this for you automatically.)

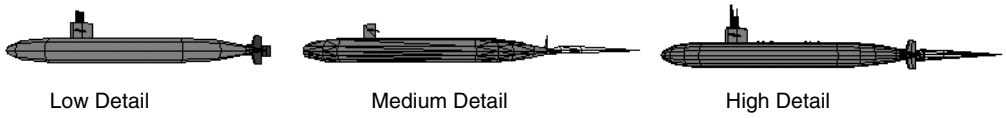


Step 8. Double-click the original LOD node (**l1**), and set its **Switch In** = 600 and **Switch Out** = 0.0. (The software may have done this for you automatically.)

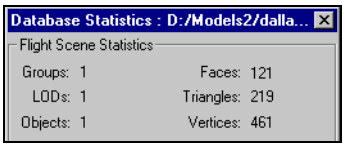


Step 9. Close the *LOD Attributes* window.

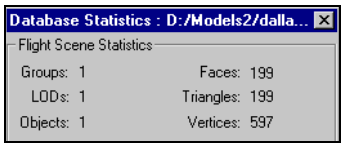
Step 10. Choose **LOD/Less Detail** (Ctrl+L) several times, and then choose **LOD/More Detail** (Ctrl+M) several times. Observe that the medium LOD now switches in between the two original LODs.



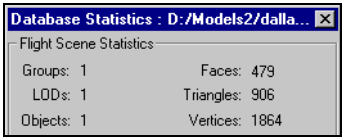
Step 11. Use **More Detail** and **Less Detail** to display each LOD again. As each LOD becomes visible, select it and choose **Info/Statistics** (Ctrl+?) to view information about the number of faces the LOD contains. Observe that the number of faces in the medium LOD is substantially lower than in the high LOD.



Low Detail



Medium Detail



High Detail

21 Making Things Move

Creator has two features that can add movement to your models: Degrees of Freedom and Animation.

A *Degree of Freedom (DOF)* is a type of node that you can insert in the database to add movement to the geometry below it. A DOF establishes a local coordinate system, and the geometry it controls moves around the axes of the coordinate system.

Animation is an attribute of Group nodes. Children of an animated Group node form the "frames" of an animation, and are displayed one after another to provide the appearance of movement.

Animation requires more modeling effort and more realtime memory than DOFs, because the geometry that comprises each frame of the animation must be modeled and loaded into memory individually. DOFs are easier to model, but they require programming effort to make them work in the realtime system.

Degrees of Freedom

A *DOF node* controls all the geometry in the tree below it. The *DOF attributes* associated with this node define a range of translational and/or rotational motion for the geometry attached to the DOF node. For example, a DOF can specify the angle through which a door swings or the horizontal or vertical distance that a window slides, so that the door or window stays in its frame and moves in a logical way.

The range of motion is specified relative to a local coordinate system, and the transformations it contains are applied to all descendants of the DOF node. This ensures that the effects of multiple degrees of freedom accumulate naturally in the database hierarchy. Consider the example of a hand attached to an arm. When you move the arm, the hand also moves because its geometry is a descendent of the arm DOF. The wrist of the hand bends and rotates relative to the position of the arm because the wrist is controlled by a DOF node that specifies its range of motion, and the wrist DOF node is a descendant of the arm's DOF node.

Overview: Creating Degrees of Freedom

To add motion using DOF nodes, you must:

1. Create the DOF node using the **Create DOF** tool.
2. Select the geometry you want to add movement to and attach it to the DOF.
3. Establish a local DOF coordinate system using **Local-DOF/Position DOF**.
4. Set limits on the movement of the geometry using **Local-DOF/Set DOF Limits**. These limits reference the local coordinate system.
5. Check the range of movement using the **Animate** or **Exercise** functions of the DOF. To check movements of more than one DOF node simultaneously, use the **DOF Viewer**.

Creating DOF Nodes

To create a DOF node:

Step 1. Set the database parent. A DOF node is equivalent to a Group node, so the database parent must be a Group, LOD or DOF node.



Step 2. Open the **Create** toolbox.



Step 3. Click **Create DOF**. The new DOF node appears in the Hierarchy view.

Displaying DOFs in the Graphics View

In the Graphics view, a DOF is represented by a set of coordinate axes that show the local origin of the node. To display DOF axes:

Step 1. Click the View panel button in the Toolbar.

Step 2. Click anywhere in the Graphics view to activate the **View** panel.



Step 3. Click **Draw DOF Axes**.

Selecting DOFs

To select a DOF in the Graphics view, set the modeling mode to **DOF**, and then click the geometry attached to the DOF node.

To select a DOF in the Hierarchy view, click the DOF node. A white border appears around the DOF node. If you select the geometry attached to the DOF, a blue border appears around the DOF node.

Positioning DOF Axes

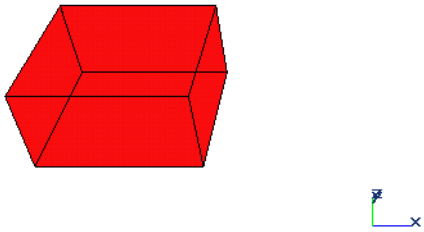
You position DOF axes to define a local coordinate system for the part of the model that is attached to the DOF. To position the DOF, you choose an origin, *x*-axis alignment point, and *y*-axis alignment point. The locations you choose for the origin and alignment points in this procedure depend on the movement you want to establish for your model. After you position the DOF axes, you will set DOF limits that specify the range of motion for the DOF.

When you create a DOF, its axes are automatically positioned at the database origin.

To reposition DOF axes:

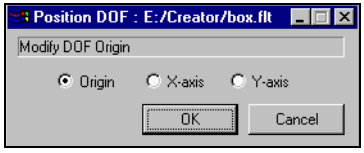
Step 1. Make sure **Draw DOF Axes** is active in the View panel. The DOF axes appear in the Graphics view.





Step 2. Select the DOF node in the Hierarchy view.

Step 3. Choose **Local-DOF/Position DOF**. The *Position DOF* dialog box appears. **Origin** is selected.



Step 4. In the Graphics view, middle-click a vertex or edge in the geometry attached to the DOF to set the origin of the local coordinate system.

Step 5. Select **X-axis** in the *Position DOF* dialog box, and then middle-click a vertex or edge in the Graphics view to set the alignment of the *x*-axis.

Step 6. Click **Y-axis** and then middle-click a vertex or edge in the Graphics view to set the alignment of the *y*-axis.

Step 7. Click **OK**.

Setting DOF Limits

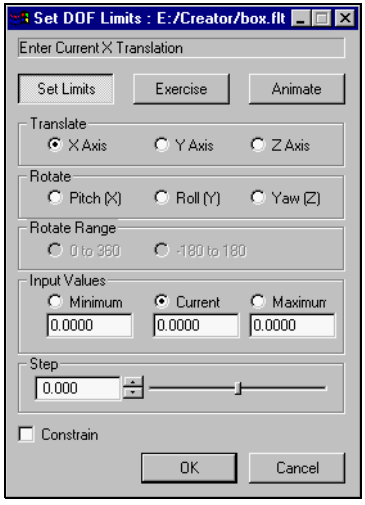
You assign movement to the geometry attached to a DOF by setting DOF limits. *Translational limits* specify the distance, in database units, that a DOF is permitted to move relative to its local coordinate system. *Rotational limits* specify the range, in degrees, through which the DOF can rotate around each axis. The term *pitch* refers to rotation around the *x* axis, *roll* refers to rotation around the *y* axis, and *yaw* refers to rotation around the *z* axis. Together, these six *transformations* allow you to specify a full range of motion in three dimensions.

Assigning Transformations to DOFs

To assign a set of transformations to a DOF node:

Step 1. Select a DOF node.

Step 2. Choose **Local-DOF/Set DOF Limits**. The *Set DOF Limits* dialog box appears, with the **Set Limits** button already selected.



Step 3. Select a transformation to edit by clicking one of the **Translate** or **Rotate** buttons. This displays the unique **Step** and **Input Values** associated with the transformation.

Step 4. Enter **Input Values** for the selected transformation. In the Graphics view, the range you specify is represented by a red construction edge for translations, or by an arc for rotations. A green vertex represents the **Current** input value. You can edit the construction points (which changes the **Input Values**) by clicking new points in the Graphics view.

Step 5. Specify the **Step** value of the selected transformation.

Step 6. Select additional transformations, and set their **Input Values**, as needed.

Step 7. Set the **Constrain** checkbox to force the DOF to stay within the limits you are specifying. If **Constrain** is cleared, the limits are copied into the DOF attributes but do not strictly control its range of motion (see "Changing DOF Attributes" on page 744).

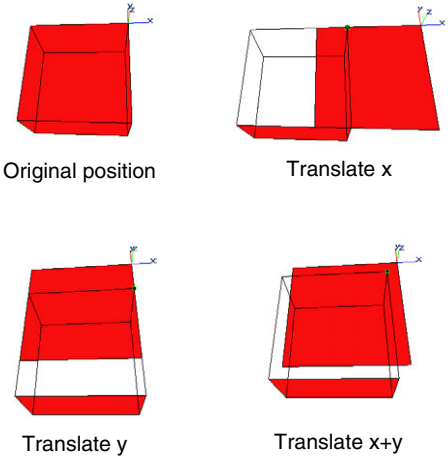
Set DOF Limits Parameters

The following parameters define the movement of the geometry associated with the DOF.

Translate

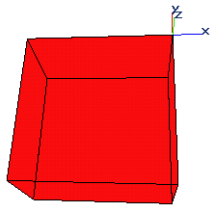
Redefines the position of the DOF's geometry. The *direction* of the translation depends on the axis or axes you specify. The *distance* the geometry moves depends on the **Input Values** you enter for that axis. The *speed* of the movement depends on the **Step** value you specify.

The initial **Input Values** are 0.00000 for all the axes, which means that no movement is allowed. You set values for the *x*-axis, *y*-axis, *z*-axis, or any combination of the three to specify a range of motion in any direction.

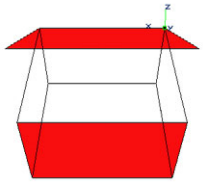


Rotate

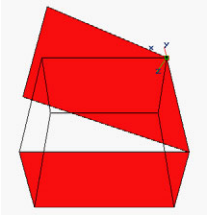
Defines motion around an axis or a combination of axes. To rotate geometry around an axis, you specify a **Rotate Range**, **Input Values**, and a **Step** value. To simulate rotation around a point, set these parameters for more than one axis.



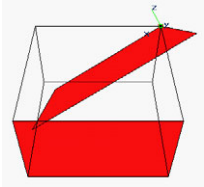
Original position



Rotate x



Rotate y

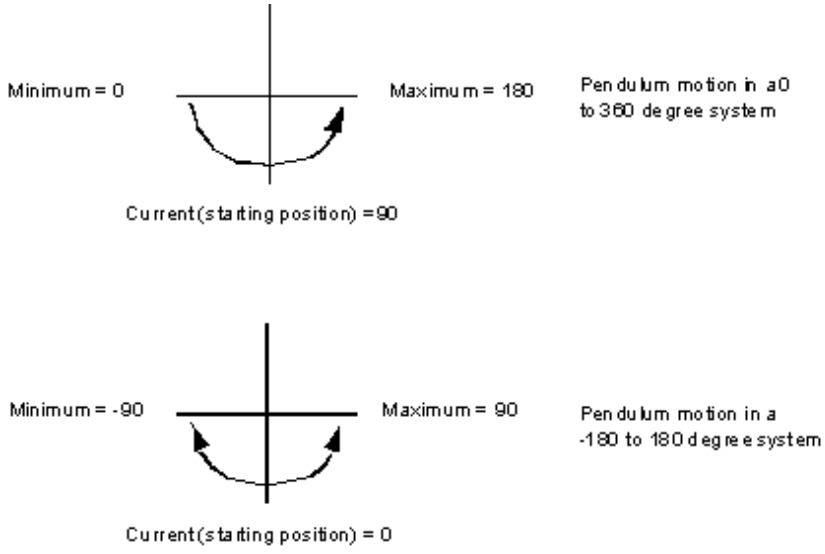


Rotate x+y

Rotate Range

Sets the range of values for the **Minimum**, **Maximum**, and **Current Input Values** applied to a **Rotate** transformation set for a DOF. If you want the starting position to be zero, for example, you might choose a **Rotate Range** of **0 to 360** if the rotation is in one direction only, and choose **-180 to 180** when the movement is in two directions.

The following illustration demonstrates how **Minimum**, **Maximum** and **Current Input Values** might be set for each **Rotate Range**.



The DOF **Rotate Range** you choose may depend on your realtime system. MultiGen's SmartScene system, for example, does not accept negative numbers. If your realtime system *does* accept negative numbers, the **Rotate Range** depends on your personal or company preference.

Input Values

The DOF **Input Values** specify the distance, in database units, that the DOF geometry moves in a **Translate** transformation, or the rotation, in degrees, that the DOF geometry moves in a **Rotate** transformation.

- **Minimum** defines the lower limit of the transformation.
- **Current** defines the position of the DOF geometry at the time the limits are set.
- **Maximum** defines the upper limit of the transformation.

Step

Defines the amount to be added or subtracted from the **Current** position of the DOF as it is exercised or animated within its range.

Constrain

Forces the DOF to stay within the limits you specify. If **Constrain** is disabled, the limits are copied into the DOF attributes but do not strictly control its range of motion.

Changing DOF Attributes

The DOF limits you set also appear in the *DOF Attributes* window. To view or edit the attributes, double-click the DOF node, or select the DOF node and choose **Attributes/ Modify Attributes** (Ctrl+=). See "DOF Attributes" on page 865 for a description of these attributes.

Exercising a DOF

To *exercise* a DOF is to drag it along the path defined by one of its transformations. To do this, click **Exercise** in the *Set DOF Limits* dialog box, select **Translate** or **Rotate**, and use the mouse to drag the DOF along the construction edge (or arc) displayed in the Graphics view.

If the **Step** value is zero, the DOF moves continuously along the edge. Otherwise, the DOF is exercised by successively adding or subtracting the value of **Step** as you drag the mouse.

Animating a DOF

Animating a DOF applies all translations and rotations defined for the DOF simultaneously. To do this, click **Animate** in the *Set DOF Limits* dialog box. For each transformation with a non-zero range, Creator starts at the **Current** position and adds or subtracts the corresponding **Step** to move the DOF within the range. As this implies, transformations associated with a zero value of **Step** are ignored. To stop the animation, click **Set Limits** or **Exercise**.

Constraining DOF Limits

To force a degree of freedom to stay within its defined limits, select the node and choose **Local-DOF/Constrain DOF**. This turns on all **Constrain** checkboxes in the attributes of the DOF (see "Changing DOF Attributes" on page 744).

Clearing DOF Limits

To allow an unlimited range of motion in a DOF node, select the node and choose **Local-DOF/Un-Constrain DOF**. This clears all **Constrain** checkboxes in the attributes of the DOF, but does not affect the DOF limits that are stored in the attributes of the node (see "Changing DOF Attributes" on page 744).

Freezing a DOF

To freeze the geometry of a DOF in its current position, select it and choose **Local-DOF/ Freeze DOF**. This copies the value of each **Current** attribute in the DOF into the corresponding **Minimum** and **Maximum** fields, sets each **Step** attribute to 0, and enables all **Constrain** checkboxes in the DOF attributes.

Using the DOF Viewer

The **DOF Viewer** allows you to preview several Degree of Freedom (DOF) nodes simultaneously to see how they interact together. For each DOF, the **Translate**, **Rotate**, and **Scale** attributes are all considered.

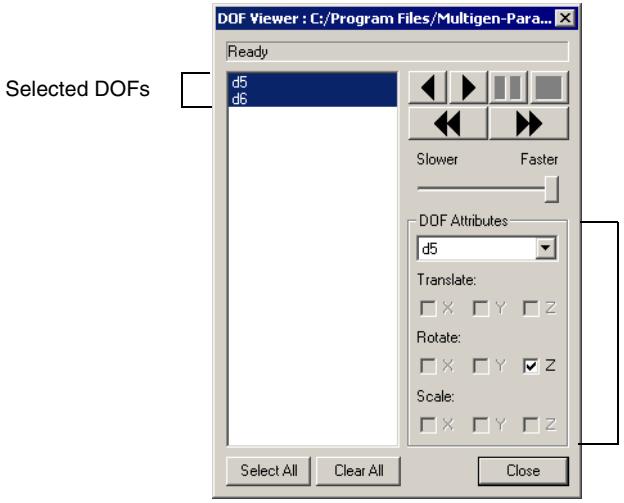
When you start the **DOF Viewer**, all currently selected DOFs are included for previewing. If no DOFs are currently selected, all DOF nodes (with non-zero step increments) in the database are eligible for previewing. You can select and deselect the attributes for individual DOF nodes while previewing with this tool. DOF movement is controlled using play, stop, and step buttons.

Previewing DOFs

To preview DOFs with the **DOF Viewer**:

Step 1. Select specific DOF nodes either in the Hierarchy or Graphics view.

Step 2. Choose **Local-DOF/DOF Viewer**. The *DOF Viewer* dialog box appears with the selected DOFs listed.



Step 3. The attributes (translation, rotation, and scale about the X, Y, Z axes) that were set when the DOFs were created appear in **DOF Attributes**. For each DOF node, you can deselect them for viewing purposes.



Step 4. Click the **Play Forward** and **Play Backward** buttons to move the DOF.



Step 5. Click the **Step Forward** and **Step Backward** buttons to move the DOF through its step increments.



At any time, you can click the **Pause** or **Stop** buttons to pause or stop the DOF movement. You can also control the speed of movement by dragging the slider.

Step 6. Click **Select All** to select all DOFs in the list, or **Clear All** to deselect all DOFs in the list.

Exercise: Creating a DOF

In this exercise, you will create a DOF to move a box top up and down. Before you begin, you may want to print this topic to make it easier to follow the instructions as you work with Creator.

This exercise is presented in the following parts:

"Getting Started" on page 747

"Create a Box" on page 747

"Create the DOF Hierarchy" on page 749

"Position the DOF Coordinate System" on page 751

"Set the DOF Limits" on page 753

"Exercise the Limits" on page 755

"Animate the Limits" on page 756

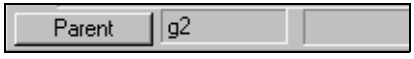
"Modify the DOF Position" on page 757

Getting Started

Step 1. Open a new file.

Step 2. Split the window horizontally to view both the Graphics and Hierarchy views.

Step 3. Notice that **g2** is already set as the parent.



Create a Box

Step 1. Choose **Object** modeling mode.



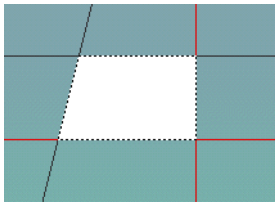


Step 2. Open the **Face** toolbox.



Step 3. Click **Rectangle**.

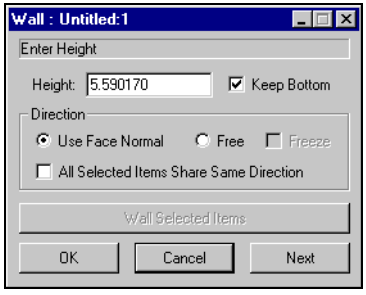
Step 4. Enter two points on the grid to create a square, and then click **OK**.



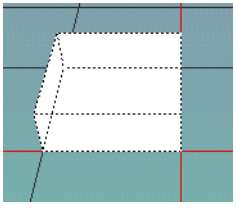


Step 5. Open the **Geometry** toolbox and click **Wall** (Shift+W).

Step 6. Click **OK** to construct a wall at the default **Height**. Be sure **Keep Bottom** is set.



You should now have a box that looks similar to this:



The database structure should look like the following illustration. (Middle-click in the Hierarchy view to display the faces.)



Step 7. Save your file (Ctrl+S) as box.flt.



Step 8. Click the **View Panel** button in the Toolbar.

Step 9. Click anywhere in the Graphics view to activate the View controls, and then click the **Grid** button to turn off the grid. (This will make it easier to see your axes placement in the next part of the exercise.)



Create the DOF Hierarchy

All objects and faces that are going to move must be children of the DOF node that controls their movement. In this procedure, you will create a DOF node, and make the box top a child of the DOF node.

Step 1. Select **g1** and click the **Parent** button at the bottom of the window to set **g1** as the new parent.

Step 2. Open the **Create** toolbox and drag it away from the Toolbox caddy to keep it open.



Step 3. Click **Create DOF**. The new DOF node (**d1**) appears as a child of **g1**.

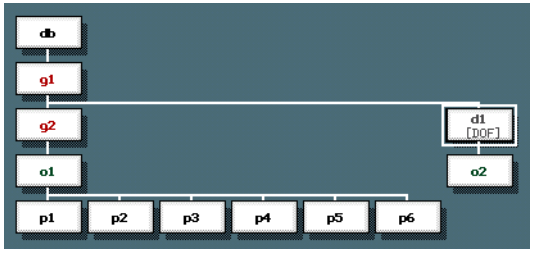




Step 4. Select **d1** and click the **Parent** button at the bottom of the window to set **d1** as the new parent.

Step 5. Click **Create Object** in the **Create** toolbox. The new object appears as a child of the DOF node. Close the **Create** toolbox.



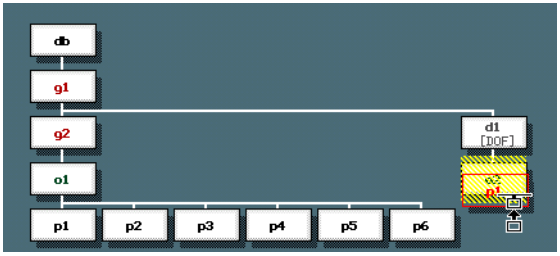


You create the new object node to avoid attaching a face node to the DOF node in Step 7. A DOF node has the same hierarchy value as a group node. Some realtime systems will discard face nodes that are attached directly to group nodes or their equivalents in the hierarchy.

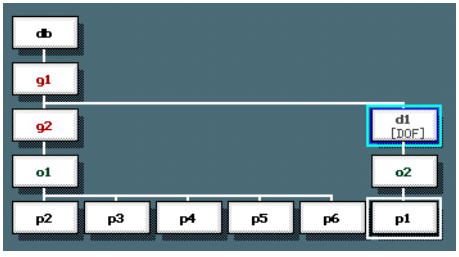
Step 6. Set the modeling mode to **Face** using the menu in the Toolbar.

Step 7. Select the top face of the box. The node (**p1**) is now selected in the hierarchy view.

Step 8. Drag **p1** onto the object node that is a child of the DOF node (**o2**), and release the mouse button.



P1 is now a child of **02**.



Step 9. Save your file (Ctrl+S).

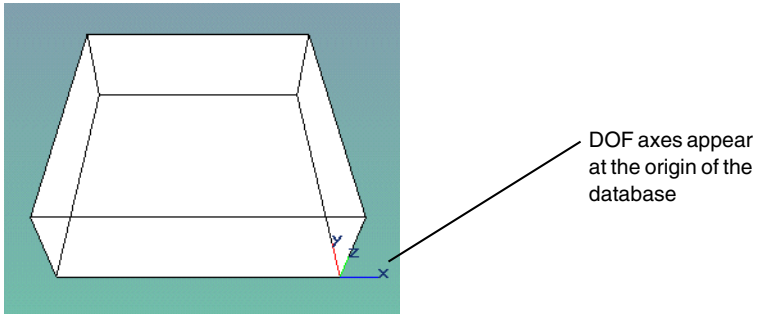
Position the DOF Coordinate System

This part of the exercise establishes the local coordinate system for the DOF. All movement controlled by the DOF node is based on this local coordinate system.

Step 1. Click anywhere in the Graphics view to activate the View panel.

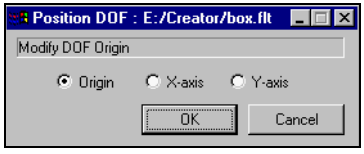
Step 2. Click **Draw DOF Axes** to display the DOF axes in the Graphics view. The initial position for any DOF you create is at the database origin (0, 0, 0).



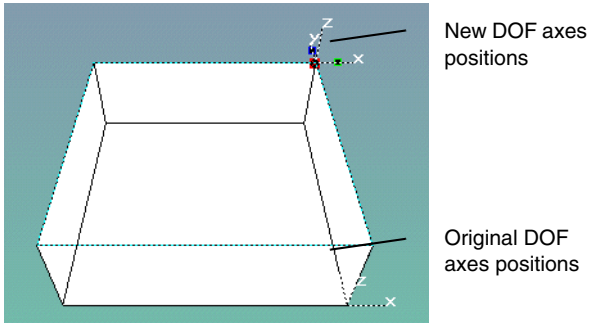


Step 3. Select the DOF node **d1**. Notice that the DOF node is outlined in white and that the DOF axes display turns white when the DOF is selected.

Step 4. Choose **Local-DOF/Position DOF**. The *Position DOF* dialog box appears.

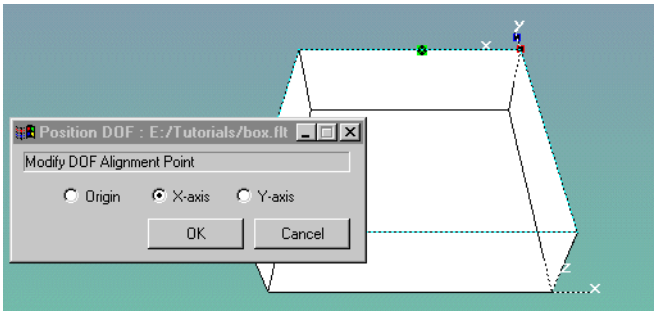


Step 5. Middle-click the upper-right corner of **p1**. This positions the local origin of the DOF at the corner vertex.



Notice that an axis display remains at the original position, to show where the axes would be positioned if you canceled the **Position DOF** operation.

Step 6. In the *Position DOF* dialog box, select **X-axis** and then, in the Graphics view, middle-click the back edge of **p1** to position the *x*-axis along that edge. (Middle-clicking ensures that the axis is aligned with the edge.)

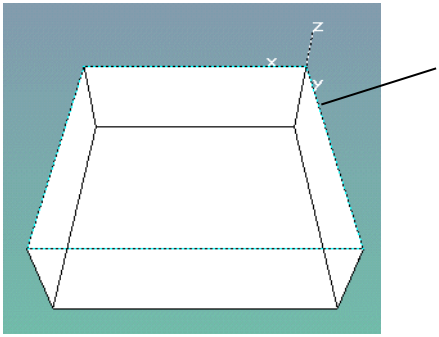


Step 7. Select **Y-axis** and middle-click the right edge of **p1** to position the *y*-axis of the DOF, and then click **OK**. Notice that the *z*-axis is still pointing upward. You have simply rotated the *x*- and *y*-axes so that the positive values are now along the edge of the box top.

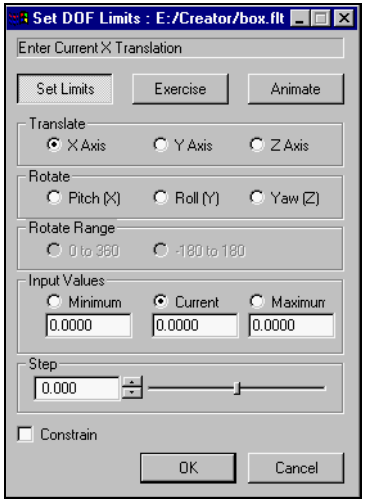
Step 8. Save the file (Ctrl+S).

Set the DOF Limits

In this exercise, you will set limits to make the box top open and close. To do this, the top of the box will rotate around the *y*-axis, which is positioned along one edge of the box top. Movement around the *y*-axis will be limited to a range of motion between the minimum and maximum angles you define. Movement along the *x*-axis and *z*-axis will be disallowed by constraining the movement to zero along these axes.



Step 1. With the DOF node still selected, choose **Local-DOF/Set DOF Limits**. The *Set DOF Limits* dialog box appears.



Step 2. In the **Rotate** area, select **Roll (Y)**.

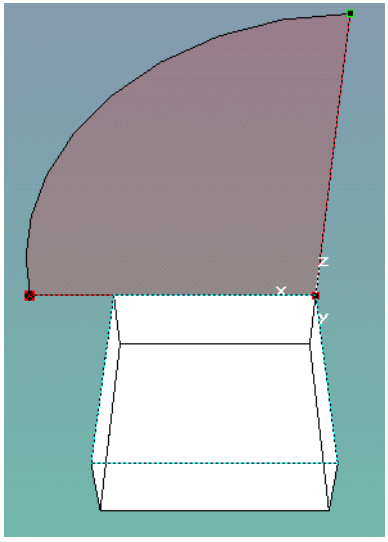
Step 3. For the **Input Values**, set **Maximum** to 90.

Step 4. Set **Step** to 1.

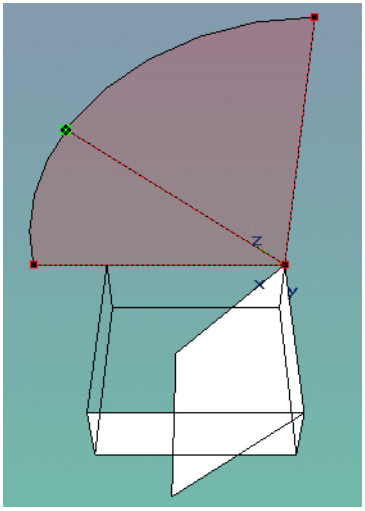
Step 5. Set the **Constrain** checkbox. Do not click **OK** yet, because you are going to test the new limits in the same dialog box.

Exercise the Limits

Step 1. Click **Exercise** in the *Set DOF Limits* dialog box. The Graphics view shows the range of motion you specified when you set the DOF limits.

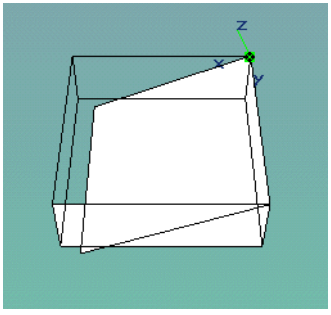


Step 2. Drag the pointer in the Graphics view to see the movement of your model.



Animate the Limits

Step 1. In the *Set DOF Limits* dialog box, click **Animate**. Watch as the box top goes through the specified range of motion.



Notice that the box top has the correct range of motion, but the rotation is in the wrong direction. This must be fixed by modifying the position of the DOF's y-axis, which we will do in the next part of the exercise.

Step 2. Click **OK** to accept the current limits and close the dialog box.

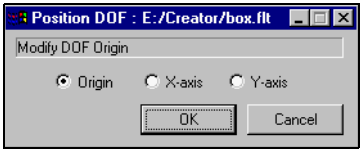
Step 3. Save the file (Ctrl+S).

Modify the DOF Position

The box top in the example moves inward. We want the top to open up and out, so we have to reverse the positive direction of the axis around which the top rotates; in this case the y-axis. To do this:

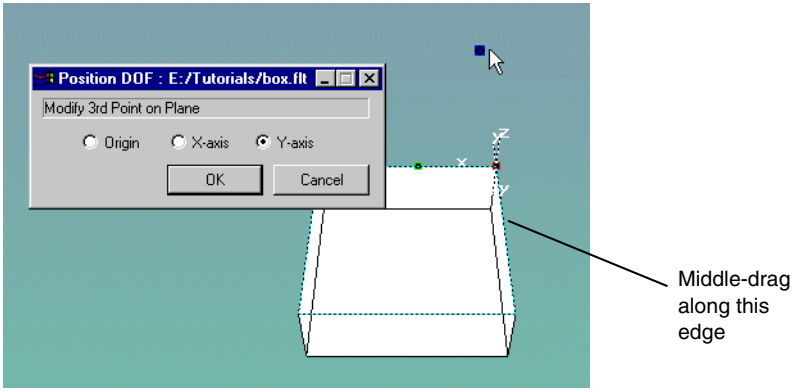
Step 1. In the Hierarchy view, select the DOF node.

Step 2. Choose **Local-DOF/Position DOF**. The **Position DOF** dialog box appears.



Step 3. Select **Y-Axis**.

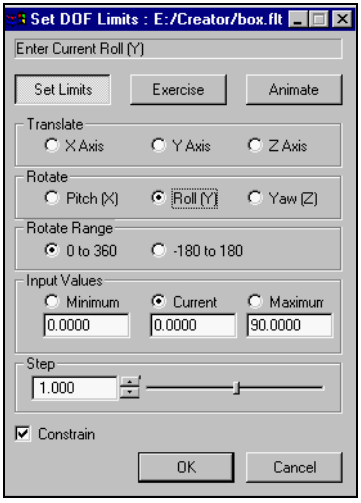
Step 4. Middle-click the edge of the box top along the y-axis and drag the pointer towards the back of the box, past the x-axis. This reverses the positive direction of the *y*-axis. (Dragging with the middle mouse constrains the movement to the edge you selected, ensuring that the repositioned *y*-axis will still be aligned with the edge.)



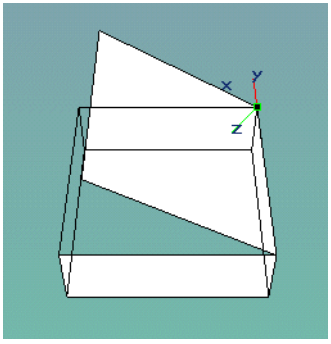
Step 5. Click **OK** in the **Position DOF** dialog box.

Step 6. **Choose Local-DOF/Set DOF Limits** to open the *Set DOF Limits* dialog box.

Step 7. In the **Rotate** area, click **Roll (Y)**. The limits you set previously should be displayed.



Step 8. Click **Animate**, and confirm that the box top is now opening up and out.



Animation

Animation sequences (such as explosions or blinking runway lights) are created by building models that represent each frame in the animation sequence, attaching these models to a group node in the order in which they are to be displayed, and selecting either the **Loop Forward** or **Loop Backward** attribute of the group node.

Creator animates a model by displaying each child of a group node, one at a time, in sequence, from left to right in the hierarchy. The process is similar to playing a film clip, with each child node forming one "frame" of the animation.

Overview: Animating a Model

To animate a model, you must:

1. Create the model.
2. Create a group node that controls the part of the model you want to animate.
3. Create the geometry for each step in the animation sequence.
4. Make each step a child of the group node.
5. Enable the animation attribute for the group node.
6. Start the animation using **Attributes/Animate**.

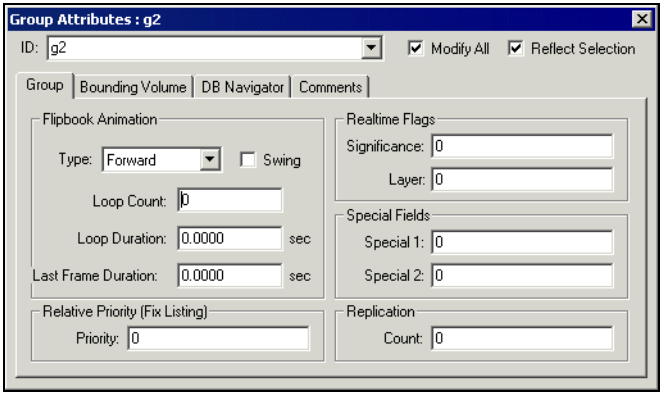
Note: As an alternative to creating geometry for each step, you can attach a replicated group or object to an animated group node, and then start the animation. Each replication acts as one frame in the animation sequence.

The following exercise demonstrates the process of creating an animation.

Setting the Animation Attributes

The **Flipbook Animation** attributes in the **Group** panel of the *Group Attributes* window control the animation sequence of the group's children, along with other related runtime attributes.

To display the attributes of a particular group, double-click the group node in the Hierarchy view, or select the node and choose **Attributes/Modify Attributes**. The *Group Attributes* window appears.



Flipbook Animation has three options: **None**, **Forward**, and **Backward**. When an animation sequence is specified, only one child of the group is displayed at a time.

- **None** disables animation.
- **Forward** displays child nodes from left to right in the hierarchy. When the last child is displayed, the sequence starts again with the child at the far left in the hierarchy.

- **Backward** displays child nodes from right to left in the hierarchy. When the last child is displayed, the sequence starts again with the child at the far right in the hierarchy.

If you select **Forward** or **Backward**, the **Swing** box becomes an additional option. When **Swing** is chosen, your animation will run from the first frame to the last, and then "swing" back in reverse order.

The default **Loop Count** is 0; a value of 0 means that the animation runs continuously. To specify a limited number of loops, enter the number of times you want the animation to play.

To specify a duration for your animation, enter the number of seconds in **Loop Duration**.

If you want to hold the last frame of the animation for a particular length of time, specify the number of seconds for **Last Frame Duration**.

Creator does not display all the runtime values which you enter here; they are passed to your runtime application.

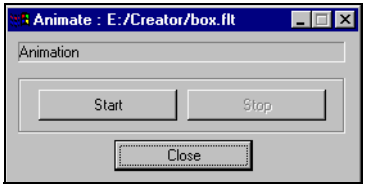
Remember, as you are working with your animation group, you can also use the Flip Book Tool to make modifications to your animation. Select the animated group node and choose the Flip Book Tool from the Geometry toolbox. When you do this, the tool detects all the "frames" (textured polygons) below the animated group node, and lets you modify them with the same controls used for creating animations.

Controlling Animation

The Creator animation feature allows you to preview one animation sequence at a time. Creator draws the animation as quickly as possible; you cannot control the speed. To display an animation sequence:

Step 1. Select a group node that has an **Animation** attribute setting of **Loop Forward** or **Loop Forwards and Backwards**.

Step 2. Select **Attributes/Animate** (Ctrl+Shift+A). The *Animate* dialog box appears.



Step 3. Click **Start** to begin the animation.

Step 4. To stop the animation, click **Stop**. Click **Start** to resume the animation.

Step 5. When you are finished, click **Close**.

Exercise: Creating an Animation

In this exercise, you will create a warning beacon with an animated blinking light. Before you begin, you may want to print this topic to make it easier to follow the instructions as you work with Creator.

Getting Started

Step 1. Open a new, untitled database.

Step 2. Split the window into Graphics and Hierarchy views.

Step 3. Click the **View Panel** button in the Toolbar, and set a Grid **Dimension** of 3.000.



Step 4. Click the **Primary Color Palette** button in the Toolbar, select a color for the beacon (olive green, for example), and close the Color palette.



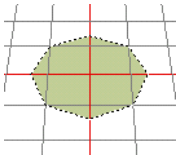
Create the Beacon



Step 1. Open the **Face** toolbox.

Step 2. Choose the **Circle** tool.

Step 3. Create an 8-sided circle with its center at the origin, and a point that is one grid unit from the center. Click **OK**.



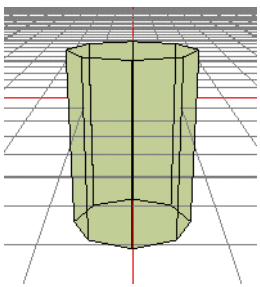
Step 4. Open the **Geometry** toolbox and drag it away from the Toolbox caddy so it stays open.



Step 5. Choose the **Wall** tool.

Step 6. Enter a **Wall Height** of 12, clear **Keep Wall Bottom**, and click **OK**.

Step 7. In the View controls, enter 9 in the **Offset** field and press **Enter**. The tracking plane, which was positioned at the bottom of the cylinder, moves up 6 units. You can set the eyepoint controls in the Toolbar to Y=0, P=25, and R=0 to get a clearer view of the tracking plane position.





Step 8. Open the **Modify Geometry** toolbox. Drag it to one side so it stays open.

Step 9. Select the beacon, and then click the **Slice** tool. The beacon is now divided into two objects.

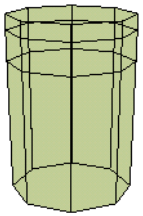


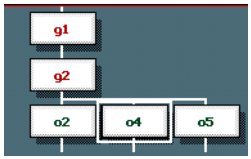
Step 10. In the View controls, enter 11 in the **Offset** field and press **Enter**. The tracking plane moves up 2 units.



Step 11. Select the top object in the beacon, and then click the **Slice** tool.

Step 12. Turn off the grid display. You should now see a cylinder that is divided into three objects.



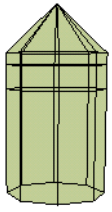


Step 13. Change to Face mode (Shift-F).

Step 14. Select the top of the cylinder and then select the **Peak** tool from the **Geometry** toolbox.



Step 15. Enter a **Peak Height** of 4, clear **Keep Bottom Face**, and then click **OK**. The beacon now has a peaked top.



Create the Animation Group

In this part of the exercise, you add "blinking lights" to the center segment of the beacon. In the process, you must move the geometry to its own group, and create two child objects: one containing the geometry for the lights when they are "on;" and one for the lights when they are "off."

Step 1. Set **g1** as the parent.

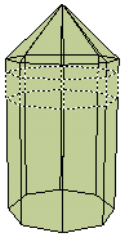


Step 2. Open the **Create** toolbox.

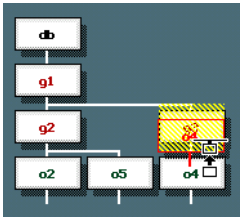


Step 3. Click **Create Group**. A new group, **g3**, appears under **g1**.

Step 4. Change to **Object** mode (Shift-O), and select the center segment of the beacon.

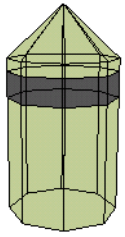


Step 5. In the Hierarchy view, drag the selected object on top of the new group, to make it a child of the group.



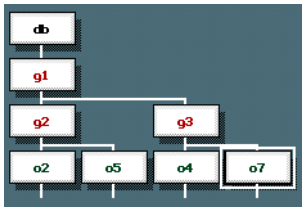
Step 6. Open the Color palette and change the current color to dark gray or black.

Step 7. Select **o4** and choose **Edit/Insert Color** (Ctrl+R). The “light” area is now defined by a band of contrasting color. This is the “lights off” geometry.



Step 8. Select **g3**, click **Set Parent**, and then select its child, **o5**.

Step 9. Open the **Duplicate** toolbox and click **Duplicate**. Click **OK** in the *Duplicate* dialog box to place the duplicate geometry in the same position as the original. The new object, **o7**, appears under **g3**. This object represents the "lights on" state of the beacon.



Step 10. Change the current color to yellow.

Step 11. Right-click **o7** to expand the tree beneath it, and then hold down the Shift key and select every other child.



Step 12. Choose **Edit/Insert Color** (Ctrl+R). The selected faces turn yellow.

Step 13. Change the current color to red.

Step 14. Select every other *yellow* polygon and choose **Edit/Insert Color** (Ctrl+R).

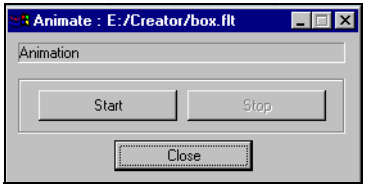


Animate and Test the Group

Step 1. Double-click the group node **g3**. The *Group Attributes* window appears.

Step 2. Under **Flipbook Animation**, choose **Loop Forward**, and then close the window.

Step 3. With the group node still selected, choose **Attributes/Animate**. The *Animate* dialog box appears.



Step 4. Click **Start**. The beacon lights blink on and off.

22 Instrumentation

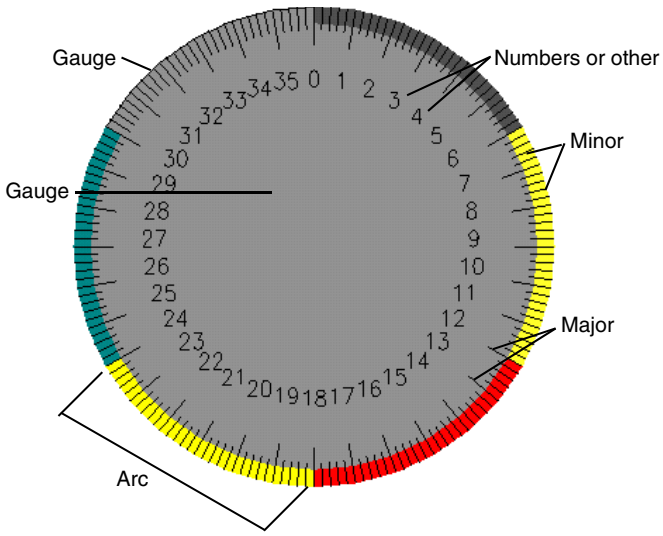
The instrumentation option provides automated layout for dial, vertical, cylindrical, and pitch ladder gauges, and includes a clipping feature to control the display of gauge components. It also includes an editor for creating and editing polygonal fonts. These fonts can be used in the gauge layouts, or can be placed elsewhere in the database using **Instruments/Place Text**.

This chapter describes how to use all the features of the Instrumentation option.

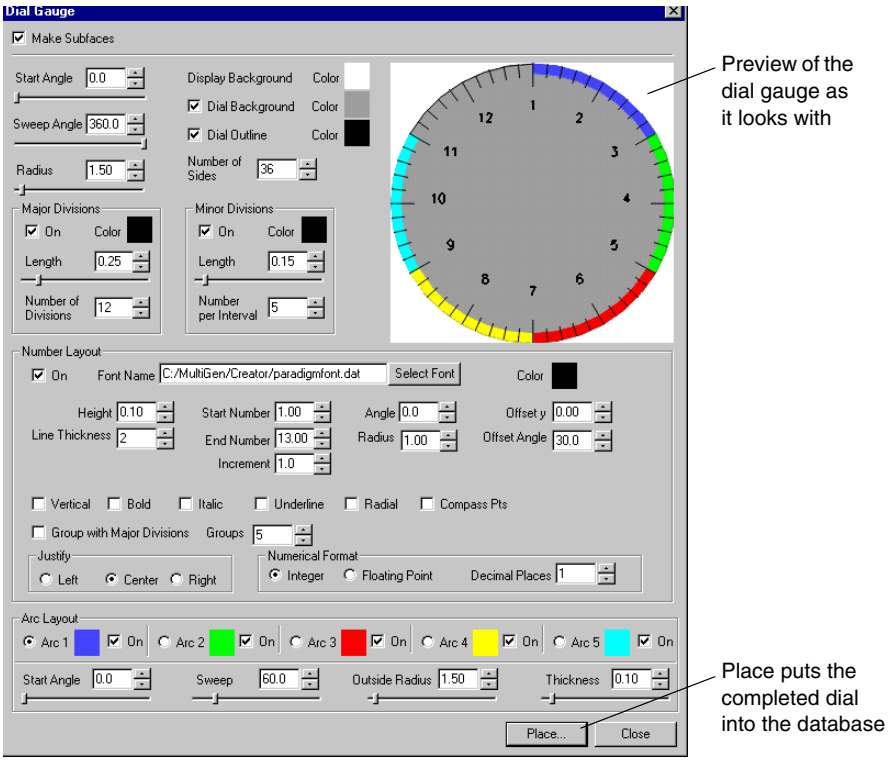
Creating a Dial Gauge

A *dial gauge* is a flat, circular gauge subdivided with a sequence of lines (or *division marks*) and numbers.

A dial gauge has these basic elements:



You create dial gauges in the *Dial Gauge* layout window. To open this window, select **Instruments/Dial Gauge**.



The display in the upper-right corner of this window shows how the dial looks at the current settings. At the bottom of the window is a **Place** button, which you click when you are ready to place the finished gauge in the database.

To design a dial gauge:

Step 1. Define the dial size, background, and general layout of the area that will contain numbers and division marks.

Step 2. Add numbers or other labels to the dial.

Step 3. Add arcs to emphasize selected areas of the dial.

Step 4. Click **Place** to create and position the gauge in the database.

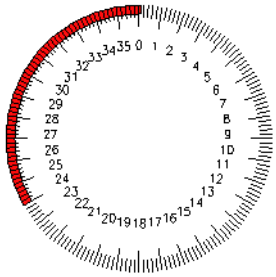
Each of these steps is described in the following sections.

Defining the Dial Background

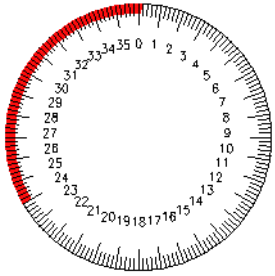
The first step in defining a dial gauge is to choose the background size and appearance and lay out the numbers and division marks.

Step 1. To add a background polygon and outline polygon to the dial, set **Dial Background** and **Dial Outline**.

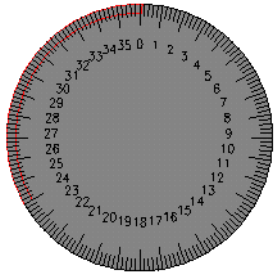
The choice of background depends on how the dial fits into your database design. You can create a dial with or without a solid background polygon and with or without an outline background polygon. Adding a wireframe outline to a solid background polygon improves the appearance of the dial when the database is drawn with antialiased lines.



Dial Background and Dial Outline off



Dial Background off and Dial Outline on



Dial Background and Dial Outline on

Step 2. If you set one or both background elements, choose their colors:

Click a **Color** button to set it to the current primary color.

To choose a different color, double-click a **Color** button to open the Color palette, choose a different color in the palette, and then click the **Color** button in the *Dial Gauge* window to assign the new current color to the background element.

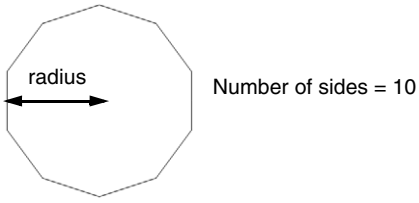
To cancel the last color change, middle-click the **Color** button.

You can also choose a contrasting **Display Background Color**. This color fills in the parts of the preview area that are not defined in the final gauge. You

can set this to the color of the instrument panel on which the gauge will rest, for example.

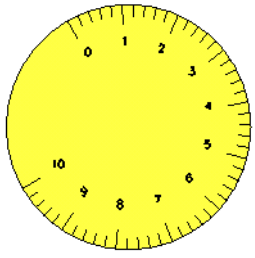
Step 3. Choose the **Number of Sides** for the circles that become the background and outline polygons.

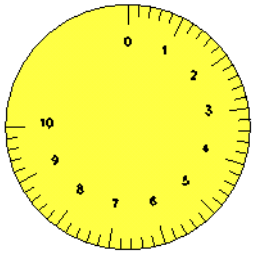
The coarse appearance of the dial shown in the following figure is a result of the low **Number of Sides**. Increasing the number makes the dial appear more circular, but also increases drawing overhead.

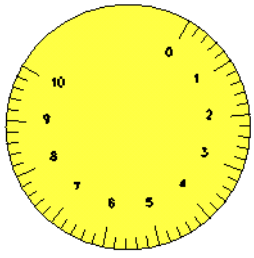


Step 4. If you are using Z-buffering to determine drawing priority, set **Make Subfaces**. If you are using fixed-listing or BSP priority, clear this option. See "Creating the Gauge Hierarchy" on page 781 for more information.

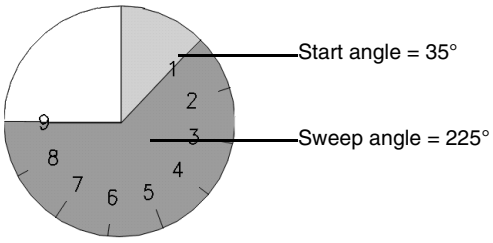
Step 5. Choose a **Start Angle** to specify the starting point on the dial for numbers and division marks. A start angle of 0 indicates the top of the dial. By default, this is 0 degrees (the top of the dial), but you can enter any number between 0 and 360 to specify a clockwise offset. Numbers greater than 180 degrees give the appearance of a counterclockwise offset.


Start Angle = 330 degrees


Start Angle = 0 degrees


Start Angle = 30 degrees

Step 6. Enter a **Sweep Angle** of 360 degrees or less to define the part of the dial that displays division marks and numbers.



Step 7. Choose a **Radius**, in database units, to specify the radius of the **Dial Background**, **Dial Outline**, and **Major/Minor Divisions** (if these divisions are enabled).

Step 8. Set the **Major Division** option if you want major divisions to appear in your dial. Major divisions divide the dial into evenly-spaced intervals. The number of intervals is determined by the **Number of Divisions** setting. You can also change the length of the major division marks by adjusting the **Length** slider.

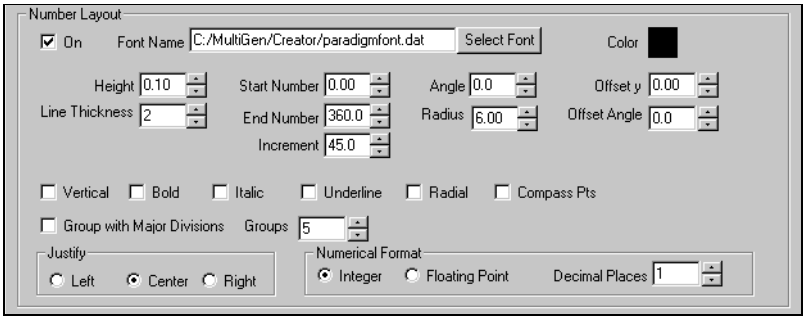
Step 9. Set the **Minor Division** option if you want minor division marks to appear in your dial. Minor divisions subdivide each major division. The number of Minor Divisions that appear between each major division is determined by the **Number per Interval** setting. You can also change the length of the minor division marks by adjusting the **Length** slider.

Step 10. Choose a color for **Major Divisions** and **Minor Divisions**.

Once you have complete the dial layout, you can add labels such as numbers or compass points to the dial.

Adding Labels to the Dial

The **Number Layout** section of the *Dial Gauge* layout window lets you add numbers or compass point labels to the dial gauge.



You can also add text or symbolic labels by creating your own text or symbol font using the **Font Editor**, which is also available from the **Instruments** menu.

To add labels to the dial:

Step 1. Set **On** in the **Number Layout** panel.

Step 2. A font is selected by default. If you want to use a different font, click **Select Font** and then select a font file (`.dat`) in the dialog box that appears.

You can create your own font, and even substitute text or graphics for numbers using the **Font Editor** (see "Using the Font Editor" on page 807). After you create the font, you can use **Select Font** to add labels to gauges.

Step 3. Click the **Color** button to set the label color to the current primary color.

To choose a different color, double-click the **Color** button to open the Color palette, choose a different color in the palette, and then click the **Color** button in the *Dial Gauge* window to assign the new current color to the labels.

To cancel the last color change, middle-click the **Color** button.

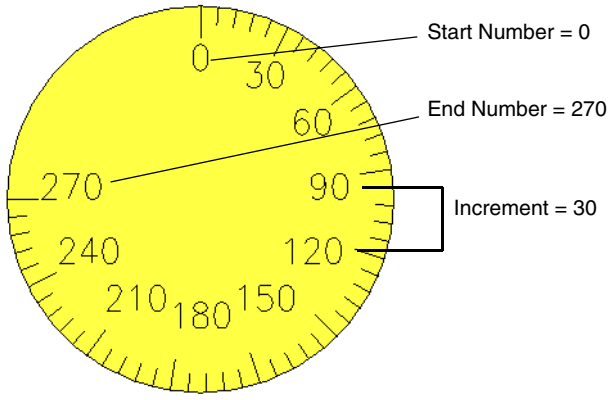
Step 4. Use the arrows to adjust the **Height** and **Line Thickness** of the numbers, if you want. **Height** controls the size of the font; **Line Thickness** controls the

width of each character, in pixels. **Line Thickness** is ignored if **Bold** is selected.

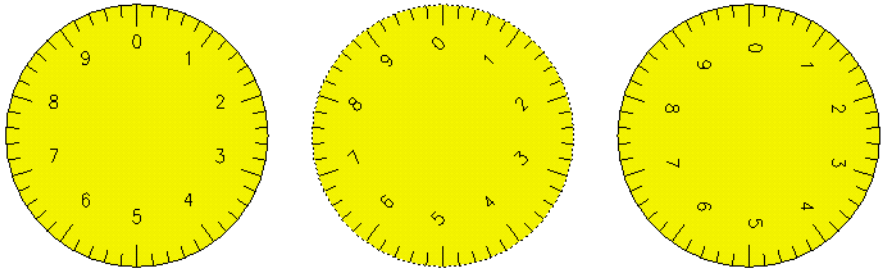
Step 5. Enter a **Start Number** to define the label that appears at the beginning of the **Start Angle**.

Step 6. Enter an **End Number** to specify the symbol that appears at the last position in the **Sweep Angle**.

Step 7. Enter an **Increment** to define where the intermediate labels appear. These labels will be spaced evenly between the **Start Number** and the **End Number**. A negative increment distributes numbers counterclockwise along the gauge.

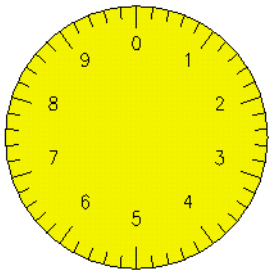


Step 8. Enter an **Angle** if you want to rotate each number or label counterclockwise around its center.

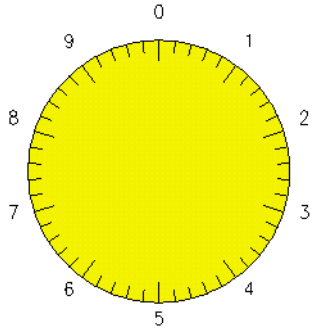


Angle = 0 Angle = 45 Angle = 270

Step 9. Select a **Radius** for the number layout. Note how this radius interacts with the radius of the **Dial Background**.



Number Layout Radius is less than Dial Background Radius

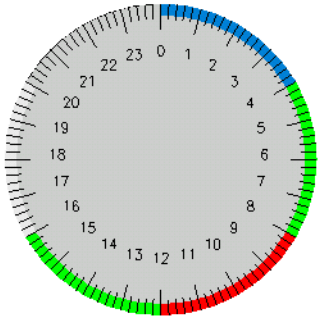


Number Layout Radius is greater than Dial Background Radius

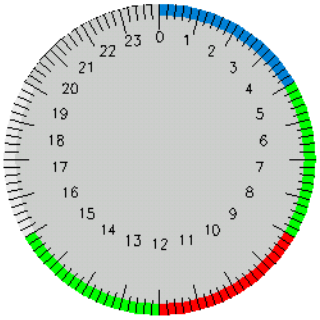
Step 10. Set **Offsets** if you want to change the placement of the numbers.

Offset y moves all numbers/labels up or down relative to the center of the dial.

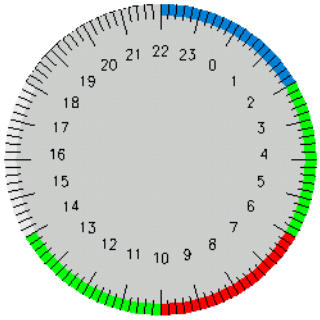
Offset Angle rotates the origin of the numbers/labels the specified number of degrees.



Number arrangement with no offsets



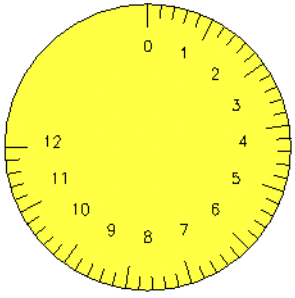
Number arrangement with Offset y = 30



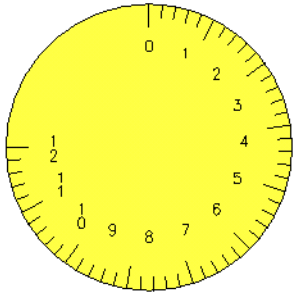
Number arrangement with Offset Angle = 30

Step 11. Choose any font effects you want by clicking the option: **Vertical**, **Bold**, **Italic**, **Underline**, **Radial**, and/or **Compass Points**.

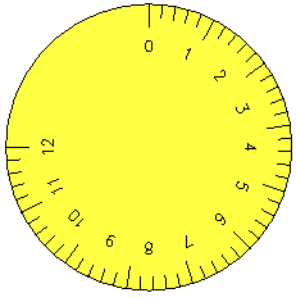
Bold, **Italic**, and **Underline** produce the standard effects found with most fonts. **Vertical** aligns digits vertically (this effect is only noticeable with numbers greater than 9). **Radial** aligns digits to follow the dial's arc.



No special effects



Vertical effect



Radial effect

Step 12. Set **Group With Major Divisions** if you want to create one group in the hierarchy for each major division mark on the gauge and attach the groups as children of a switch node. (This allows you to dynamically control the display of different parts of the gauge.) If you set this option, it overrides

Make Subfaces. See "Creating the Gauge Hierarchy" on page 781 for more information.

Adding Arcs

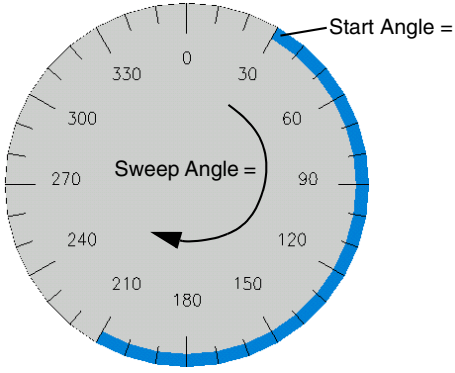
Arcs are solid polygons that are placed along the edge of the dial gauge, usually to be used as indicators. For example, speedometers commonly have "red zones" to indicate when the speed could be damaging to the vehicle's engine.

A single dial layout can contain up to five arcs. To create arcs:

Step 1. In the **Arc Layout** area, set **On** for each arc you want to add to the dial.

Step 2. Click the option button next to an arc to select it for editing.

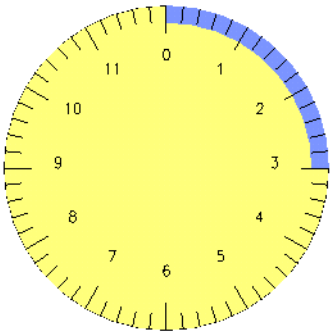
Step 3. Choose a **Start Angle** to set the starting point for the arc. The **Start Angle** is expressed as a clockwise offset from the top of the dial.



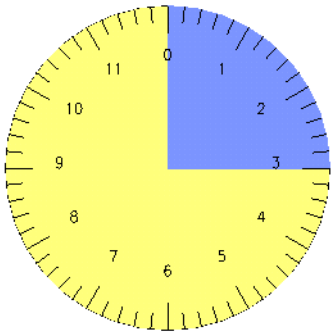
Step 4. Choose a **Sweep Angle** to set the length of the arc around the perimeter of the dial.

Step 5. Choose an **Outside Radius** for the arc. For an arc that follows the perimeter of the background, for example, set the **Outside Radius** to the same value as the **Radius** you set for the background (see "Defining the Dial Background" on page 772).

Step 6. Set the **Thickness** of the Arc. **Thickness** can be set to any value between zero and the value you set for the background **Radius**.



Arc Thickness is set to 1/10 the radius of the background



Arc Thickness is set to the same value as the radius of the background

Defining Additional Arcs

To create a dial gauge with more than five arcs, do the following:

Step 1. Define a dial gauge containing five arcs.

Step 2. Click **Place** to create the gauge in the database (see "Completing the Gauge" on page 780).

Step 3. Choose **Dial gauge** from the **Instruments** menu to reopen the *Dial Gauge* window.

Step 4. Turn off all dial parameters except arcs.

Step 5. Define the arcs you want to add.

Step 6. Click **Place** and add the new arcs to the original dial.

Completing the Gauge

The final step in creating a dial gauge is to place the gauge in the database. Before you place the gauge, you must decide how you want to arrange the hierarchy of the gauge components.

Creating the Gauge Hierarchy

You can create components of the gauge as subfaces of the background polygon (by setting **Make Subfaces** at the top of the window), or you can group major divisions under a switch node (by setting **Group With Major Divisions** at the bottom of the window, and then setting the number of **Groups**).

Make Subfaces determines whether the polygons representing the arcs, numbers, and division marks of the dial are subfaces of the background polygon, as follows:

- If **Make Subfaces** and **Dial Background** are both on, the faces of the gauge are attached to the background polygon in the database.
- If **Make Subfaces** is on and **Dial Background** is off, the faces of the gauge are attached to the gauge's central polygon.
- If **Make Subfaces** is off, Creator builds a hierarchy for the gauge that combines the polygons representing each arc, division mark, and number into separate groups.

The **Group With Major Divisions** option at the bottom of the *Dial Gauge* window overrides **Make Subfaces**.

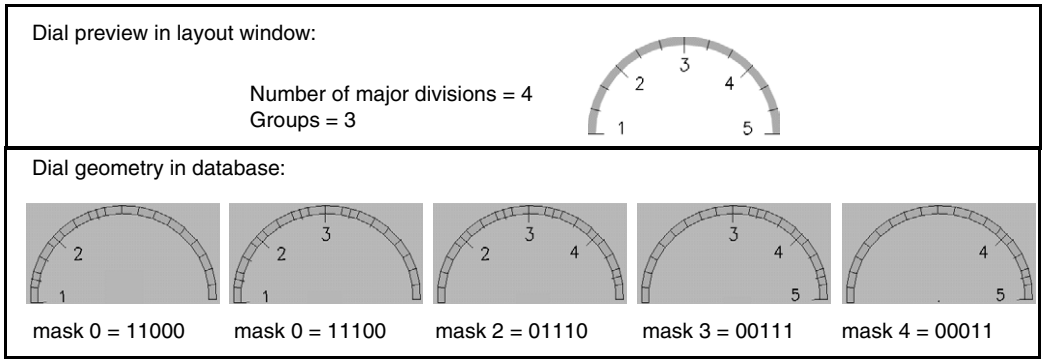
Group With Major Divisions creates one group in the hierarchy for each major division mark on the gauge and attaches the groups as children of a switch node, allowing you to dynamically control the display of different parts of the gauge. This is useful in displaying a portion of a moving dial gauge, such as a heading indicator, because the switch linked to the heading can do a preliminary culling of the numbers and divisions.

When **Group With Major Divisions** is on:

- One group node contains each major division mark plus all the minor division marks and numbers within the major division.
- These groups are attached to a switch node that contains one mask for each group.
- The **Groups** value determines the number of groups switched on in each mask. The middle set of groups is turned on in the middle mask. This mask is successively shifted left one place to create lower masks and successively shifted right one place to create higher masks.

Consider the dial in the example below, which contains four major divisions. **Groups** is set to 5.

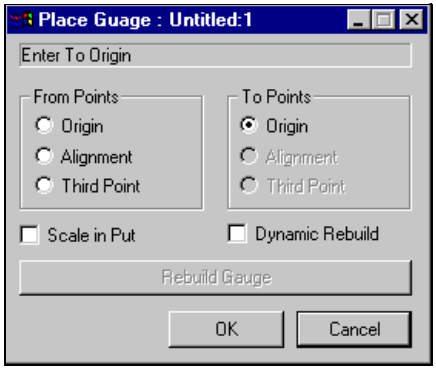
- The switch node contains five groups, one for each major division mark.
- The switch node contains five masks, one for each group.
- Because **Groups** = 3, three groups are switched on in each mask. The middle mask switches on the middle three groups. Lower masks are successively shifted left and higher masks are successively shifted right.



Placing a Gauge in the Database

When the preview of the gauge meets your requirements, you are ready to create the gauge and place it in the database. To do this:

Step 1. Click the **Place** button in the gauge layout window. The *Place Gauge* dialog box appears.



Step 2. Minimize the gauge layout window, if necessary, and then click a point in the Graphics view to set the **Origin** of the gauge. The **To Point** display in the *Place Gauge* dialog box changes to **Alignment**.

Step 3. Click a point in the Graphics view to set the alignment point for the gauge. (This vertex is shown in green until you click **OK** to create the gauge.)

Step 4. Choose any of the other options as needed:

Scale In Put scales the gauge using the distance between the origin and alignment point. (This option is not available with Pitch Ladder gauges.

Rebuild Gauge rebuilds the gauge in the database using the current set of layout parameters. This lets you edit gauge parameters while placing the gauge in the database. **Rebuild Gauge** does not become available until you change parameters in the gauge layout window.

Dynamic Rebuild rebuilds the gauge in the database automatically if one of the layout parameters changes.

Step 5. Click a third point in the Graphics view to complete the gauge, and then click **OK**.

Central Polygon: Dial and Cylindrical Gauges

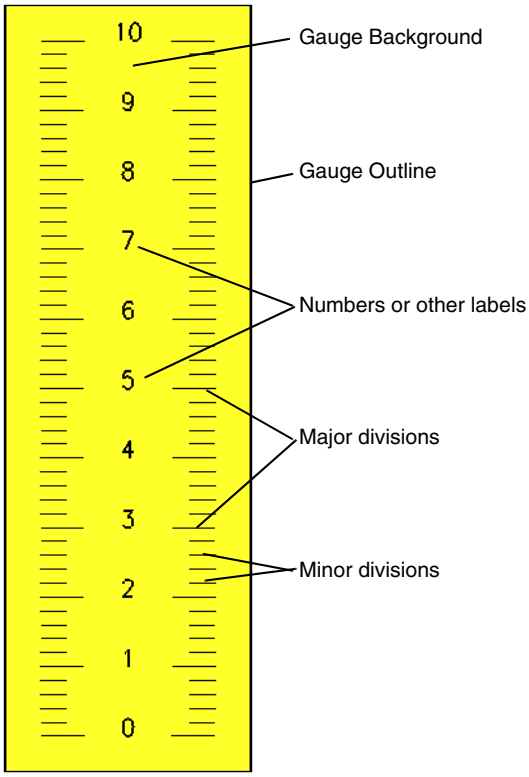
When you place a dial gauge or cylindrical gauge in the database, Creator creates a one-vertex polygon that marks the center of the gauge.

- The dial gauge central polygon appears at the center of the radius of the dial.
- The cylindrical gauge central polygon appears at the center of the radius at the bottom of the cylinder.

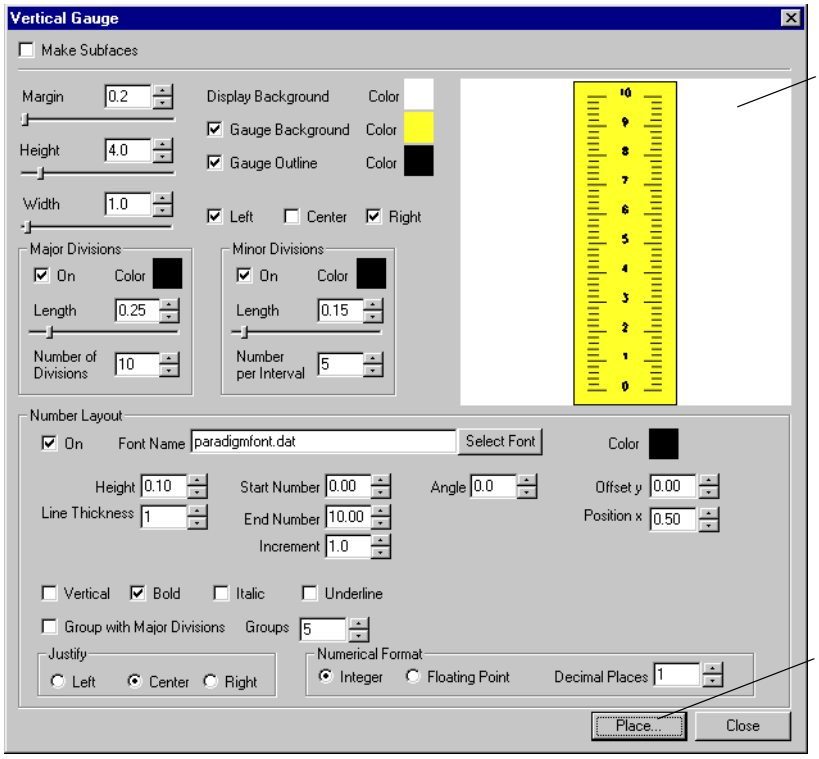
The central polygon makes it easy to add needles or other controls to the gauge. If you do not want a central polygon, delete it from the final geometry of the gauge.

Creating a Vertical Gauge

A *vertical gauge* is a rectangular gauge containing numbers and horizontal division marks. This is a common gauge type that is used to indicate air speed, altitude, fuel levels, and so forth. A vertical gauge has these basic elements:



You create vertical gauges in the *Vertical Gauge* layout window. To open this window, select **Instruments/Vertical Gauge**.



The display in the upper-right corner of this window shows how the gauge looks at the current settings. At the bottom of the window is a **Place** button, which you click when you are ready to place the finished gauge in the database.

To design a vertical gauge:

Step 1. Define the gauge size, background, and general layout of the area that will contain numbers and division marks.

Step 2. Add labels to the gauge.

Step 3. Click **Place** to create and position the gauge in the database.

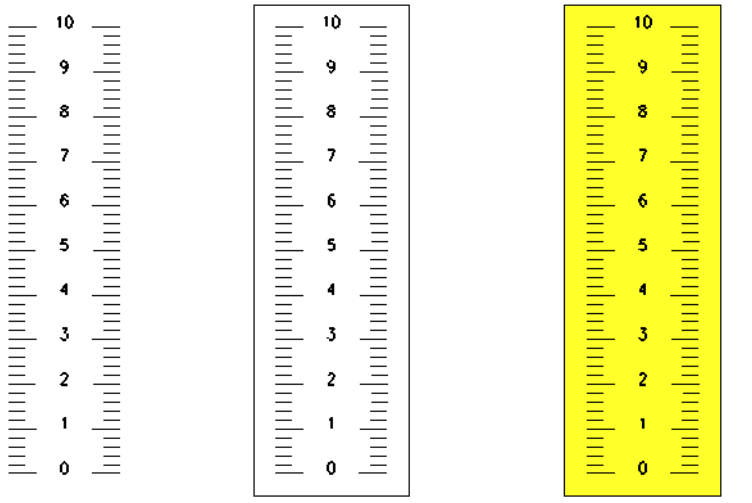
Each of these steps is described in the following sections.

Defining the Gauge Background

The first step in defining a vertical gauge is to choose the background size and appearance and lay out the numbers and division marks.

Step 1. To add a background polygon and outline polygon to the gauge, set **Gauge Background** and **Gauge Outline**.

The choice of background depends on how the gauge fits into your database design. You can create a gauge with or without a solid background polygon and with or without an outline background polygon. Adding a wireframe outline to a solid background polygon improves the appearance of the gauge when the database is drawn with antialiased lines.



Gauge Background and Gauge Outline off

Gauge Background off and Gauge Outline on

Gauge Background and Gauge Outline on

Step 2. If you set one or both background elements, choose their colors:

Click a **Color** button to set it to the current primary color.

To choose a different color, double-click a **Color** button to open the Color palette, choose a different color in the palette, and then click the **Color** button in the *Vertical Gauge* window to assign the new current color to the background element.

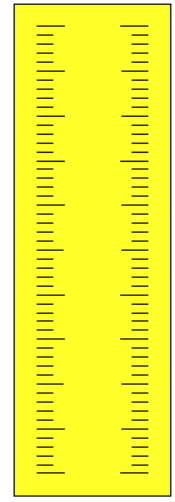
To cancel the last color change, middle-click the **Color** button.

You can also choose a contrasting **Display Background Color**. This color fills in the parts of the preview area that are not defined in the final gauge. You can set this to the color of the instrument panel on which the gauge will rest, for example.

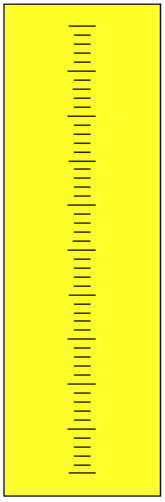
Step 3. Choose the **Height** and **Width** of the background and outline polygons.

Step 4. Set the **Margin** between the division lines and the outer margin of the gauge.

Step 5. Set the combination of **Left**, **Right**, and **Center** to position the division lines.



Left and Right are set, Center is cleared



Center is set, Left and Right are cleared

Step 6. If you are using Z-buffering to determine drawing priority, set **Make Subfaces**. If you are using fixed-listing or BSP priority, clear this option (see "Creating the Gauge Hierarchy" on page 781).

Step 7. Set the **Major Division** option if you want major divisions to appear in your gauge. Major divisions divide the gauge into evenly-spaced intervals. The number of intervals is determined by the **Number of Divisions** setting. You can also change the length of the **Major Division** marks by adjusting the **Length** slider.

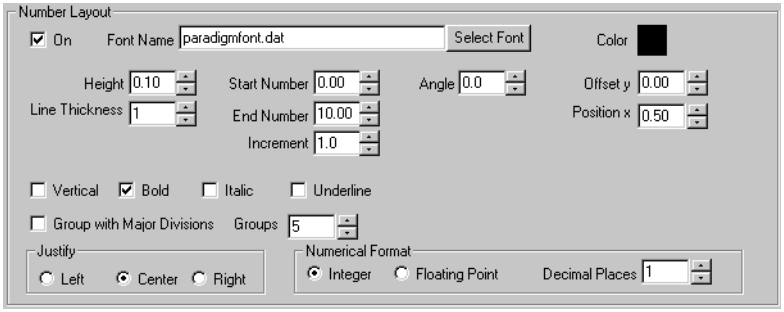
Step 8. Set the **Minor Division** option if you want minor division marks to appear in your gauge. minor divisions subdivide each major division. The number of minor divisions that appear between each major division is determined by the **Number per Interval** setting. You can also change the length of the **Minor Division** marks by adjusting the **Length** slider.

Step 9. Choose a color for **Major Divisions** and **Minor Divisions**.

Once you have completed the gauge layout, you can add numbers to the gauge.

Adding Labels to the Gauge

The **Number Layout** section of the *Vertical Gauge* layout window lets you add numbers to the gauge.



To add labels to the gauge:

Step 1. Set **On** in the **Number Layout** panel.

Step 2. A font is selected by default, but if you want to use a different font, click **Select Font** and then select a font file (`.dat`) in the dialog box that appears.

You can create your own font, and even substitute text or graphics for numbers using the **Font Editor** (see "Using the Font Editor" on page 807). After you create the font, you can use **Select Font** to add labels to gauges.

Step 3. Click the **Color** button to set the label color to the current primary color.

To choose a different color, double-click the **Color** button to open the Color palette, choose a different color in the palette, and then click the **Color** but-

ton in the *Vertical Gauge* window to assign the new current color to the labels.

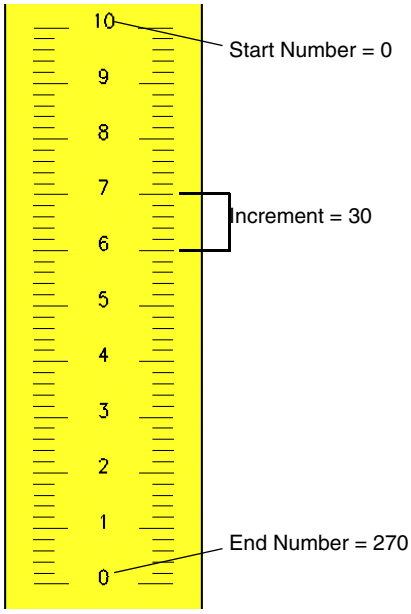
To cancel the last color change, middle-click the **Color** button.

Step 4. Use the arrows to adjust the **Height** and **Line Thickness** of the numbers, if you want. **Height** controls the size of the font; **Line Thickness** controls the width of each character.

Step 5. Enter a **Start Number** to define the label that appears at the bottom of the gauge.

Step 6. Enter an **End Number** to specify the symbol that appears at the top of the gauge.

Step 7. Enter an **Increment** to define the intermediate labels that appear. These labels will be spaced evenly between the **Start Number** and the **End Number**. A positive **Increment** starts the numbering at the bottom of the gauge. A negative **Increment** starts the numbering at the top of the gauge.



Step 8. Enter an **Angle** if you want to rotate each number or label counter-clockwise around its center.

Step 9. Set offsets if you want to change the placement of the numbers.

Offset y moves all numbers/labels up/down relative to the gauge's center.

Position x determines the placement of the major and minor divisions relative to the left edge of the gauge.

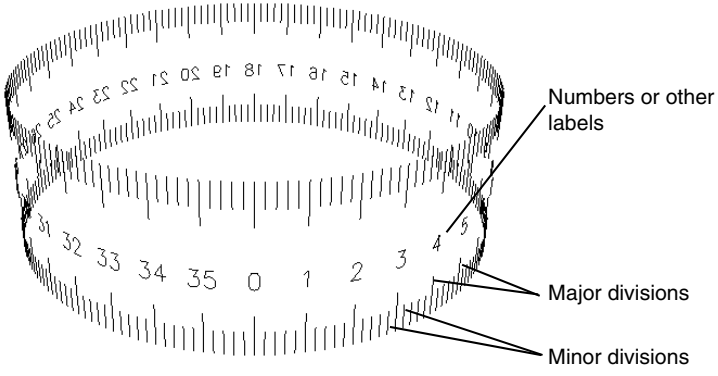
Step 10. Choose any font effects you want by clicking the option: **Vertical**, **Bold**, **Italic**, and **Underline**. **Bold**, **Italic**, and **Underline** produce the standard effects found with most fonts. **Vertical** aligns digits vertically (this effect is only noticeable with numbers greater than 9).

Completing the Gauge

The final step in creating a vertical gauge is to place the gauge in the database. See "Completing the Gauge" on page 780 for instructions.

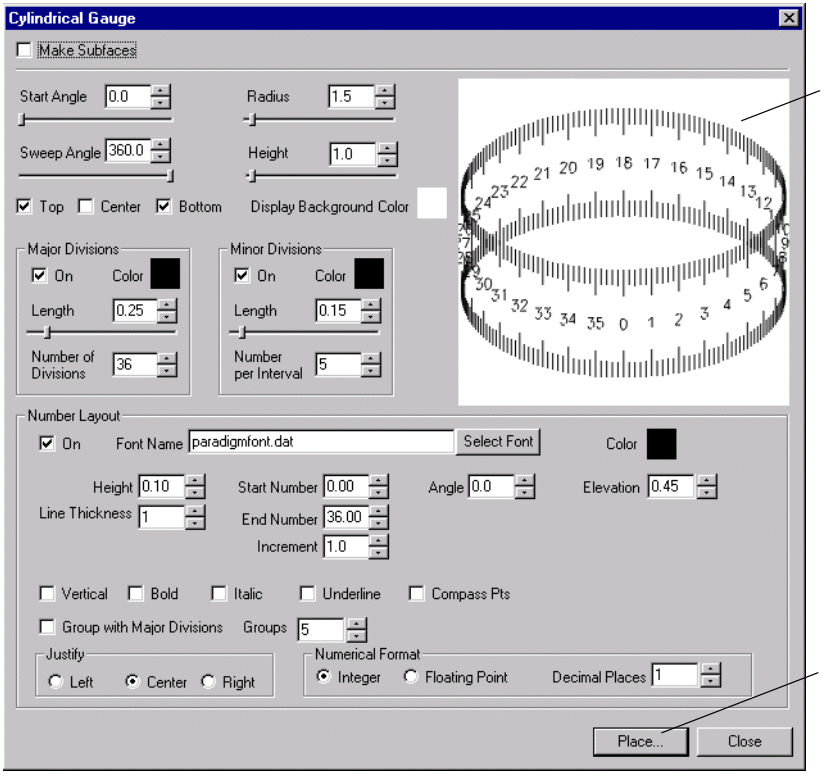
Creating a Cylindrical Gauge

A *cylindrical gauge* places numbers and division marks around the outside of a cylinder. A cylindrical gauge has these basic elements:



Creator does not build the geometry of a cylinder when you create a cylindrical gauge; it adds division marks and numbers to an existing set of background polygons. To create a complete cylindrical gauge, you create the background of the gauge using the standard Creator modeling tools, lay out the number and division lines in the *Cylindrical Gauge* window, and then use **Place** to apply the gauge to the background.

To open the *Cylindrical Gauge* layout window, select **Instruments/Cylindrical Gauge**.



The display in the upper-right corner of this window shows how the gauge looks at the current settings. At the bottom of the window is a **Place** button, which you click when you are ready to place the finished gauge in the database.

To design a cylindrical gauge:

Step 1. Define the numbers and division marks that form the gauge structure.

Step 2. Add labels.

Step 3. Click **Place** to create and position the gauge in the database.

Each of these steps is described in the following sections.

Defining the Gauge Structure

The first step in defining a cylindrical gauge is to choose the size and layout of the gauge.

Step 1. Select the **Display Background Color** to the same color as the background on which you are placing the gauge, to give the most accurate preview of the gauge.

Click a **Color** button to set the **Display Background Color** to the current primary color.

To choose a different color, double-click a **Color** button to open the Color palette, choose a different color in the palette, and then click the **Color** button in the *Cylindrical Gauge* window to assign the new current color to the background element.

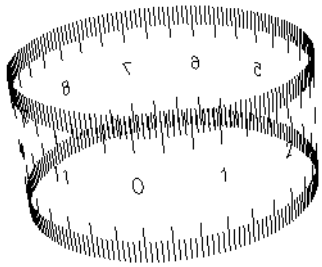
To cancel the last color change, middle-click the **Color** button.

You can also choose a contrasting **Display Background Color**. This color fills in the parts of the preview area that are not defined in the final gauge. You can set this to the color of the background on which the gauge will rest, for example.

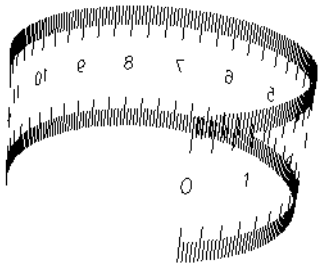
Step 2. Choose the **Height** and **Width** of the gauge.

Step 3. Set the **Start Angle** to position the beginning of the gauge. A **Start Angle** of zero places 0 in the center when you view the gauge from the front.

Step 4. Set a **Sweep Angle** between 0 and 360 degrees to define how far around the cylinder the division lines and numbers will extend.



Sweep Angle of 360 degrees



Sweep Angle of 270 degrees

Step 5. If you are using Z-buffering to determine drawing priority, set **Make Subfaces**. If you are using fixed-listing or BSP priority, clear this option. See "Creating the Gauge Hierarchy" on page 781 for more information about **Make Subfaces**.

Step 6. Set the **Major Division** option if you want major divisions to appear in your gauge. Major divisions divide the gauge into evenly-spaced intervals. The number of intervals is determined by the **Number of Divisions** setting. You can also change the length of the **Major Division** marks by adjusting the **Length** slider.

Step 7. Set the **Minor Division** option if you want minor division marks to appear in your gauge. Minor divisions subdivide each major division. The number of minor divisions that appear between each major division is determined by the **Number per Interval** setting. You can also change the length of the **Minor Division** marks by adjusting the **Length** slider.

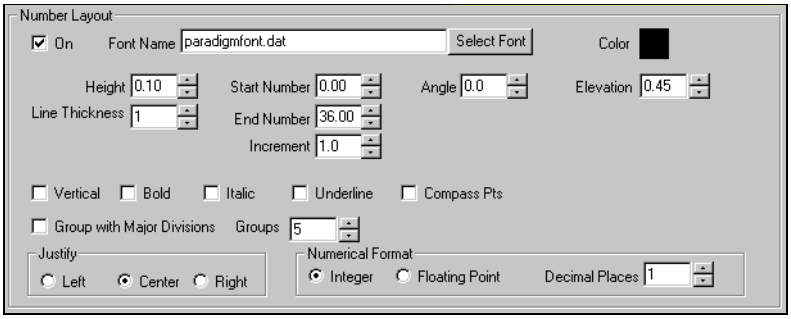
Step 8. Choose a color for **Major Divisions** and **Minor Divisions**.

Step 9. Set any combination of **Top**, **Center**, and **Bottom** to position division marks on the gauge.

Once you have complete the gauge layout, you can add numbers to the gauge.

Adding Labels to the Gauge

The **Number Layout** section of the *Cylindrical Gauge* layout window lets you add numbers to the gauge.



To add labels to the gauge:

Step 1. Set **On** in the **Number Layout** panel.

Step 2. A font is selected by default, but if you want to use a different font, click **Select Font** and then select a font file (`.dat`) in the dialog box that appears.

You can create your own font, and even substitute text or graphics for numbers using the **Font Editor** (see "Using the Font Editor" on page 807). After you create the font, you can use **Select Font** to add labels to gauges.

Step 3. Click the **Color** button to set the label color to the current primary color.

To choose a different color, double-click the **Color** button to open the Color palette, choose a different color in the palette, and then click the **Color** button in the *Cylindrical Gauge* window to assign the new current color to the labels.

To cancel the last color change, middle-click the **Color** button.

Step 4. Use the arrows to adjust the **Height** and **Line Thickness** of the numbers, if you want. **Height** controls the size of the font; **Line Thickness** controls the width of each character.

Step 5. Enter a **Start Number** to define the label that appears at the bottom of the gauge.

Step 6. Enter an **End Number** to specify the symbol that appears at the top of the gauge.

Step 7. Enter an **Increment** to define where the intermediate labels appear. These labels will be spaced evenly between the **Start Number** and the **End Number**.

Step 8. Enter an **Angle** if you want to rotate each number or label counterclockwise around its center.

Step 9. Adjust the **Elevation** to position the numbers on the gauge.

Step 10. Choose any font effects you want by clicking the options **Vertical**, **Bold**, **Italic**, **Underline**, and **Compass Pts**.

Bold, **Italic**, and **Underline** produce the standard effects found with most fonts. **Vertical** aligns digits vertically (this effect is only noticeable with numbers greater than 9).

Compass Pts substitutes compass point characters for numbers, as follows: N for 0, E for 9 or 90, S for 27 or 270, W for 18 or 180.

Completing the Gauge

The final step in creating a cylindrical gauge is to place the gauge on a cylinder in the database. See “Completing the Gauge” on page 780 for instructions.

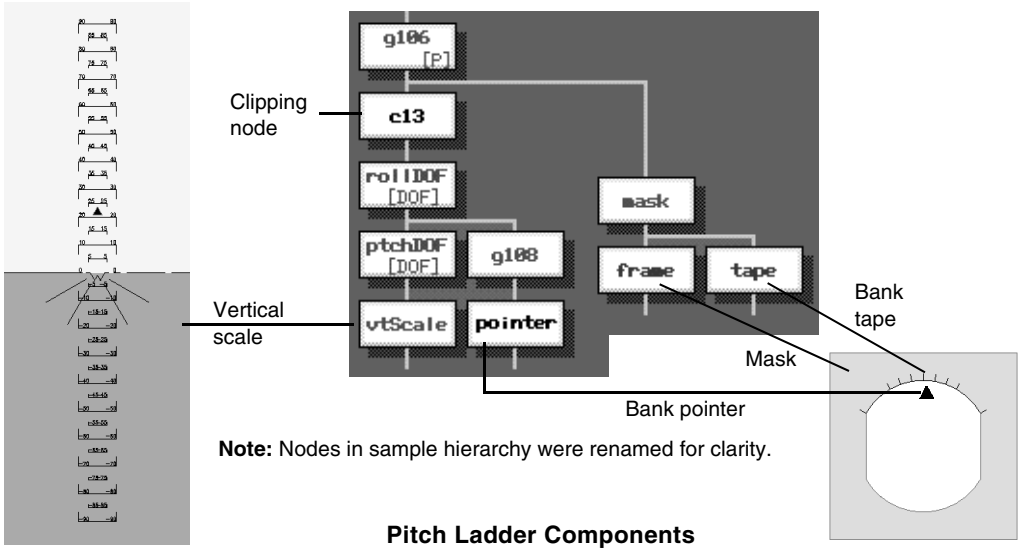
Creating a Pitch Ladder

A *pitch ladder* is commonly used to show roll and pitch in a model of an airplane cockpit. The geometry of a pitch ladder contains the following components. You can include or omit any of these components as you design the gauge.

- The *vertical scale* consists of two rectangular polygons: a sky polygon ruled every 5° (from 0° to 90°) and a ground polygon ruled every 5° (from 0° to -90°). The vertical scale pitches up or down and rolls left or right with the eyepoint.

- A *mask* frames the vertical scale, and a *clipping polygon* restricts the display of the vertical scale to the area of the mask.
- A *bank pointer* provides a measure of eyepoint roll by rotating left or right relative to a fixed *bank tape*.

The following figure shows how each component appears in the database hierarchy.

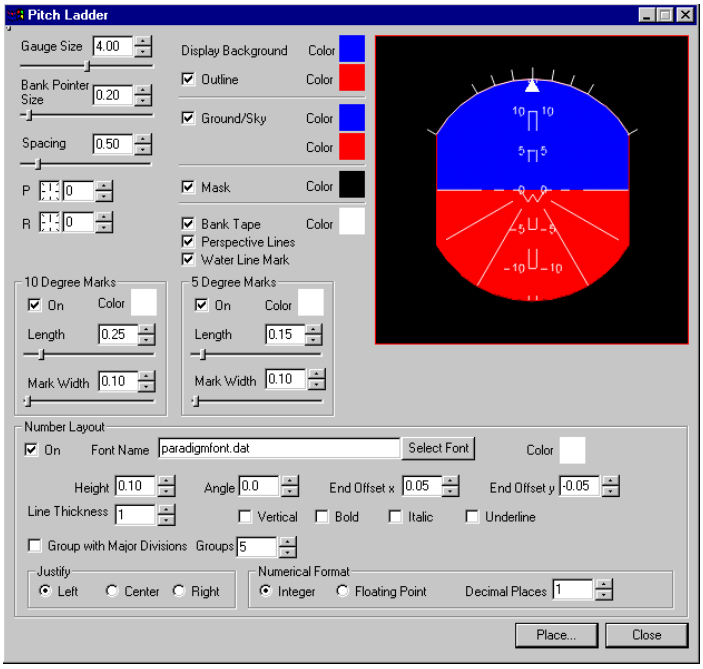


Pitch Ladder Components

The vertical scale is attached to two DOF nodes in the hierarchy.

- The first DOF allows the vertical scale to roll (rotate around the *y*-axis) from -180° to +180°. This DOF also controls the bank pointer.
- The second DOF allows the vertical scale to translate up or down along its own length to produce the effect of pitching the eyepoint relative to the horizon.

You create pitch ladder gauges in the *Pitch Ladder* layout window. To open this window, select **Instruments/Pitch Ladder**.



The display in the upper-right corner of this window shows how the dial looks at the current settings. At the bottom of the window is a **Place** button, which you click when you are ready to place the finished gauge in the database.

To design a pitch ladder:

Step 1. Define the size, background, mask, markings, and general layout of the gauge area that will contain numbers and degree marks.

Step 2. Add degree marks.

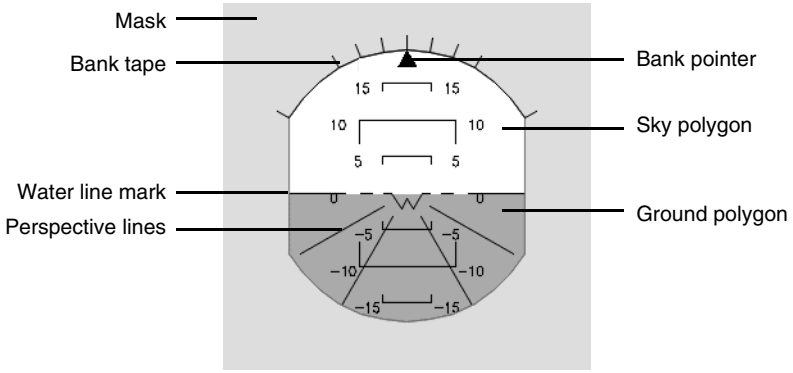
Step 3. Add numbers or other labels to the pitch ladder.

Step 4. Click **Place** to create and position the gauge in the database.

Each of these steps is described in the following sections.

Defining the Pitch Ladder Background

The background of the pitch ladder has the following components.



To define the background elements of the pitch ladder:

Step 1. Adjust the **Gauge Size** to define the height and width of the pitch ladder, in database units. Set this value carefully, because you will not be able to use the **Scale in Put** option in the **Place Gauge** dialog box (see "Completing the Gauge" on page 780) to automatically resize the pitch ladder as you place it in the database.

Step 2. Set **Mask** if you want to create a polygonal mask in front of the vertical scale. The vertical scale is clipped to the area of the mask, whether or not a mask is included in the gauge.

Step 3. If you did not set **Mask**, choose a **Display Background** color. (**Display Background Color** is only visible when **Mask** is cleared.)

To set **Display Background** color to the current primary color, click the **Color** button.

To choose a different color, double-click the **Color** button to open the Color palette, choose a different color in the palette, and then click the **Color** button in the *Pitch Ladder* window to assign the new current color to the **Display Background**.

To cancel the last color change, middle-click the **Color** button.

Step 4. If you set a **Mask**, you can set **Outline** and then choose a color to select an outline color for the perimeter of the gauge, and of the mask area.

Step 5. Set **Ground/Sky** to add two background polygons to the vertical scale. These polygons meet at the horizon (0 degree elevation). You can then select a different color for each polygon.

Step 6. Adjust the **Bank Pointer Size** to define the size of the triangular bank pointer at the top of the gauge. If you do not want a bank pointer to appear in the gauge, set the **Bank Pointer Size** to zero.

Step 7. Adjust the **Spacing** to set the distance, in database units, between 5 degree and 10 degree marks on the pitch ladder.

Step 8. Adjust the **Pitch** to move the vertical scale up or down behind the mask.

Step 9. Adjust the **Roll** to rotate the vertical scale and bank pointer counterclockwise or clockwise within the mask.

Step 10. Set **Bank Tape** to add a bank indicator to the sky polygon.

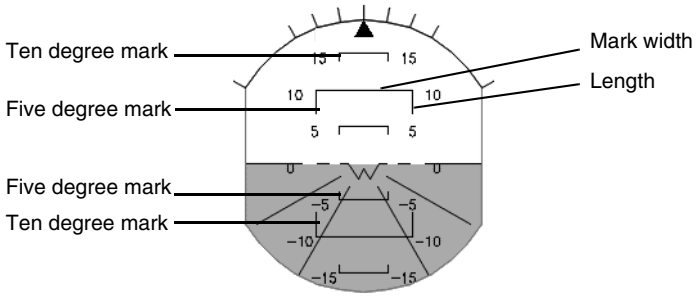
Step 11. Set **Perspective Lines** to add radiating perspective lines to the ground polygon.

Step 12. Set **Water Line Mark** to add a dividing line and W label between the sky and ground polygons.

Step 13. Set the **Color** next to **Bank Tape** to the color you want to apply to the bank tape, perspective lines, water line mark and bank pointer.

Adding Degree Marks

Ten-degree marks appear at ten-degree intervals along the gauge. Five-degree marks appear at odd five-degree intervals (between ten-degree marks).



To add degree marks to the pitch ladder:

Step 1. Set the corresponding **On** checkbox next to the type of mark (10 and/or 5 degree) that you want to add to the gauge.

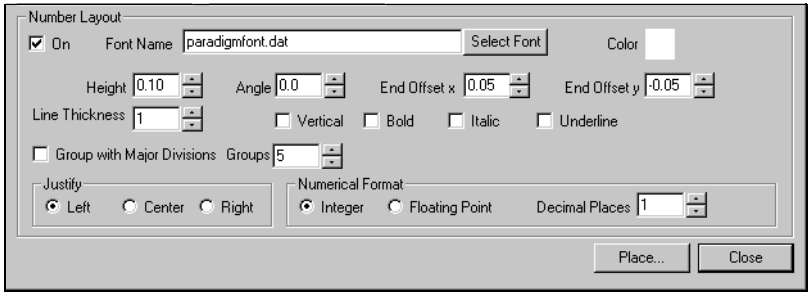
Step 2. Choose a **Length** to define the size of the vertical line at the end of each 5 or 10 degree mark.

Step 3. Choose a **Width** to set the size of the horizontal line that defines each 5 or 10 degree mark.

Step 4. Choose a color for each type of mark.

Adding Labels to the Gauge

The **Number Layout** section of the *Pitch Ladder* layout window lets you add numbers or labels to the gauge.



To add numbers or labels:

Step 1. Set **On** in the **Number Layout** panel.

Step 2. A font is selected by default, but if you want to use a different font, click **Select Font** and then select a font file (.dat) in the dialog box that appears.

You can create your own font, and even substitute text or graphics for numbers using the **Font Editor** (see "Using the Font Editor" on page 807). After you create the font, you can use **Select Font** to add labels to gauges.

Step 3. Click the **Color** button to set the label color to the current primary color.

To choose a different color, double-click the **Color** button to open the Color palette, choose a different color in the palette, and then click the **Color** button in the *Pitch Ladder* window to assign the new current color to the labels.

To cancel the last color change, middle-click the **Color** button.

Step 4. Use the arrows to adjust the **Height** and **Line Thickness** of the numbers, if you want. **Height** controls the size of the font; **Line Thickness** controls the width of each character.

Step 5. Enter an **Angle** if you want to rotate each number or label counterclockwise around its center.

Step 6. Set offsets if you want to change the placement of the numbers.

End Offset y moves all numbers/labels up or down relative to the center of the gauge.

End Offset x moves all numbers/labels towards or away from the center of the gauge.

Step 7. Choose any font effects you want by clicking the options **Vertical**, **Bold**, **Italic**, and **Underline**.

Bold, **Italic**, and **Underline** produce the standard effects found with most fonts. **Vertical** aligns digits vertically (this effect is only noticeable with numbers greater than 9).

Completing the Gauge

The final step in creating a pitch ladder gauge is to place the gauge in the database. See "Completing the Gauge" on page 780 for instructions.

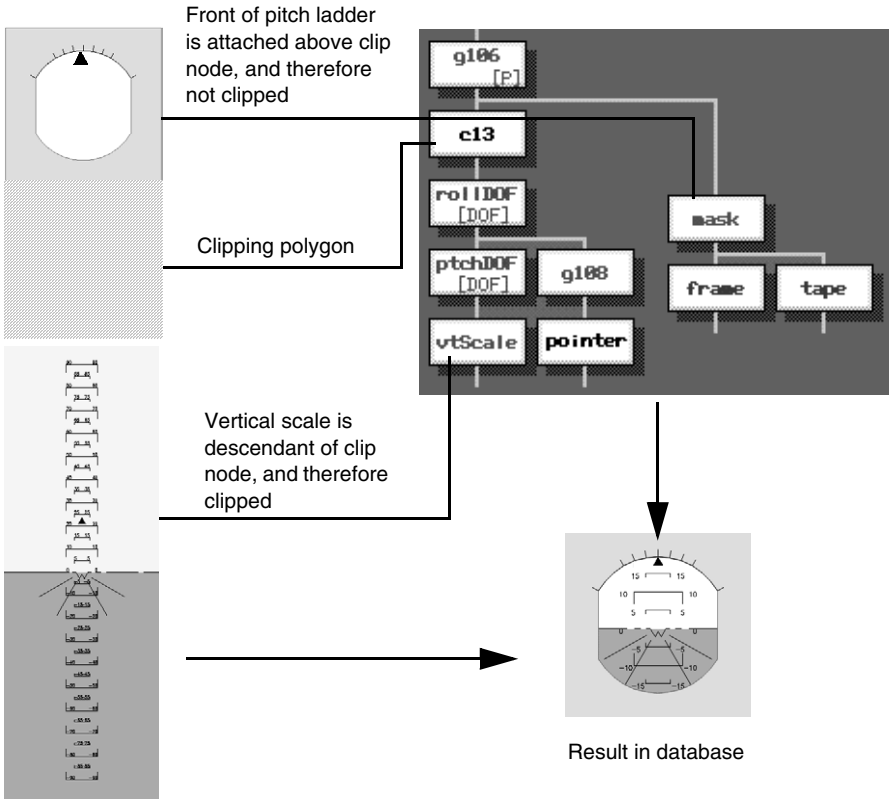
Note: Because a pitch ladder requires a more complicated hierarchy than other types of gauges, the *Pitch Ladder* window does not contain a **Make subfaces** checkbox, and the *Place Gauge* window does not give you the option to scale the gauge as you place it in the database.

Clipping Nodes and Planes

A *clipping node* is a hierarchical element that restricts the display area of its descendants. A clipping node contains the definition of a four-sided polygon. The four edges of this polygon, plus the polygon itself, define five clipping planes that can be independently enabled and disabled. When the database is drawn, the descendants of the clipping node are tested against enabled clipping planes. Geometry behind enabled clippings plane is not drawn.

Clipping nodes are often used to model instruments. For example, in the pitch ladder shown in the following figure, a clipping node is inserted between the mask (the front,

or case, of the pitch ladder) and the vertical scale. The realtime software moves the vertical scale up or down behind the clipping node to change the pitch-ladder display.



Using a Clipping Node to Define a Pitch Ladder

Creating Clipping Nodes

To set up a clipping node:

Step 1. Select the group node or group-like node that is directly above the geometry you want to control with the clip node and then click **Parent**. "Group-like" nodes include DOF nodes, clip nodes, and BSP nodes.



Step 2. Open the View panel.



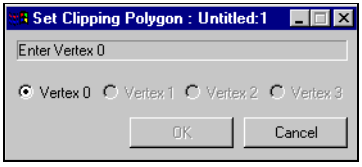
Step 3. Click **Draw Clip Nodes**.

Step 4. Open the **Create Tools** palette.

Step 5. Click **Create Clip**.

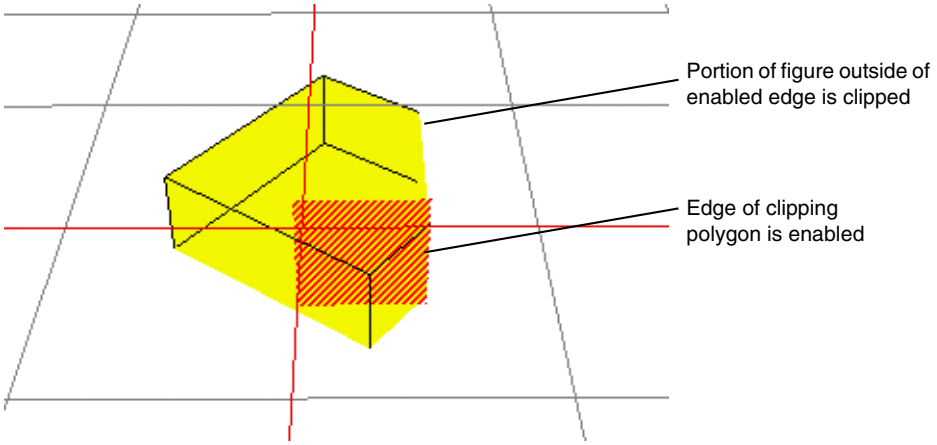
Step 6. Select the new clipping node and choose **Instruments/Set Clipping Polygon**.

The *Set Clipping Polygon* dialog box prompts you to define the clipping polygon.



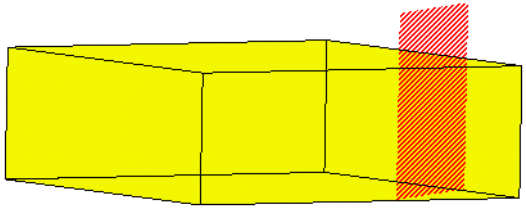
Step 7. Enter four vertices to define the clipping polygon.

Each edge of the clipping polygon defines a clipping plane that is perpendicular to the clipping polygon and faces towards the center of the polygon. (You will enable clipping edges in the *Clip Attributes* dialog box later in this procedure.)

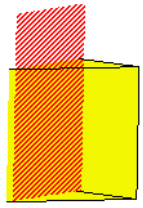


The clipping polygon itself is the clipping node's backplane.Unlike normal polygons, clipping polygons are not automatically converted to front-fac-

ing. The front of the clipping polygon is the side on which the vertices appear in counter-clockwise order. This is significant if you enable backplane clipping in the *Clip Attributes* dialog box, which inhibits the display of geometry behind the clipping polygon.

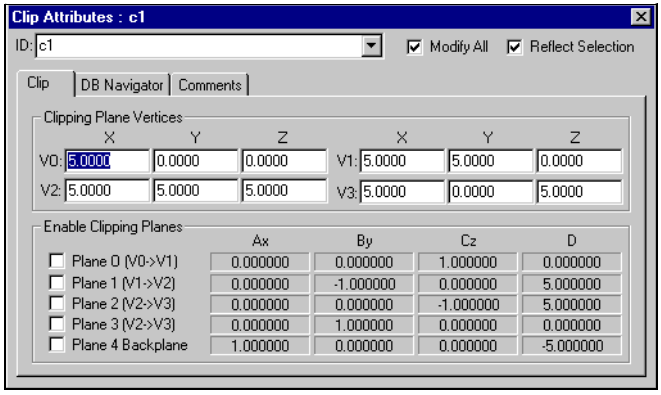


Before backplane clipping is enabled, the entire figure is displayed



After backplane clipping is enabled, only the portion of the figure in front of the clipping plane is displayed

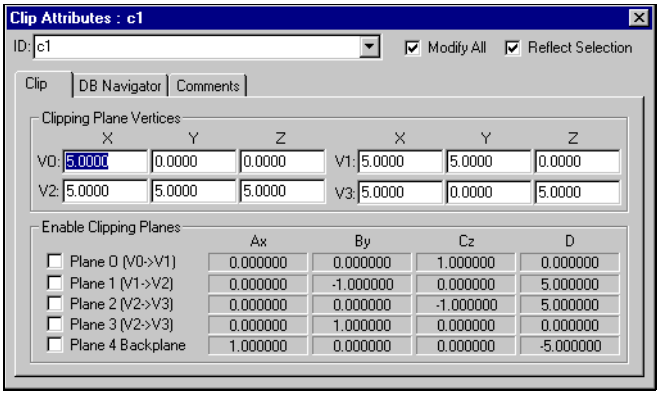
Step 8. In the Hierarchy view, double-click the new clipping node. The *Clip Attributes* window appears.



Step 9. In the **Enable Clipping Planes** area, choose the planes you want to enable. The Graphics view updates to show the effect of enabling each clipping plane.

Clipping Node Attributes

Clip attributes appear when you double-click a clipping node or select a clipping node and choose **Modify Attributes** (Ctrl+=) from the **Attributes** menu. The **DB Navigator** and **Comments** panels are common to every node type and are described in the *MultiGen Creator User's Guide*. The **Clip** panel has attributes that are specific to clip nodes.



ID

Specifies the identifier (name) of the node.

Vertex coordinates 0-3

Specify the four vertices of the clipping polygon. The edge between each pair of vertices defines a clipping plane (perpendicular to the clipping polygon and facing towards the center of the polygon). The clipping polygon, itself, defines the clipping node's backplane.

Enable Clipping Planes

Independently enables and disables each clipping plane. Geometry is clipped if it is attached to the clipping node and placed behind an enabled plane. For reference, the equation of each clipping plane appears to the right of its name.

Plane 4 Backplane enables backplane clipping, which clips geometry that is attached to the node and placed behind the clipping plane.

Nesting Clipping Nodes

Clipping nodes can be nested, that is, one clipping node may be a descendant of another clipping node in the hierarchy. However, among nested clipping nodes, a maximum of six clipping planes can be enabled at one time. (This restriction does not apply to clipping nodes that are not nested in the hierarchy.) The results of enabling more than six clipping planes among nested clipping nodes is undefined.

Displaying Clipping Polygons



Click **Draw Clip Nodes** in the View panel to toggle the display of clipping polygons on or off in the Graphics view of the database. Clipping planes appear as areas of shaded red lines and are visible only from the forward-facing side.

The clipping polygon plus each of its four edges define five clipping planes. **Draw Clip Nodes** displays the defining polygon but not the actual clipping planes.

Using the Font Editor

The **Font Editor** creates and modifies the characters of two-dimensional text fonts. These characters can be composed of solid polygons or unclosed wireframe polygons. You can use the fonts you create and edit in the **Font Editor** to create labels in any of the gauges available from the **Instruments** menu, and you can also place the text anywhere in your database using **Instruments/Place Text**.

> **Note:** The fonts used with the **Font Editor** are not the same as the three-dimensional fonts composed of OpenFlight character models that are required by the **3D Text** tool in the **Geometry** toolbox.

Editing a Font

To create a new font, you must edit an existing font. Creator includes two fonts, `paradigmfont.dat` and `nasafont.dat`, which are installed by default in the same directory as the Creator application.

To edit a font:

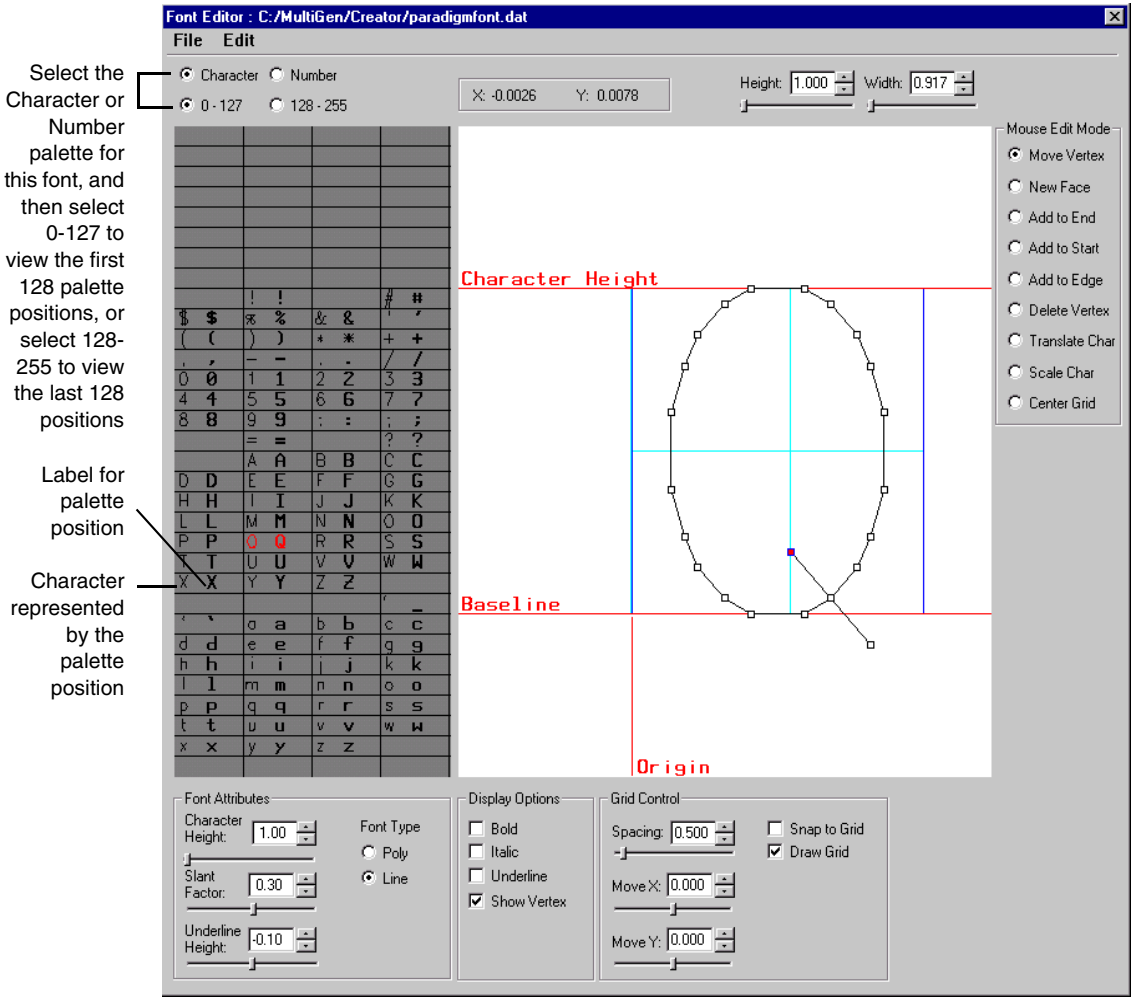
Step 1. Select **Instruments/Font Editor**. A dialog prompts you to choose a font file. By convention, font files have the extension `.dat`.

Step 2. Select a font file and click **OK**. The *Font Editor* window appears.

Step 3. On the left side of this window is the character palette, which shows half of the 255 possible characters or numbers at one time. If the palette position has a label, the label appears on the right. The character that is represented by that position is shown on the left. Not all positions will be filled or labeled.

To switch between character and number palettes, click either **Character** or **Number**.

To switch between the first and last set of palette positions, click either **0-127** or **128-255**.

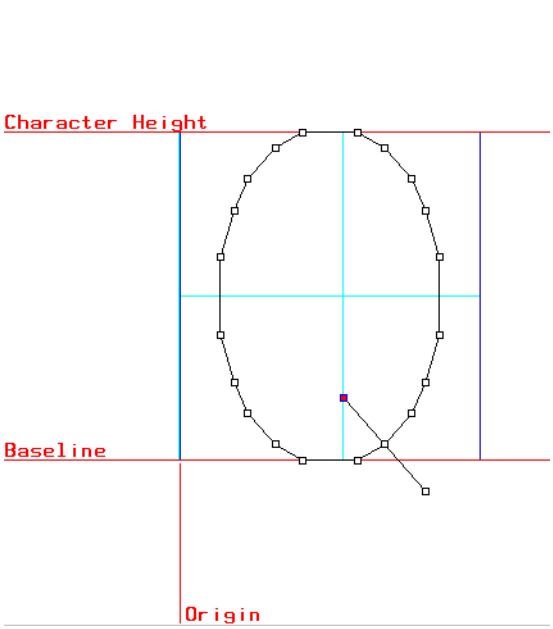


Step 4. Click a palette position to select its character for editing. The character appears in the preview area to the right of the character palette.

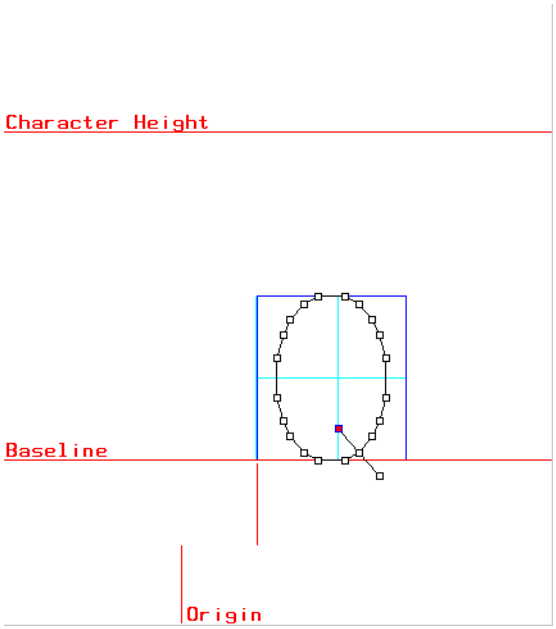
Step 5. Change the **Font Attributes**, if necessary:

The preview area shows the **Character Height** and **Baseline** indicators for the *font* (not for the character). To adjust the distance between these two markers, edit the **Character Height** setting, which is located below the character palette.

When you change the **Character Height**, the preview of the character you are editing adjusts to fit the new value; the **Character Height** and **Baseline** indicators in the preview area do not move.



Character Height for the font equals 1; Height of the character being edited equals 1.



Character Height for the font is changed to 2; Height of the character being edited still equals 1.

The **Slant Factor** specifies how much to slant italic characters. You see the effect only if you set the **Italic Display Option**.A value of 1 equals a 45 degree slant.

The **Underline Height** determines the offset from the baseline to an underline, if the character is underlined. You see the effect only if you set the **Underline Display Option**

Bold displays the character in boldface.

Show Vertex displays the vertices of the character. The first vertex of each face is drawn in magenta.

Step 6. Set the **Height** and **Width** of the character you are editing by adjusting the sliders at the top of the window. The **Character** and **Baseline** indicators in the preview area show the new size of the character relative to the font.

Step 7. Set the **Grid Controls**.

Spacing sets the distance, in database units, between grid marks in the preview area.

Move X and **Move Y** specify the position of the grid relative to the origin. Any offset will be applied when you place the character in the database.

Snap to Grid forces all vertex moves to the nearest grid position.

Draw Grid displays the grid in the preview area.

Step 8. Select a **Mouse Edit Mode** to specify what kind of edit you want to make to the character, and then click in the preview area to position vertices. The **X:** and **Y:** values displayed near the top of the window indicate the currently-selected vertex's offset from the origin. If you make a mistake, use commands in the **Edit** menu to undo the operation.

Click **Move Vertex** and then click a point in the preview area to move the closest vertex to that point. In this mode you can also drag a vertex to a new position.

Click **New Face** and then click a point in the preview area to enter the first vertex of a new face. The **Mouse Edit Mode** changes to **Add to End**.

Click **Add to End** and then click a point in the preview area to add a vertex to the end of the polygon that is closest to the place you click.

Click **Add to Start** and then click a point in the preview area to add a new vertex to the end of the polygon that is closest to the place you click. To add a vertex to a specific polygon, click **Add to Start**, click very close to the edge of the polygon to create a new vertex, and then drag the new vertex into position.

Click **Add to Edge** and then click a point in the preview area to add a vertex to the edge of the polygon that is closest to the point you click.

Click **Delete Vertex** and then click a point in the preview area to delete the vertex that is closest to the point you click.

Click **Translate Char** and then click a point in the preview area to reposition the entire character. The direction and distance of translation are determined by the direction and distance from the point where the baseline crosses the origin to the next point you click in the preview area.

Click **Scale Char** and then drag the pointer in the preview area to change the size of the character.

Click **Center Grid** and then click a point in the preview area to align the grid to that point.

Step 9. Select **File/Save** to save your edits. You can select another character for editing, or select **File/Close** to close the **Font Editor**.

Placing 2D Text in the Database

To place text in the database, you can create a texture pattern that contains the text you need, or you can create polygonal text—text polygons that are part of the geometry of the database. **Instruments/Place Text** allows you to place 2D polygonal text anywhere in the database.

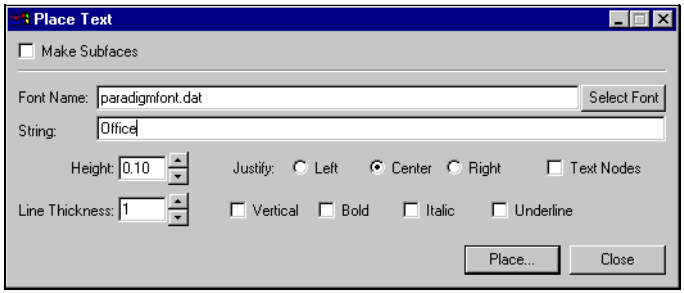
Note: To place 3D text, you must use the **3D Text** tool (see "3D Text Tool" on page 119).

To place 2D text:

Step 1. Select the object or face on which you want to place the text and click **Parent**. If you want to the text polygons to be subfaces of the face on which you place them, you must set a face as the parent.

Step 2. Open the Color palette and select a color for the text.

Step 3. Select **Instruments/Place Text**. The *Place Text* dialog box appears.



Step 4. In the **String** field, type the text you want to place in the database.

Step 5. If you want the text polygons to be subfaces, set **Make Subfaces**.

Step 6. Click **Select Font** and choose a font file from the dialog box that appears.

For information about creating and editing font files, see "Using the Font Editor" on page 807.

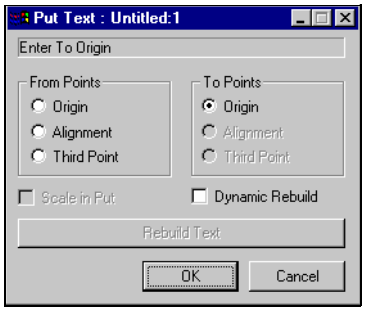
Step 7. Select a **Height** and **Line Thickness** for the characters, in database units.

Step 8. Select one of the **Justify** options to set the alignment of the characters.

Step 9. Set any font effects (**Vertical**, **Bold**, **Italic**, and **Underline**) that you want to apply to the text.

Step 10. If you want to insert the text string as dynamic text in a text node, set **Text Nodes** (see "Creating Dynamic Text" on page 814). This choice supersedes **Make Subfaces**.

Step 11. Click **Place**. The **Put Text** dialog box appears.



Step 12. Click points int the Graphics view to set the **Origin**, **Alignment**, and **Third Point** coordinates, which position the text string in the database.

Step 13. Set **Dynamic Rebuild** to rebuild the text automatically if you change one of the parameters in the **Place Text** dialog box (for example, if you change **Justify** from **Center** to **Right**), otherwise, click **Rebuild Text** when you want to manually rebuild the text polygons.

Note: **Scale in Put** is never available when using **Put** to place text.

Step 14. Click **OK**.

Creating Dynamic Text

Text nodes let you set up placeholders in the database for text strings that are changed by the realtime software. Before you add dynamic text to your database, make sure it is supported by your realtime software.

Consider the example of a digital speedometer in the dashboard of a car. The number displayed on the speedometer changes as the car accelerates or decelerates. To model an instrument like this with static text, you could attach the digits 0-9 in each column of the speedometer and use a switch node to turn the digits on or off.

The same effect can be achieved with dynamic text by placing a text node in the database and assigning the numerical range of the speedometer to the node.

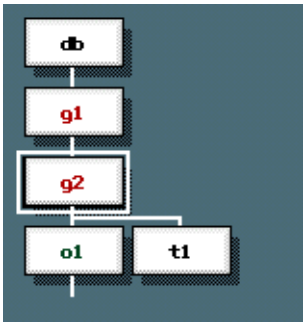
Creating Text Nodes

Text nodes are created automatically when you set the **Text Nodes** option in the *Place Text* dialog box (see "Placing 2D Text in the Database" on page 812). To create a text node manually:

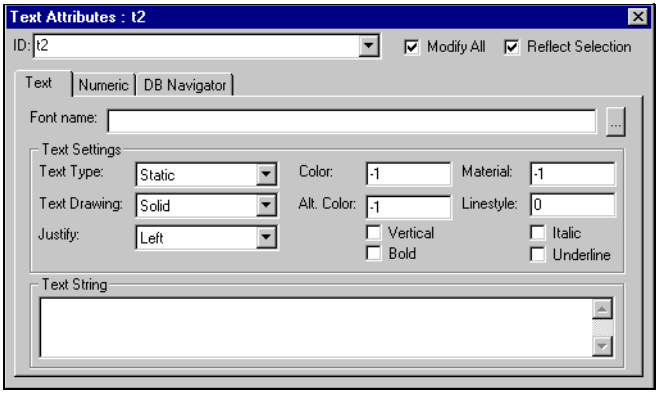
Step 1. Set a group node as the parent.

Step 2. Open the **Create** toolbox.

Step 3. Click the **Create Text** tool. A text node appears under the parent group node.



Step 4. Double-click the text node, or select it and choose **Attributes/Modify Attributes (Ctrl+=)**. The *Text Attributes* window appears.



Step 5. In the **Font Name** field, enter the path and name of the font file you want to use, for example `c:/MultiGen-Paradigm/creator/paradigmfont.dat`.

Step 6. In the **Text Field**, type the text string that you want to appear in the database.

Step 7. Select a **Text Type**

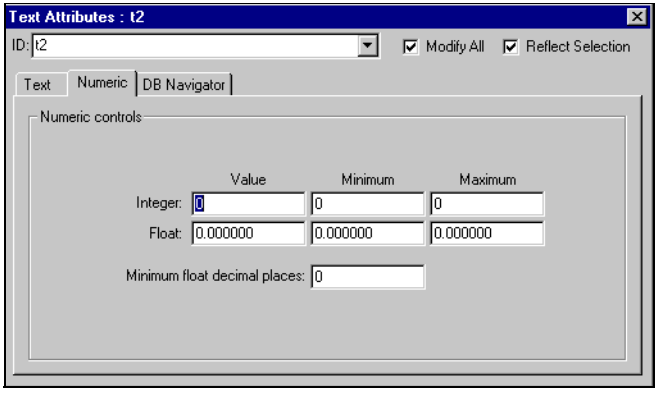
Static is not supported for dynamic text. (Remember, this window contains text attributes for all text types.)

Text indicates that this node contains dynamic text.

Integer indicates that this node is integer-valued. The **Value** in the **Integer** field of the **Numeric** panel of this window is evaluated, converted to an ASCII string, and placed in the **Text Field** each time the node is drawn.

Float indicates that this node is floating point-valued. The **Value** in the **Float** field of the **Numeric** panel of this window is evaluated, converted to an ASCII string, and placed in the **Text Field** each time the node is drawn.

Step 8. If you selected **Integer** or **Float**, click the **Numeric** tab and enter the appropriate values.



Step 9. In the **Text** panel, select a **Text Drawing** method:

Solid generates text polygons that are filled with the Color assigned to the text node.

Closed Wireframe generates closed wireframe text polygons. The Color assigned to the text node determines the color of the wires (edges).

Unclosed Wireframe generates unclosed wireframe text polygons. The Color assigned to the text node determines the color of the wires (edges).

Alt Color Wire/Solid generates text polygons filled with the color assigned to the text node but outlined with edges in the node's **Alt(ernate) Color**.

Step 10. Set a **Justify** option to insert the text to the **Left**, **Right**, or **Center** of the local origin of the text node.

Step 11. Enter palette indexes for the **Color**, **Alt Color**, **Material**, and **Line Style** you want to apply to the text.



Color contains the Color palette index of the color assigned to the text node. To assign this color, select the text node and click **Insert Color** in the **Properties** toolbox or choose **Edit/Insert Color (Ctrl+R)**.



Alt Color contains the Color palette index of the alternate color assigned to the text node. To assign this color, select the text node, press Alt, and click **Insert Color** in the **Properties** toolbox or press Alt and choose **Edit/Insert Color (Ctrl+R)**.

Material specifies the Material palette index of the material assigned to the text node. The material assignment only affects solid shaded surfaces. For more information about materials, see the *MultiGen Creator User's Guide*.

Line Style specifies the Line Style palette index assigned to the text node. **Line Style** defines the appearance of the polygon wires, or edges. For information about line styles, see "Line Styles" on page 821.

Step 12. Set all the **Text Style** options you want to assign to the text.

Vertical places the text in a vertical column.

Bold assigns a line thickness of two to the text. This overrides the Line thickness value.

Italic displays the text in italics.

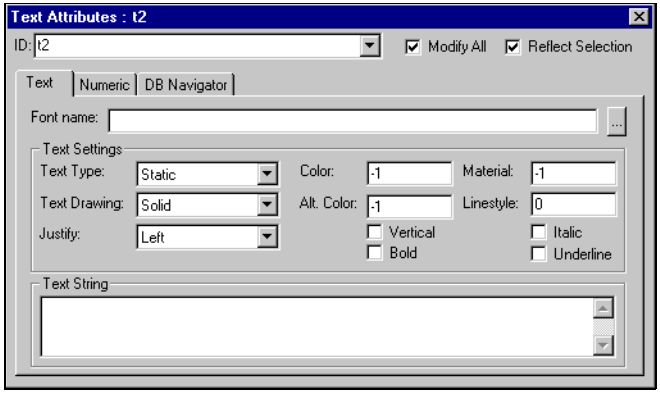
Underline underlines the text.

Step 13. Close the *Text Attributes* window. The text is displayed in the Graphics view.

Text Node Attributes

A text node places dynamic text strings in the database. The text is dynamic because the value of the text node is evaluated each time the text node is redrawn. Because a realtime database scene is redrawn many times every second, changing the contents of the text node results in an immediate change in the display of the text.Text nodes are created automatically when you set the **Text Nodes** checkbox in the *Place Text* dialog box (see "Placing 2D Text in the Database" on page 812) or when you use the Create Text tool (see "Creating Text Nodes" on page 815).

To display text node attributes, double-click a text node, or select a text node and then choose **Attributes/Modify** (Ctrl+=). The *Text Attributes* window appears.



Text Attributes

The following attributes appear in the **Text** tab in the *Text Attributes* window.

Font Name

Specifies the name of the font file used to generate the text string.

Text Type

Controls whether or not this text node contains an ASCII string, an integer value, or a floating point value.

- **Static** text strings are not supported.
- **Text** indicates a text node contains a dynamic text string.
- **Integer** indicates the text node is integer-valued. In this case, the Integer value located in the **Numeric** panel is evaluated, converted to an ASCII string, and placed in the Text field each time the text node is drawn.
- **Float** indicates the text node is floating point-valued. In this case, the Float value in the **Numeric** panel is evaluated, converted to an ASCII string, and placed in the Text field each time the text node is drawn.

See "Numeric Attributes" on page 820.

Text Drawing

Selects the drawing type of each polygon in the dynamic text. Note that most fonts are designed to be used with a specific drawing type.

- **Solid** generates text polygons that are filled with the Color assigned to the text node.
- **Closed Wireframe** generates closed wireframe text polygons. The Color assigned to the text node determines the color of the wires (edges).
- **Unclosed Wireframe** generates unclosed wireframe text polygons. The color assigned to the text node determines the color of the wires (edges).
- **Altcolor Wire/Solid** generates text polygons filled with the color assigned to the text node but outlined with edges in the node's Alt(ernate) color.

Justify

Inserts the text to the left, right, or center of the local origin of the text node.

Color

Contains the Color palette index of the color assigned to the text node. To assign this color, select the text node and click **Insert Color** in the **Properties** toolbox or choose **Edit/ Insert Color** (Ctrl+R).

Alt. Color

Contains the Color palette index of the alternate color assigned to the text node. To assign this color, select the text node, press Alt, and click **Insert Color** in the **Properties** toolbox or press Alt and choose **Edit/Insert Color** (Ctrl+R).

Material

Specifies the Material palette index of the material assigned to the text node. Material assignment only affects solid shaded surfaces.

Linestyle

Specifies the Line Style palette index assigned to the text node. The line style defines the appearance of the polygon wires, or edges (see "Line Styles" on page 821).

Font Effects

Control the formatting of the text.

Vertical places the text in a vertical column. **Bold** assigns a line thickness of two to the text. This overrides the **Linestyle** value. **Italic** displays the text in italics. **Underline** underlines the text.

Text String

Contains the ASCII string that generates text in the database. If **Text Type** is **Integer** or **Float**, this field contains the ASCII string corresponding to the text node's current **Value**.

Numeric Attributes

These attributes appear in the **Numeric** tab in the *Text Attributes* window. Numeric attributes control the insertion of numeric-valued text when the node's **Text Type** attribute is set to **Integer** or **Float**.

You supply a starting value and a range of allowable values. The realtime system evaluates the integer or floating point value of the text node, converts the value to an ASCII string, and places the ASCII string in the Text field each time the text node is drawn in the database scene.

Value

Specifies the starting value for the text node.

Minimum

Sets the lowest value in the range at which the text is displayed.

Maximum

Sets the highest value in the range at which the text is displayed.

Minimum Float Decimal Places

Specifies the number of decimal places used when a floating point value is converted to a text string and placed in the **Text String** field in the **Text** panel.

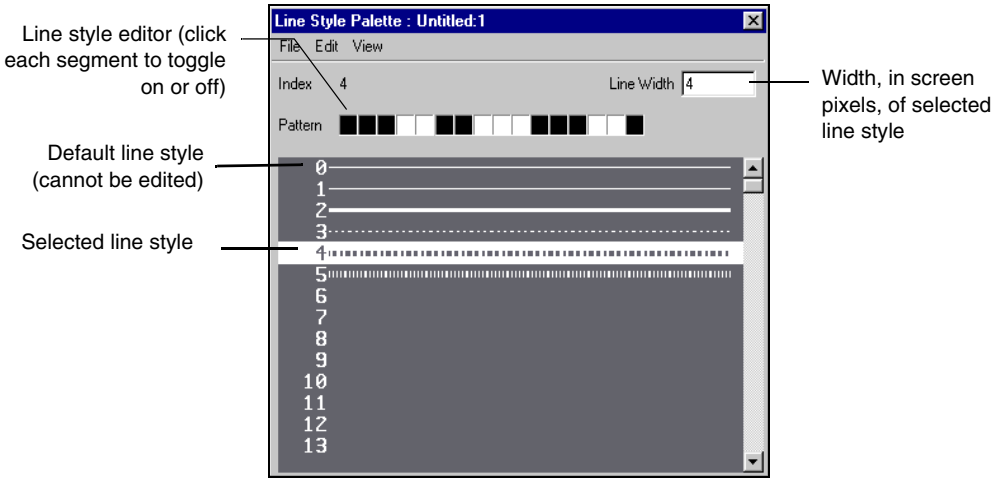
Line Styles

Line style controls the appearance of edges for each face displayed in Wireframe or Wireframe over Solid drawing mode. It also controls the outline drawn around faces selected in the database.

Line Style Palette

Line styles are defined in the database Line Style palette. Each database file has its own Line Style palette. To open the Line Style palette, choose **Palettes/Line Style**. The palette contains a line editor, plus space for defining up to 255 black-and-white lines. Line 0 is the Creator default, and cannot be edited.

Each line segment displayed in the Line Style palette corresponds to one screen pixel, meaning it will always be displayed with one pixel on the workstation, regardless of the eyepoint.



To Create a New Line Style

Step 1. Choose **Edit/New** to select the first empty position in the Line Style palette.

Step 2. Toggle pixels on or off in the selected line by clicking them in the **Pattern** field of the Line Style editor.

Step 3. Set the width of the selected line by editing the **Line Width** value.

To Modify a Line Style

Step 1. In the Line Style palette, select the line style you want to edit.

Step 2. Toggle pixels on or off in the selected line by clicking them in the **Pattern** field of the line style editor.

Step 3. Set the width of the selected line by editing the **Line Width** value.

Alternatively, you can use the **Copy** and **Paste** functions to modify a line style:

Step 1. Select a line style similar to what you want and click **Edit/Copy.**

Step 2. Select the line style you want to modify and click **Edit/Paste**

Step 3. Set the width of the selected line by editing the **Line Width** value.

To Delete a Line Style

Select a line style in the palette and choose **Edit/Delete**.

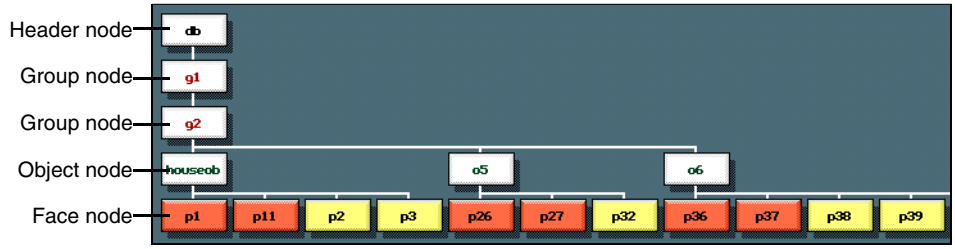
Assigning a Line Style

Line styles are associated with face attributes. Each line style in the palette is assigned a Line Style palette index. When a line style is applied to a face, the line style index number is assigned to the face attribute. All faces are created with a line style attribute of 0 (the index of the default line style).

Assigning a line style to a face requires the use of two tools in the **Properties** toolbox, **Insert Line Style** and **Line Style From Face**.

23 OpenFlight Nodes and Attributes

The database hierarchy defines the relationship of the nodes in a database to each other. The nodes in the hierarchy are linked together in an inverted tree, which grows from a single database header node at the top, down to object, face, and vertex nodes at the bottom. (Vertex nodes do not appear in the Hierarchy view.)



Each node in the tree contains information that the realtime system needs to be able to display the database. This information consists of:

- Attributes that uniquely identify the node and control how it is processed.
- Links to the children of the node.

For more information on database hierarchy, see "Working in the Hierarchy View" on page 279. For a description of each type of OpenFlight node, see "Database Node Types" on page 292.

Every node in the database has an *Attributes* window in which you can view and edit the values of its attributes and navigate through the database hierarchy. The following topics describe:

- OpenFlight node types, such as groups, objects, and faces, that organize the database into pieces that are both easy to edit and are optimized for realtime display.
- Attributes that control the behavior and appearance of each node.
- Attribute Resolution, a function that removes inconsistencies in the color, material, and texture palettes of different databases that will be displayed together.

- Attribute searching, which lets you search for nodes in the database that have specific attribute values assigned to them.
- Attribute reports.
- An exercise to help you to learn how to use some of the features of attribute windows.

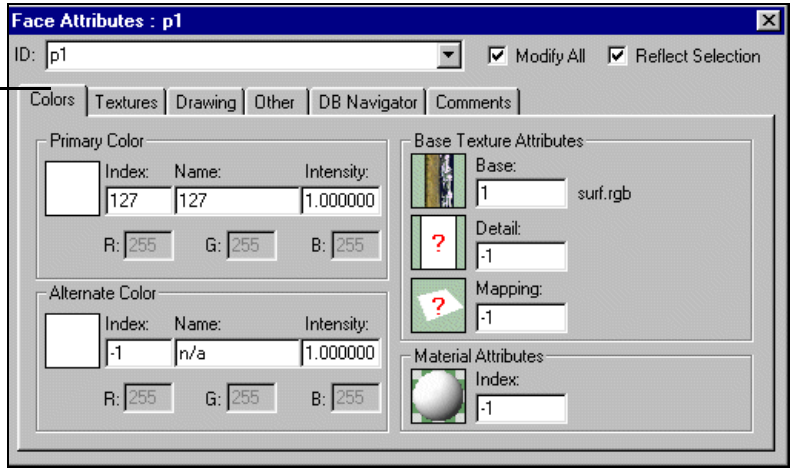
Opening an Attributes Window

Creator stores attribute data for every type of node in the database. These attributes define the graphical, physical, and theoretical characteristics of the node. To display and edit attributes, select an item and choose **Attributes/Modify Attributes** (Ctrl+=), or double-click the item in the Graphics or Hierarchy view.

To open an *Attributes* window, do one of the following:

- Select an item and choose **Attributes/Modify Attributes** (Ctrl+=).
- If nothing is selected when you choose **Attributes/Modify Attributes**, the *Header Attributes* window appears. When more than one item is selected, the node IDs appear in a list at the top of the *Attribute* window.

This list shows all the faces that were selected when the Face Attributes window was opened.



- Double-click an item in the Graphics or Hierarchy view. This selects the item and opens its *Attributes* window.

- In the Graphics view, the type of node selected is determined by the modeling mode. For example, In object modeling mode, you can select objects and display object attributes.
- In the Hierarchy view, the node you click becomes selected and its attributes are displayed, regardless of the modeling mode.

Closing an Attributes Window



To close an *Attributes* window, click the **Close**button in the upper-right corner of the window.

To close all open *Attributes* windows, choose **Attributes/Close Attribute Windows**.

Reflect Selection

Although you can open as many *Attributes* windows as you like, you may prefer to keep only a single window open. To do this, set the **Reflect Selection** checkbox in the upper-right corner of any *Attributes* window. Now each time you select a node in the Hierarchy view, the *Attributes* window changes to show the attributes of the selected node. This is convenient when you want to minimize the amount of screen space used by *Attributes* windows or navigate through the database hierarchy using the DB Navigator Panel (see "DB Navigator Panel" on page 828).

If you clear the **Reflect Selection** checkbox, a separate *Attributes* window appears each time you double-click a node. This is useful if you want to compare the attributes of different nodes.

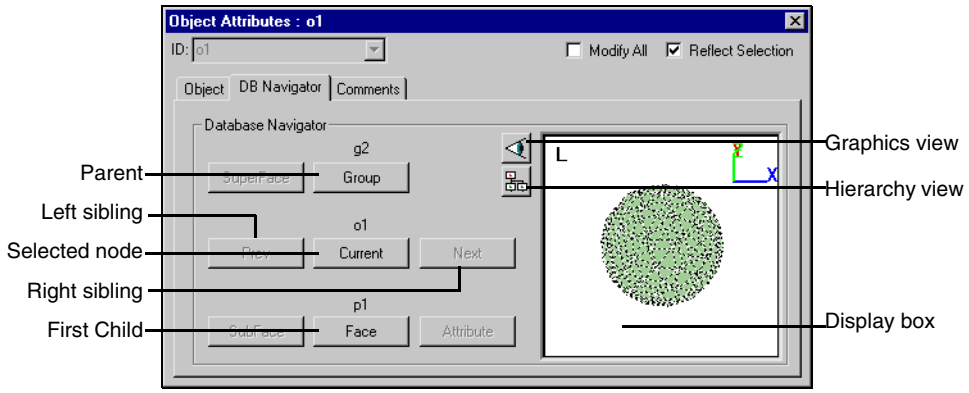
Modify All

The **Modify All** checkbox applies the changes you make in the open *Attributes* window to *every node (of the same type) that is currently selected*. This is useful when you want to change a number of nodes at the same time. For example, you might select several group nodes and turn on all their bounding boxes at once.

Make sure you clear **Modify All** when you are done editing a group of attributes. If you forget, you may find yourself making unexpected changes in the database.

DB Navigator Panel

All *Attributes* windows have a DB Navigator panel that displays the geometry or hierarchy of the node and contains buttons that open the node's parent, left sibling, right sibling, and first (left-most) child. To access this panel, click the **DB Navigator** tab in any *Attributes* window.



Navigation Buttons

The following buttons appear in the DB Navigator panel

Superface

Opens the *Attributes* window of the superface of the current node. This button is available only when viewing the attributes of a face that has another face as its parent.

Parent

Opens the *Attributes* window of the node above the current node. The label of this button changes to show the type of the parent node. For example, the parent of an object node is (usually) labeled **Group**.

Prev

Opens the *Attributes* window of the node to the immediate left of the current node.

Next

Opens the *Attributes* window of the node to the immediate right of the current node.

Subface

Opens the *Attributes* window of the subface of the current node. This button is available only when viewing the attributes of a face that has another face as its child.

Child

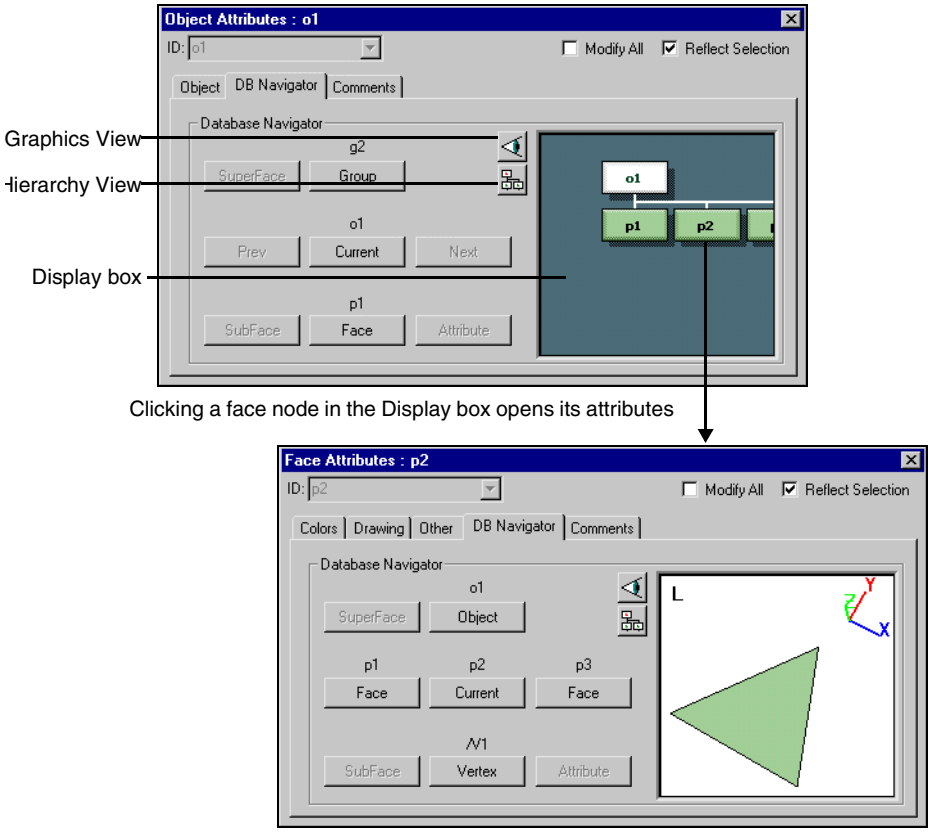
Opens the *Attributes* window of the first (extreme left) child of the current node. The label of this button changes to show the type of the child node. For example, the child of an Object node is labeled **Face**.

Attribute

Edits the SmartScene® attributes of the current node. (The SmartScene Option is not available in Creator. **Attribute** is included for backward compatibility with OpenFlight files created in MultiGen II.)

Display Box

The Display box in the DB Navigator panel shows either a Graphics view or a Hierarchy view of the current node, depending on whether you select the **Graphics View** button or the **Hierarchy View** button. Clicking a node in the Display box Hierarchy view opens that node in the *Attributes* window.



Comments Panel

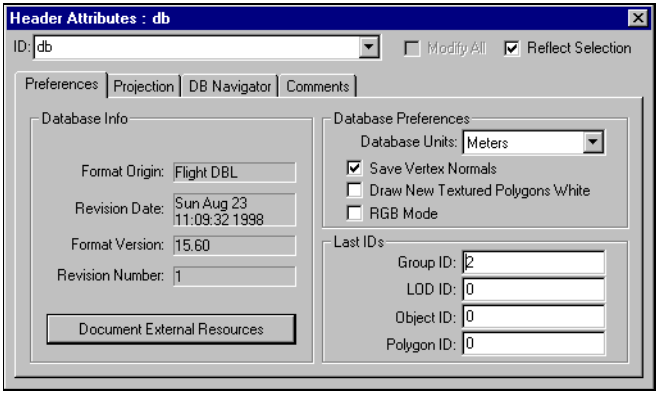
Each *Attributes* window has a **Comments** tab. The Comments panel is generally used to add modeling comments to the database.

Be careful before entering comments. The **Modify Delaunay** function in the Terrain option uses group comments to store information on how terrain polygons are generated Some realtime applications also use this field to store attributes that are not normally defined for OpenFlight.

Header Attributes

The database header is the top node in the database hierarchy. It is created automatically with each new file and is represented by the node labeled **db** in the Hierarchy view.

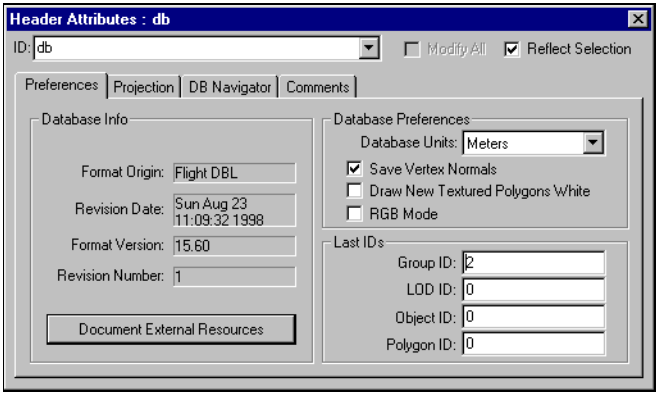
To display the *Header Attributes* window, double-click an empty area in the Graphics or Hierarchy view, double-click the **db** node in the Hierarchy view, or choose **Attributes/Modify Attributes** (Ctrl+=) with nothing selected.



The *Header Attributes* window has four panels: **Preferences**, **Projection**, **DB Navigator**, and **Comments**. The following topics describe these panels.

Header Attributes: Preferences Panel

The **Preferences** panel in the *Header Attributes* window has the following parameters:



Format Origin

Indicates the database format. This is useful if you are importing or exporting databases to or from other formats.

Revision Date

Displays the date and time the database was last saved.

Format Version

Displays the OpenFlight version number of the database.

Revision Number

Indicates the number of times the file has been saved to disk.

Document External Resources

Opens a *Status Log* window listing the external references and texture files referenced by the database.

Database Units

Selects the unit of measure for this file. Changing **Units** does not affect database coordinates. For example, when **Meters** is selected, a 10-unit long object is 10 meters long; changing **Units** to **Kilometers** makes the same object 10-kilometers long without modifying its vertices. The unit of measure for the file is displayed on the status bar, at the bottom right corner of the database window.

Save Vertex Normals

Controls whether vertex normals, calculated during shading, are saved with the database. The default is to save vertex normals.

Draw New Textured Polygons White

Controls whether new polygons that underlie texture patterns are white or colored. If this checkbox is set, the **Draw New Textured Polygons White** face attribute is set for each subsequently created face. This prevents the assigned polygon color from blending with texture patterns.

RGB Mode

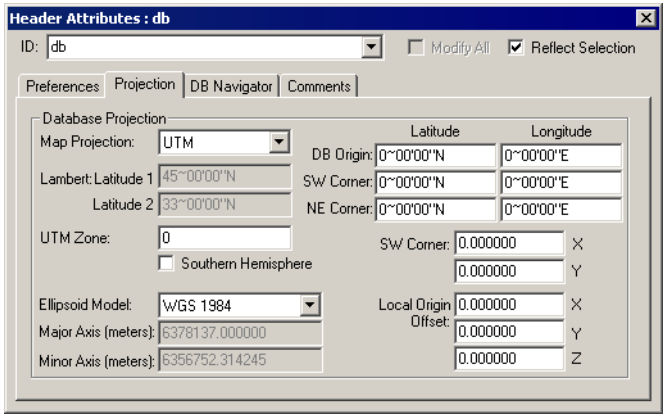
Enables the **RGB** face attribute. If this checkbox is cleared, Creator draws each face using its index into the color palette and the **RGB** face attribute is unavailable (see "RGB Mode" on page 408).

Last IDs

Shows the last number used for the ID of each indicated node type. For example, if is 3, the next group you create will be **g4**.

Header Attributes: Projection Panel

The **Projection** panel in the *Header Attributes* window has the following parameters:



Map Projection

Indicates the type of map projection used in a database created with the Terrain option. If you choose UTM, you can change the current UTM zone number, and indicate if the database is located in the southern hemisphere.

Ellipsoid Model

Indicates the type of ellipsoid used for the map projection. If you choose **User Defined Ellipsoid**, enter the Major Axis and Minor Axis for the ellipsoid.

SW Corner (of XY)

The database coordinates of the southwest corner. This information is generated automatically for databases created with the Terrain option.

DB Origin

Displays the latitude and longitude of the database origin. This information is generated automatically for databases created with the Terrain option.

SW Corner and NE Corner

Displays latitude and longitude at the boundaries of the database. This information is generated automatically for databases created with the Terrain option.

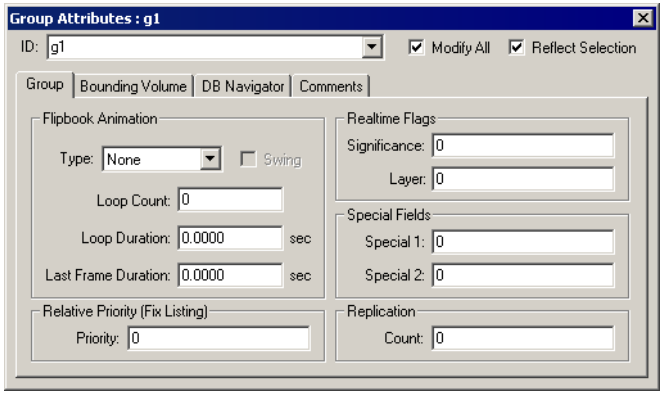
Lambert

Contains two latitude lines used to generate a Lambert projection. This information is generated automatically for databases created with the Terrain option.

Local Origin Offset

The offset x, y, and z coordinates of the tile origin from the database origin.

Group Attributes



The *Group Attributes* window has four panels: **Group**, **Bounding Volume**, **DB Navigator**, and **Comments**. The following topics describe these panels.

Group Attributes: Group Panel

The **Group** panel in the *Group Attributes* window has the following parameters:

Flipbook Animation

Selects the type of animation sequence attached to a group (see "Animation" on page 759).

Replication Count

Controls the number of copies of a group that are displayed in the Graphics view (see "Replicate Tool" on page 162).

Relative Priority

Controls the position of a node relative to its siblings. The hierarchy is periodically sorted so that relative priority among siblings increases from left to right in the Hierarchy view (see "Database Traversal Order" on page 688). By assigning numbers to groups, you can control how they are drawn using Fixed List or BSP drawing modes (see "Relative Priority" on page 688).

Significance

Provides a spare value to use with the realtime system. Significance is sometimes used to throw out nodes if the realtime system does not have time to process everything in the database. This field is ignored by Creator.

Layer

Provides a method of tagging sets of groups for the realtime system. This field is ignored by Creator.

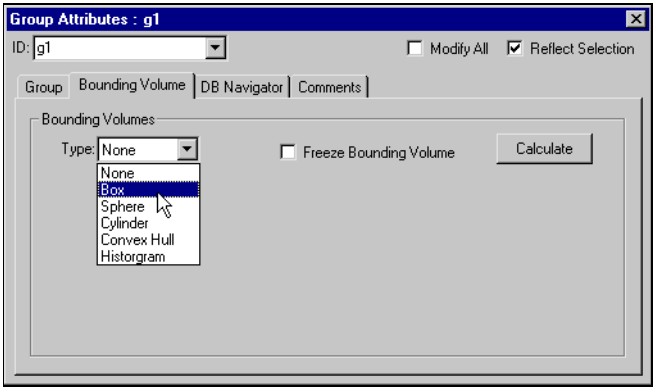
Special 1 and Special 2

Provide spare integer values to use with the realtime system. These fields are ignored by Creator.

Group Attributes: Bounding Volume Panel

Bounding volumes are relatively simple shapes that a realtime system uses to quickly estimate the geometric size and location of a group. For information on using bounding volumes, see "Bounding Volumes" on page 694.

Note: Different parameters are shown in the *Group Attributes* window. depending on which bounding volume type is selected.



Type

Selects the shape of the bounding volume. By default, **None** is selected. Selecting **Box**, **Sphere**, **Cylinder**, **Convex Hull**, or **Histogram** causes Creator to calculate the corresponding volume and display its parameters in the lower part of the *Group Attributes* window.

The fields in the Bounding Volume panel vary, depending on the type of bounding volume you choose.

Freeze Bounding Volume

Freezes the values that define the bounding volume. Creator recalculates bounding volumes each time the database is saved unless the **Freeze Bounding Volume** checkbox is set. You can specify your own bounding volume by turning on **Freeze** and editing the bounding parameters.

Calculate

Explicitly recalculates the bounding volume of the group. If **Freeze Bounding Volume** is set, **Calculate** does not change the **Center** and **Orientation** of the bounding volume.

Center

Defines the center of spherical and cylindrical bounding volumes. In a box bounding volume, this coordinate defines the center of rotation for **Orientation**.

Front Bottom Left and Back Top Right

Define diagonally opposed corners of a box bounding volume.

Orientation

Changes the orientation of the bounding volume by applying **Yaw** (rotation about the z-axis), **Pitch** (rotation about the x-axis), and **Roll** (rotation about the y-axis).

Radius

Defines the radius of a cylindrical or spherical bounding volume.

Height

Defines the height of a cylindrical bounding volume.

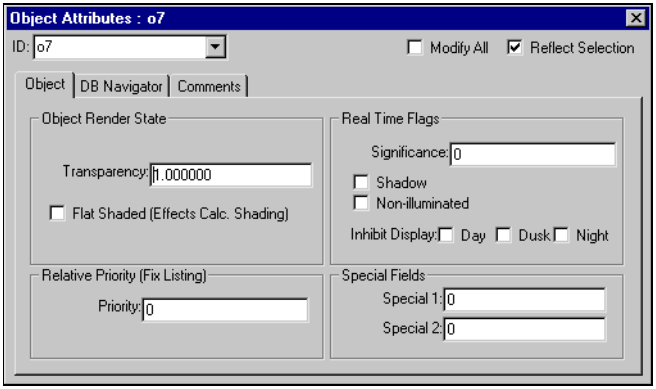
Number of Increments

Controls the number of divisions in a bounding histogram (a bounding volume made up of a grid of rectangular columns, each with a unique height).

Calculate Plateaus From

Selects whether the posts that define a bounding histogram are even at the bottom and vary along the top (if **Z maximum** is selected) or are even at the top and vary along the bottom (if **Z minimum** is selected).

Object Attributes



Object Attributes: Object Panel

The **Object** panel in the *Object Attributes* window has the following parameters:

Transparency (Range 0.0 to 1.0)

Defines the amount of transparency of an object. A value of 0.0 indicates that the object is completely transparent; a value of 1.0 indicates that the object is completely opaque. Transparency is inherited by faces and subfaces attached to the object.

Flat Shaded

Forces an object to be displayed in its assigned color. Not shaded.

Relative Priority

Controls the position of a node relative to its siblings. The hierarchy is periodically sorted so that relative priority among siblings increases from left to right in the Hierarchy view (see “Database Traversal Order” on page 688). By assigning numbers to groups, you can control how they are drawn in Fixed List or BSP drawing modes (see “Relative Priority” on page 688).

Significance

Provides a spare value to use with the realtime system. **Significance** is sometimes used to throw out nodes if the realtime does not have time to process everything in the database. This field is ignored by Creator.

Shadow

Indicates the object is a shadow description for a group. Normally, this means the object is a flat projection of a three-dimensional model that the realtime system can distort and project onto the ground.

Non-illuminated

Disables shading. Any shading information associated with this object is ignored by the realtime system.

Inhibit Display

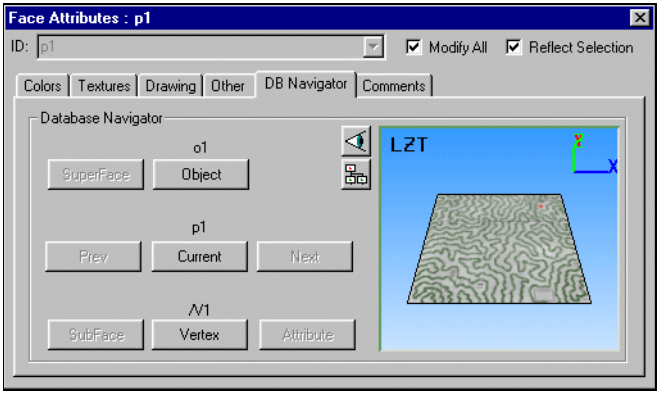
Indicates that an object should not be displayed during **Day**, **Dusk**, or **Night**. These checkboxes are provided for the realtime system and are ignored by Creator.

Special 1 and 2

Provide spare integer values for use with the realtime system. These fields are ignored by Creator.

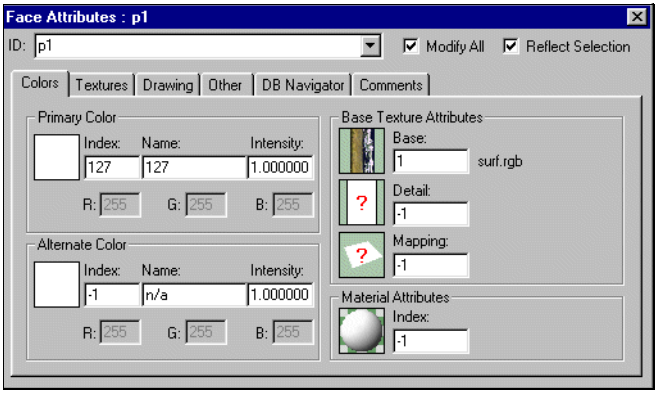
Face Attributes

Face attributes control the appearance and rendering characteristics of individual polygons (faces) in the database.To view and change the attributes of a face, double-click it or select it and choose **Edit/Attributes** (see "Exercise: Using Attribute Windows" on page 882).



Face Attributes: Colors Panel

The Colors panel in the *Face Attributes* window is divided into four groups: **Primary Color**, **Alternate Color**, **Base Texture Attributes**, and **Material Attributes**.



Primary Color: Index

Contains the Color palette index of the color assigned to this face (see “Color Palette Index” on page 405). A value of -1 indicates that no primary color has been assigned, in which case the primary color defaults to white.

Primary Color: Name

Contains the symbolic color name of the face. If the face's **Index** and **Name** do not reference the same color in the palette, **Name** takes precedence (see "Color Names" on page 402).

Primary Color: Intensity

Contains an intensity multiplier (range 0.0 to 1.0) for the **Index** assigned to the face. If this value is less than 1.0, it applies the full intensity value of the color (see "Color Intensity" on page 406).

Primary Color: RGB

Contains RGB color values that specify the color of the face when RGB drawing mode is enabled in the database header attributes (see "RGB Mode" on page 408). When the database is in RGB mode, the **Index**, **Name** and **Intensity** are ignored (see "RGB Mode" on page 408).

Alternate Color: Index

Contains the Color palette index for the face's alternate color (see "Color Palette Index" on page 405). A value of -1 indicates that no alternate color has been assigned, in which case the alternate color defaults to white.

Alternate Color: Name

Contains the symbolic color name of the face. If the face's **Index** and **Name** do not reference the same color in the palette, **Name** takes precedence (see "Color Names" on page 402).

Alternate Color: Intensity

Contains an intensity multiplier (range 0.0 to 1.0) for the **Index** assigned to the face. If this value is less than 1.0, it applies the full intensity value of the color (see "Color Names" on page 402).

Alternate Color: RGB

Contains RGB color values that specify the color of the face when RGB drawing mode is enabled in the database header attributes. When the database is in RGB mode, the **Index**, **Name** and **Intensity** are ignored (see "RGB Mode" on page 408).

Texture Attributes: Base

Contains the Texture palette index of the texture applied to this face (see "Texture Palettes" on page 422). If no texture has been applied, this field contains a value of -1.

Texture Attributes: Detail

Contains the Texture palette index of the detail texture applied to this face (see "Detail Texture" on page 530). If a detail texture has not been applied, this field contains a value of -1.

Texture Attributes: Mapping

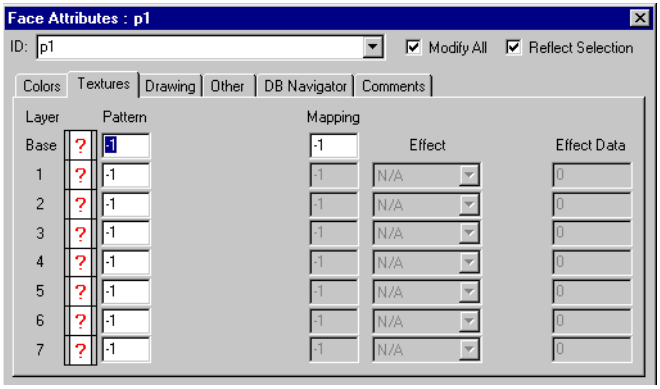
Contains the index value of the selected texture mapping in the texture mapping palette (see "The Texture Mapping Palette" on page 516). If no texture has been applied, or if an applied texture has not been added to the Texture Mapping palette, this field contains a value of -1.

Material Attributes: Index

Contains the index value of the material assigned to the face (see "The Material Palette" on page 387). If no material has been applied, this field contains a value of -1.

Face Attributes: Textures Panel

The Textures panel in the *Face Attributes* window contains the parameters for the MultiTexture feature that control which texture layers are applied to a face and how each texture layer blends with other texture layers. For more information about the MultiTexture feature, see "MultiTexture" on page 470.



Layer

Indicates the layer number of the texture. Layer 0 from the Texture list in the View panel is the **Base** layer in this panel.

Pattern

Indicates the Texture palette index number. The index number for each texture is also located in the *Texture Palette* window.

Mapping

Indicates the Texture mapping index number. A **-1** appears if the texture does not have a texture map in the Texture Mapping palette.

Effect

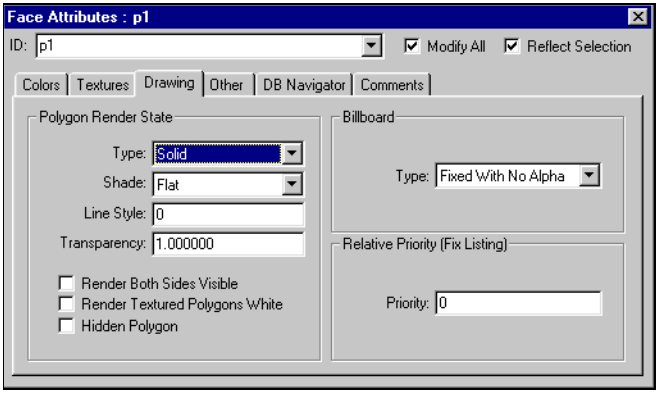
Sets the rendering for multitexture effects to **texture env**, **bump map**, or **custom** for the texture layer. Creator renders multitexture effects using **texture env**, which blends the texture layer with the underlying layers using the texture environment algorithm that you set in the *Texture Attributes* window. You can select **bump map** if your runtime system creates bump maps, or **custom** for an effect that you have defined. If you select **custom**, you can also enter a number that your runtime system uses to render the effect.

Effect Data

Contains a number for your runtime system to render multitexture effects. This text field is optional.

Face Attributes: Drawing Panel

The Drawing panel in the *Face Attributes* window contains the parameters that control how the face is drawn in Creator and rendered in the realtime system.



Type

Sets the drawing mode for this face to **Solid**, **Closed Wireframe**, or **Wireframe**. This setting overrides the drawing mode set with the **Draw Mode** control in the View panel.

Shade

Sets the shading for this face to **Flat**, **Lit**, **Gouraud**, or **Lit Gouraud**.

Line Style

Contains the index number in the database Line Style palette for the line style of the face's edges. This feature is available only with the CreatorPro Instrumentation option. See "Line Styles" on page 821 for more information.

Transparency

Indicates the transparency of the face, in the range of 0.0 (clear) to 1.0 (opaque), on platforms that support alpha blending. The net transparency applied to a face, as drawn by Creator, is calculated by multiplying the **Transparency** field by the transparency component of the material assigned to the face (if any). For example, if face transparency is 0.5, and the alpha value of a material assigned to the face is 0.5, the net transparency applied is 0.25 (more transparent). Changes to the face transparency automatically propagate to subfaces.

Render Both Sides Visible

Turns on the display of color and texture on both the frontfacing and backfacing sides of the polygon. (By default, only frontfaces are visible.)

Render Textured Polygons White

Overrides the assigned face color and makes the face white if it is textured. Creator sets this checkbox automatically when the **Draw New Textured Polygons White** header attribute is set (see "Draw New Textured Polygons White" on page 833). Setting this checkbox prevents face color from blending with the colors of applied textures.

Hidden Polygon

Controls whether this face is displayed in the Graphics view. Set this checkbox if you do not want the polygon to be displayed. Hidden faces can be turned on and off in Creator by choosing **View/Hidden Faces**.

Billboard Type

Controls the use of the polygon as a *billboard* (a template for a texture pattern that represents an entire object). Note that applying a transparent material to a textured face effectively turns the face into a billboard without setting this attribute. Billboard settings are:

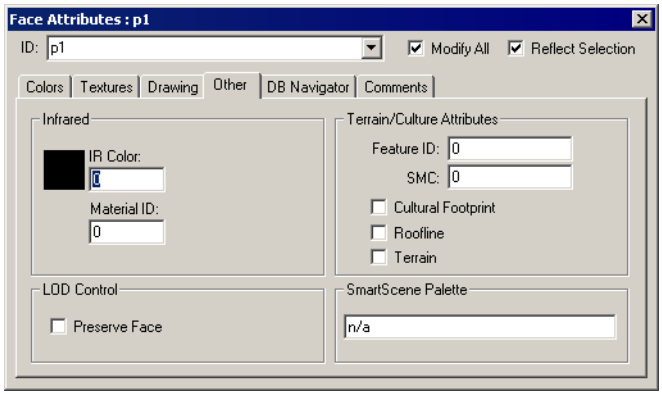
- **Fixed With No Alpha** turns off transparency. If a texture containing transparency is applied to the face, the transparent area of the texture pattern blends with the color of the face.
- **Fixed With Alpha** turns on transparency. If a texture containing transparency is applied to the face, the transparent area of the texture pattern blends with the face behind this face.
- **Axis With Alpha** turns on transparency and makes the polygon into a billboard (see "Billboards" on page 380).
- **Point With Alpha** turns on transparency and causes the polygon to be rotated arbitrarily to face the eyepoint.

Relative Priority

Controls the position of a node in the Hierarchy view relative to its siblings. The hierarchy is periodically sorted to ensure that relative priority increases from left to right in the Hierarchy view (see "Relative Priority" on page 688).

Face Attributes: Other Panel

The Other panel in the *Face Attributes* window contains parameters that are set by special options. These include infrared, LOD, terrain, and SmartScene settings.



IR Color

Contains an intensity code for displaying the face in Infrared mode (see "Infrared Color" on page 415).

Material ID

Defines the material assigned to this face in Infrared mode (see "Infrared Color" on page 415).

LOD Control: Preserve Face

Preserves all the vertices of the face during automatic level-of-detail generation (see "Generating an LOD Automatically" on page 720).

DFAD Feature ID

Contains the Feature Type of the selected face. A face has a **DFAD Feature ID** only if it is part of a cultural feature on terrain (see "Adding Features to Terrain" on page 1127 for more information).

Surface Material Code

Contains the code that describes the composition of the surface of the selected face. A face has a **Surface Material Code** only if it is part of a cultural feature on terrain (see "Adding Features to Terrain" on page 1127 for more information).

Cultural Footprint

Indicates that the face is a polygon that is cut into the terrain (see "Automatic Pre-Projection" on page 1129 for more information).

Roofline

Indicates that the face is part of a roof.

Terrain

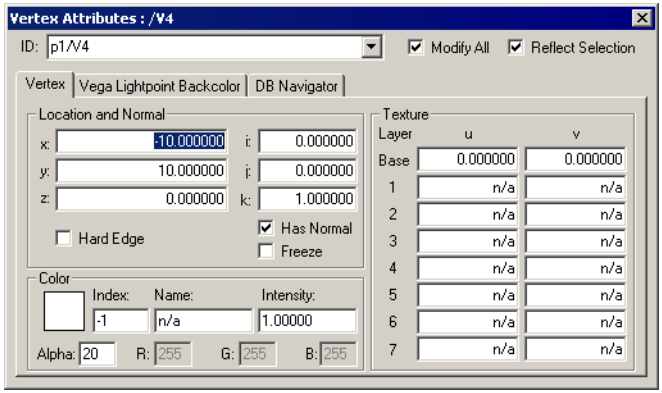
Flags this face as part of the terrain.

SmartScene Palette

Provided for compatibility with the MultiGen Series II SmartScene option (not available in Creator).

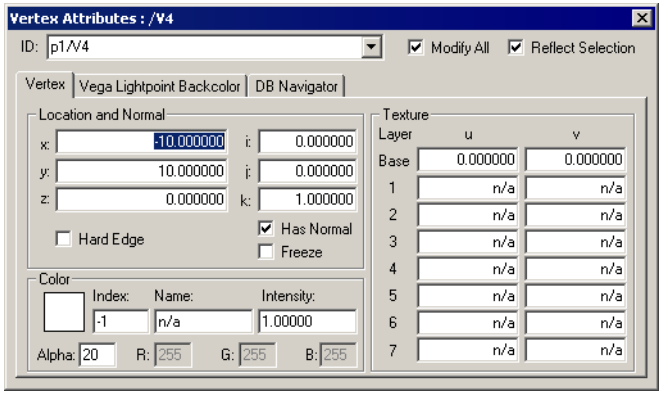
Vertex Attributes

The *Vertex Attributes* window displays the characteristics of the selected vertex. This window has three panels: **Vertex, Vega Lightpoint Backcolor,** and **DB Navigator**.



Vertex Attributes: Vertex Panel

The **Vertex** panel is divided into three groups: **Location and Normal**, **Color**, and **Texture**



The **Vertex** panel has the following attributes:

Location and Normal

Indicates the *x*, *y*, and *z* components of a vertex coordinate.

Contains **i**, **j**, and **k** values that indicate the direction of the vertex normal in Cartesian space. This corresponds to *x*, *y*, and *z* in the coordinate system. A vertex normal is computed from the average of the normals of the faces that share the vertex.

Hard Edge

Specifies that the *face* normal is used for shading, rather than the vertex normal.

Has Normal

Indicates that this vertex contains a vertex normal. Clearing and resetting this checkbox resets the normal to its default value. Shading characteristics of each face are controlled by the face's vertex normals.

Freeze

Prevents the vertex normal from being changed by **Calculate Shading**. The normal is set the first time the database is shaded, regardless of whether this checkbox is selected.

Index

Contains the Color palette index of the color assigned to this vertex (see "Color Palette Index" on page 405).

Name

Contains the symbolic color name of the vertex. If the face's **Index** and **Name** do not reference the same color in the palette, **Name** takes precedence (see "Color Names" on page 402).

Intensity

Contains an intensity multiplier (range 0.0 to 1.0) for the **Index** assigned to the vertex. If this value is less than 1.0, it applies the full intensity value of the color (see "Color Intensity" on page 406).

Alpha

Contains an alpha value that specifies the amount of transparency for the vertex. A smaller value produces a greater amount of transparency. When you calculate shading, the alpha component is interpolated across a face that has either Gouraud or Lit Gouraud shading. The result is a fading effect from an opaque vertex to a transparent vertex.

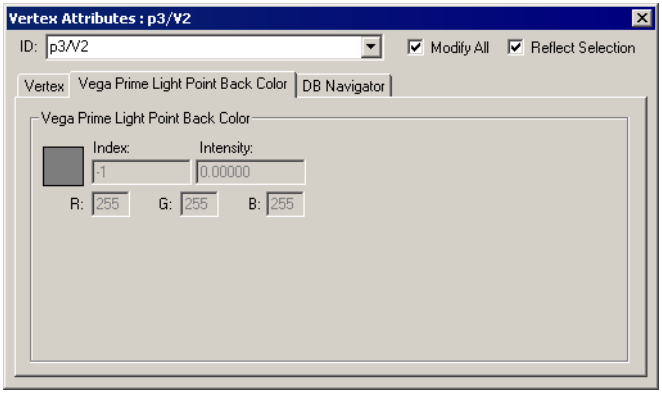
RGB

Contains RGB color values that specify the color of the vertex when RGB drawing mode is enabled in the database header attributes. When the database is in RGB mode, the Color **Index**, **Name** and **Intensity** are ignored (see "RGB Mode" on page 408).

Texture uv

Contains the ***u*** and ***v*** vectors used to map texture layers onto the parent face (see "Texture Tools" on page 444). Layers of textures can be mapped and blended on a face using the MultiTexture feature.

Vertex Attributes: Vega Prime Light Point Back Color Panel



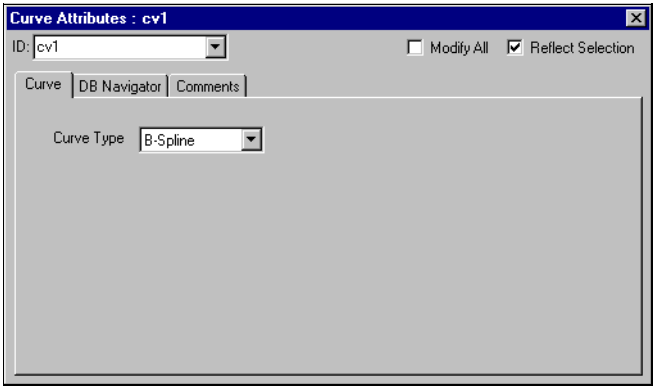
The **Vega Prime Light Point Back Color** panel has the following attributes:

Vega Prime Light Point Back Color

If the vertex is a lightpoint, contains **Index**, **Name**, **Intensity**, and **RGB** values for its back color.

Curve Attributes

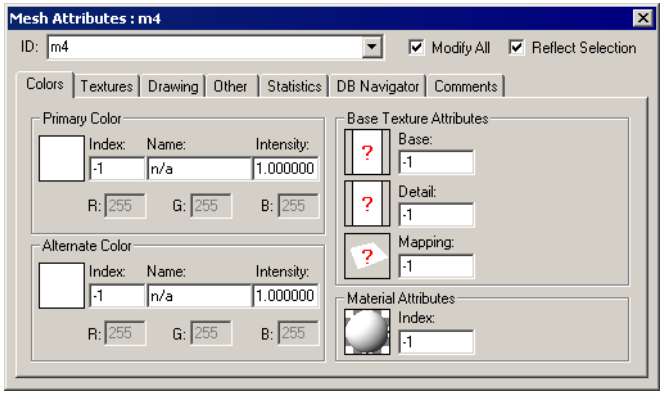
The *Curve Attributes* window displays the attributes of geometry created with the **Curve Tool**. This window has three panels: **Curve Type**, **DB Navigator**, and **Comments**.



The Curve panel contains one attribute setting, **Curve Type**. The **Curve Type** changes the selected curve type to Bezier, B-spline, or Cardinal.

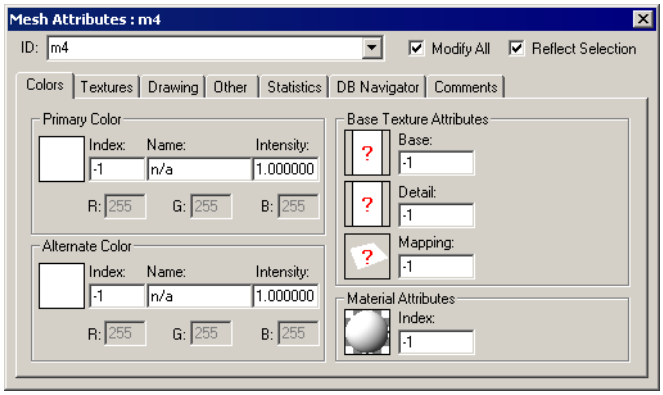
Mesh Attributes

Mesh attributes control the appearance and rendering characteristics of individual polygons (faces) that have been combined into a Mesh node in the database. All geometry in a single mesh shares the same set of attributes.



Mesh Attributes: Colors Panel

The Colors panel is divided into four groups: **Primary Color**, **Alternate Color**, **Base Texture Attributes**, and **Material Attributes**.



Primary Color: Index

Contains the Color palette index of the color assigned to this face (see "Color Palette Index" on page 405). A value of -1 indicates that no primary color has been assigned, in which case the primary color defaults to white

Primary Color: Name

Contains the symbolic color name of the face. If the face's **Index** and **Name** do not reference the same color in the palette, **Name** takes precedence (see "Color Names" on page 402).

Primary Color: Intensity

Contains an intensity multiplier (range 0.0 to 1.0) for the **Index** assigned to the face. If this value is less than 1.0, it applies the full intensity value of the color (see "Color Intensity" on page 406).

Primary Color: RGB

Contains RGB color values that specify the color of the face when RGB drawing mode is enabled in the database header attributes (see "RGB Mode" on page 408). When the database is in RGB mode, the **Index**, **Name** and **Intensity** are ignored (see "RGB Mode" on page 408).

Alternate Color: Index

Contains the Color palette index for the face's alternate color (see "Color Palette Index" on page 405). A value of -1 indicates that no alternate color has been assigned, in which case the alternate color defaults to white.

Alternate Color: Name

Contains the symbolic color name of the face. If the face's **Index** and **Name** do not reference the same color in the palette, **Name** takes precedence (see "Color Names" on page 402).

Alternate Color: Intensity

Contains an intensity multiplier (range 0.0 to 1.0) for the **Index** assigned to the face. If this value is less than 1.0, it applies the full intensity value of the color (see "Color Names" on page 402).

Alternate Color: RGB

Contains RGB color values that specify the color of the face when RGB drawing mode is enabled in the database header attributes. When the database is in RGB mode, the **Index**, **Name** and **Intensity** are ignored (see "RGB Mode" on page 408).

Texture Attributes: Base

Contains the Texture palette index of the texture applied to this face (see "Texture Palettes" on page 422). If no texture has been applied, this field contains a value of -1.

Texture Attributes: Detail

Contains the Texture palette index of the detail texture applied to this face (see "Detail Texture" on page 530). If a detail texture has not been applied, this field contains a value of -1.

Texture Attributes: Mapping

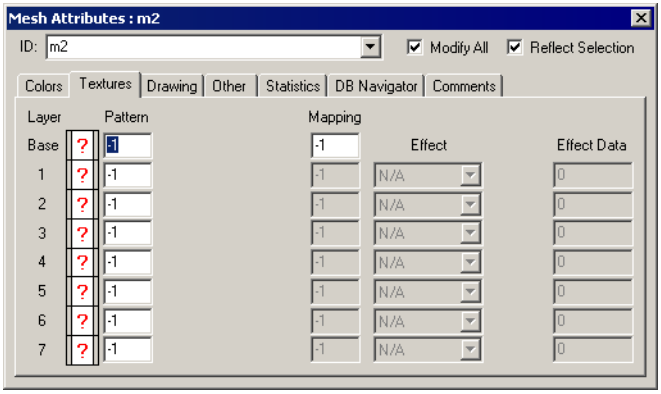
Contains the index value of the selected texture mapping in the texture mapping palette (see "The Texture Mapping Palette" on page 516). If no texture has been applied, or if an applied texture has not been added to the Texture Mapping palette, this field contains a value of -1.

Material Attributes: Index

Contains the index value of the material assigned to the face (see "The Material Palette" on page 387). If no material has been applied, this field contains a value of -1.

Mesh Attributes: Textures Panel

The Textures panel in the *Mesh Attributes* window contains the parameters for the MultiTexture feature that control which texture layers are applied to a face and how each texture layer blends with other texture layers. For more information about the MultiTexture feature, see "MultiTexture" on page 470.



Layer

Indicates the layer number of the texture. Layer 0 from the Texture list in the View panel is the **Base** layer in this panel.

Pattern

Indicates the Texture palette index number. The index number for each texture is also located in the *Texture Palette* window.

Mapping

Indicates the Texture mapping index number. A **-1** appears if the texture does not have a texture map in the Texture Mapping palette.

Effect

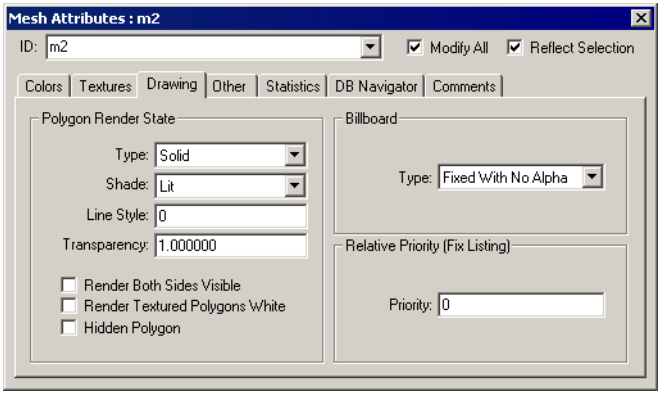
Sets the rendering for multitexture effects to **texture env**, **bump map**, or **custom** for the texture layer. Creator renders multitexture effects using **texture env**, which blends the texture layer with the underlying layers using the texture environment algorithm that you set in the *Texture Attributes* window. You can select **bump map** if your runtime system creates bump maps, or **custom** for an effect that you have defined. If you select **custom**, you can also enter a number that your runtime system uses to render the effect.

Effect Data

Contains a number for your runtime system to render multitexture effects. This text field is optional.

Mesh Attributes: Drawing Panel

The Drawing panel contains the parameters that control how the face is drawn in Creator and rendered in the realtime system.



Type

Sets the drawing mode for this face to **Solid**, **Closed Wireframe**, or **Wireframe**. This setting overrides the drawing mode set with the **Draw Mode** control in the View panel.

Shade

Sets the shading for this face to **Flat**, **Lit**, **Gouraud**, or **Lit Gouraud**.

Line Style

Contains the index number in the database Line Style palette for the line style of the face's edges. This feature is available only with the CreatorPro Instrumentation option. See "Line Styles" on page 821 for more information.

Transparency

Indicates the transparency of the face, in the range of 0.0 (clear) to 1.0 (opaque), on platforms that support alpha blending. The net transparency applied to a face, as drawn by Creator, is calculated by multiplying the **Transparency** field by the transparency component of the material assigned to the face (if any). For example, if face transparency is 0.5, and the alpha value of a material assigned to the face is 0.5, the net transparency applied is 0.25 (more transparent). Changes to the face transparency automatically propagate to subfaces.

Render Both Sides Visible

Turns on the display of color and texture on both the frontfacing and backfacing sides of the polygon. (By default, only frontfaces are visible.)

Render Textured Polygons White

Overrides the assigned face color and makes the face white if it is textured. Creator sets this checkbox automatically when the **Draw New Textured Polygons White** header attribute is set (see "Draw New Textured Polygons White" on page 833). Setting this checkbox prevents face color from blending with the colors of applied textures.

Hidden Polygon

Controls whether this face is displayed in the Graphics view. Set this checkbox if you do not want the polygon to be displayed. Hidden faces can be turned on and off in Creator by choosing **View/Hidden Faces**.

Billboard Type

Controls the use of the polygon as a *billboard* (a template for a texture pattern that represents an entire object). Note that applying a transparent material to a textured face effectively turns the face into a billboard without setting this attribute. Billboard settings are:

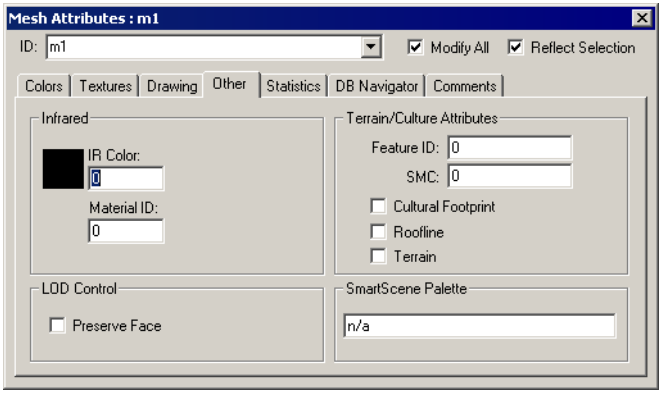
- **Fixed With No Alpha** turns off transparency. If a texture containing transparency is applied to the face, the transparent area of the texture pattern blends with the color of the face.
- **Fixed With Alpha** turns on transparency. If a texture containing transparency is applied to the face, the transparent area of the texture pattern blends with the face behind this face.
- **Axis With Alpha** turns on transparency and makes the polygon into a billboard (see "Billboards" on page 380).
- **Point With Alpha** turns on transparency and causes the polygon to be rotated arbitrarily to face the eyepoint.

Relative Priority

Controls the position of a node in the Hierarchy view relative to its siblings. The hierarchy is periodically sorted to ensure that relative priority increases from left to right in the Hierarchy view (see "Relative Priority" on page 688).

Mesh Attributes: Other Panel

The Other panel contains parameters that are set by special options. These include infrared, LOD, terrain, and SmartScene settings.



IR Color

Contains an intensity code for displaying the face in Infrared mode (see "Infrared Color" on page 415).

Material ID

Defines the material assigned to this face in Infrared mode (see "Infrared Color" on page 415).

LOD Control: Preserve Face

Preserves all the vertices of the face during automatic level-of-detail generation (see "Generating an LOD Automatically" on page 720).

DFAD Feature ID

Contains the Feature Type of the selected face. A face has a **DFAD Feature ID** only if it is part of a cultural feature on terrain (see "Adding Features to Terrain" on page 1127 for more information).

Surface Material Code

Contains the code that describes the composition of the surface of the selected face. A face has a **Surface Material Code** only if it is part of a cultural feature on terrain (see "Adding Features to Terrain" on page 1127 for more information).

Cultural Footprint

Indicates that the face is a polygon that is cut into the terrain (see "Automatic Pre-Projection" on page 1129 for more information).

Roofline

Indicates that the mesh is part of a roof.

Terrain

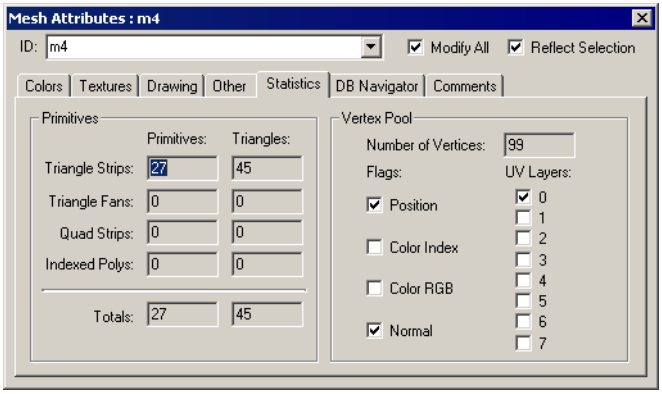
Flags this face as part of the terrain.

SmartScene Palette

Provided for compatibility with the MultiGen Series II SmartScene option (not available in Creator).

Mesh Attributes: Statistics Panel

The Statistics panel contains information about the primitives, triangles, and vertices in the mesh. All of the information in this panel is read-only.



Triangle Strips

The number of primitives and triangles in triangle strips within the mesh.

Triangle Fans

The number of primitives and triangles in triangle fans within the mesh.

Quad Strips

The number of primitives and triangles in quad strips within the mesh.

Indexed Polys

The number of primitives and triangles in indexed polygons within the mesh.

Number of Vertices

The total number of vertices in the mesh. The vertices within the vertex pool have the same attributes and are stored independently.

Flags

The attributes of the vertices. If **Normals** is set, for example, all of the vertices have normals.

UV Layers

The texture layers in which the vertices have *u*,*v* coordinates.

LOD Attributes

Levels of Detail (LODs) are sets of models that represent the same object or terrain area with varying degrees of complexity. LOD attributes control the behavior of LODs. The *LOD Attributes* window has three panels: **LOD**, **DB Navigator**, and **Comments**. The following topics describe these panels and provide additional information about LODs.

Switch Nodes and Switch Attributes

A *switch node* contains masks that control the display of its children.

Switch masks are attributes of each switch node. When a switch node is created, it holds a single mask (Mask 0) containing one vertical bar for each child of the switch node, with the left-most bar representing the left-most child in the hierarchy.

To enable the correct display of switch nodes when you open a database, turn on the **Enable Switch Nodes** Flight preference (see "Flight Preferences" on page 60). If this preference is cleared, all the children of the switch node are displayed until the first time you edit its masks.

To Create a Switch Node

Step 1. Set a Group-like node as the database parent.



Step 2. Open the **Create** toolbox.



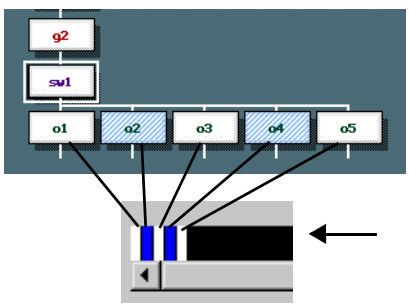
Step 3. Click **Create Switch**.

Step 4. Attach the groups or objects to the Switch node.

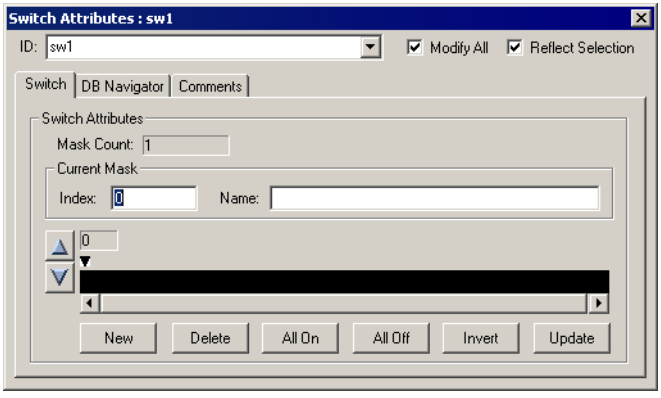
Step 5. Double-click the Switch node to open the *Switch Attributes* window (see "Switch Attributes" on page 862), and click **New** to create a mask for editing.

Step 6. Click the bars in the mask to control the display of the corresponding children. A white bar turns on the display of the corresponding child; a blue bar turns off the display.

If the Switch node has a large number of children, you may prefer to turn the child nodes on or off by using **Toggle Display**, and then click **Update** to assign the display settings to the mask.

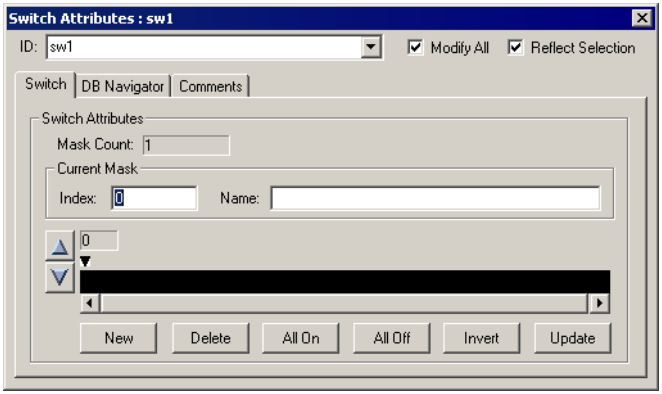


The mask turns the children of the switch node on or off



Switch Attributes

A *switch node* contains masks that control the display of its children. These switch masks are attributes of each switch node.



The *Switch Attributes* window has three panels: **Switch**, **DB Navigator**, and **Comments**.

Switch Panel Attributes

Current Mask Index

Shows the index of the mask that is selected for editing. The selected mask controls the display of the children of the switch node in the Graphics view.

Name

Each mask index can be assigned a name (to 255 characters) that becomes available to the runtime.

Mask Count

Shows the total number of masks that have been created for this Switch node.

Arrow Buttons



Select the previous or next mask defined for the Switch node.

New

Adds a new mask to the Switch node.

Delete

Deletes the selected mask. Masks with higher indices automatically move down to fill the empty index position.

All On

Turns on the display of all the children of the Switch node.

All Off

Turns off the display of all the children of the Switch node.

Invert

Toggles the bars in the mask. Each child that was previously off is turned on; each child that was on is turned off.

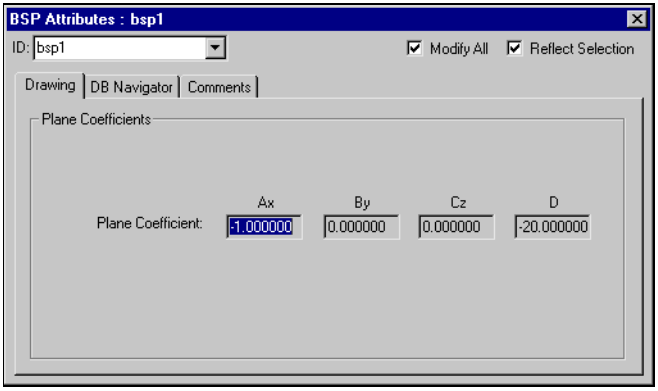
Update

Updates the mask to match the display status of the children of the Switch node. This lets you set a mask by manually turning on and off the display of the Switch node's children using **Toggle Display**.

BSP Attributes

Binary Separating Planes (BSPs) are virtual planes that help determine the drawing order of elements in the hierarchy.

The *BSP Attributes* window has three panels: **Drawing**, **DB Navigator**, and **Comments**.



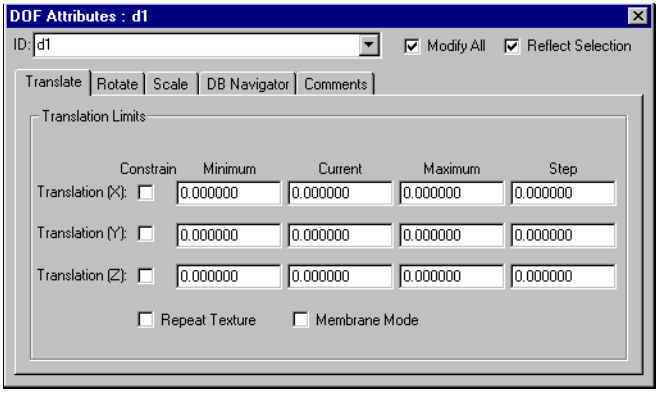
BSP Attributes: Drawing Panel

The Drawing Panel has one attribute: the Plane Coefficient, which displays the coefficients of the plane attached to the BSP node.

DOF Attributes

Degrees of Freedom (DOFs) are nodes whose attributes define a range of translational and/or rotational motion for the geometry attached to the DOF node.

The *DOF Attributes* window has five panels: **Translate**, **Rotate**, **Scale**, **DB Navigator**, and **Comments**.



DOF Attributes: Translate Panel

The **Translate** panel in the *DOF Attributes* window has the following parameters:

Constrain

Controls the range of motion for the DOF relative to a coordinate axis (x, y, or z).

Current

Specifies the current position of the DOF within the range defined by **Minimum** and **Maximum**. If you specify a value outside the range defined by **Minimum** and **Maximum**, those values are automatically extended to include the new **Current** value.

Minimum

Defines the lower limit on the range of the translation.

Maximum

Defines the upper limit on the range of the translation.

Step

Specifies the amount to add or subtract to the current position of the DOF when it is animated.

Repeat Texture

This attribute supports the SmartScene Option, which is not available in Creator. **Repeat Texture** is included for backward compatibility with OpenFlight files created in MultiGen II.

Membrane Mode

This attribute supports the SmartScene Option, which is not available in Creator. **Membrane Mode** is included for backward compatibility with OpenFlight files created in MultiGen II.

DOF Attributes: Rotate Panel

The **Rotate** panel in the *DOF Attributes* window has the following parameters:

Constrain

Controls the range of rotation for the DOF around a coordinate axis (x, y, or z).

Current

Specifies the current position of the DOF within the range defined by **Minimum** and **Maximum**. If you specify a value outside the range defined by **Minimum** and **Maximum**, those values are automatically extended to include the new **Current** value.

Minimum

Defines the lower limit on the range of the rotation.

Maximum

Defines the upper limit on the range of the rotation.

Step

Specifies the amount to add or subtract to the current position of the DOF when it is animated.

DOF Attributes: Scale Panel

The **Scale** panel in the *DOF Attributes* window has the following attributes:

Constrain

Controls the range of scale for the node relative to a coordinate axis (x, y, or z).

Current

Specifies the current position of the DOF within the range defined by **Minimum** and **Maximum**. If you specify a value outside the range defined by **Minimum** and **Maximum**, those values are automatically extended to include the new **Current** value.

Minimum

Defines the lower limit on the range of the scale.

Maximum

Defines the upper limit on the range of the scale.

Step

Specifies the amount to add or subtract to the **Current** position of the DOF when it is animated.

Sound Attributes

Sound files can be assigned to sound nodes in the database, for activation in the realtime system. Sound attributes control the physical properties of a sound.

The *Sound Attributes* window has three panels: **Sound**, **DB Navigator**, and **Comments**.

Light Source Attributes

Light source nodes allow you to vary the position or direction of light sources and restrict their effect to specific portions of the hierarchy. Unlike modeling lights, which are used only for preview purposes, light source nodes are exported to the realtime system as part of the database.

Generic Attributes

Some applications maintain node types that are not normally supported by OpenFlight. When Creator reads a database containing unrecognized nodes, a generic node is created for each unrecognized node type. The figure below shows the display of a generic node (type 254) in the hierarchy view of a database.



Attribute Searching

Attribute searches search your database for specific node attributes. By setting values for specific nodes in your database, you can customize your database so that any node or combination of nodes can easily be selected.

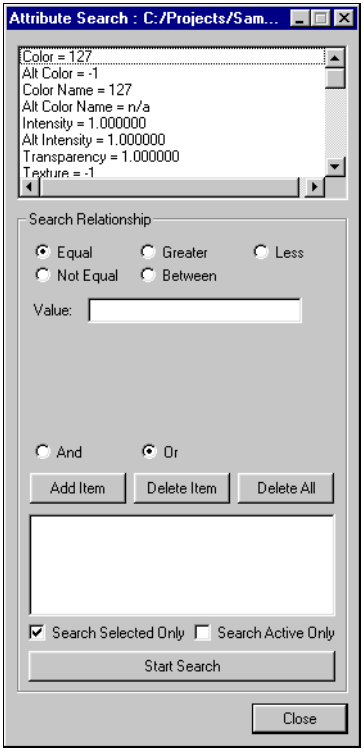
If you have the Creator Pro option, you can filter the feature data you import into a terrain database by performing a Feature Attribute Search. You can also search for specific cultural features based on their attributes. See the *Online Help Summary; MultiGen Creator* for instructions.

Performing an Attribute Search

The attribute search function selects items in the database by attribute value. This is especially useful in conjunction with **Modify All** for selecting and editing the attributes of more than one item at a time. (See “Modify All” on page 827.)

To Start an Attribute Search

Step 1. Choose **Attribute/New Attribute Search** to open the *Attribute Search* dialog box.



The top part of the *Attribute Search* dialog box displays a list of the node attributes defined for the current modeling mode. If anything is selected in the database when you start the search, the attribute values of the first item in the **Select List** are also displayed in the top part of the *Attribute Search* dialog box.

In the *Attribute Search* dialog box, the **Search Relationship** option buttons control the search conditions. The number and type of these controls change depending on which node attribute is selected in the top part of the *Attribute Search* dialog box.

The lower part of the *Attribute Search* dialog box displays a list of previously defined search conditions and additional controls for performing and restricting a search.

Step 2. Select an attribute to search for from the list at the top of the dialog box.

Step 3. Select a **Search Relationship** (**Equal**, **Greater**, **Less**, **Not Equal**, or **Between**) for the attribute value. Only relationships that have meaning for the selected attribute are displayed in this part of the dialog box.

Step 4. Enter the value (or range of values) of the selected attribute. If you choose **Between**, **Value** will change to **Low Value** and **High Value**, so that you can enter a range of values.

Step 5. If the attribute accepts one of a list of values (for example, **Bounding Volume Type** can be Sphere, Cylinder, Convex Hull, or Histogram), select the value from the list in the **Value** field. If the attribute requires a numeric value, enter the value in the **Value** field.

Step 6. Click **Add Item**. The new search condition appears in the list at the bottom of the *Attribute Search* dialog box.

Step 7. Add or delete search conditions until you are satisfied with the search.

To add a search condition, click **And** or **Or** (this defines the relationship between previous search conditions and the next search condition), and repeat the previous steps.

To delete a search condition, select it in the list at the bottom of the *Attribute Search* dialog box and click **Delete Item**. To delete all search conditions, click **Delete All**.

The following two checkboxes determine how the search proceeds:

Search selected only restricts the attribute search to items that are selected in the database when you click **Start Search**. If this checkbox is cleared, items are searched whether or not they are selected.

Search active only restricts the attribute search to items that are being displayed in the Graphics view (see "Hierarchy Tools" on page 229). If this checkbox is cleared, items are searched whether or not they are displayed.

Step 8. Click **Start Search** to perform the attribute search. The search results are shown in the Select panel. If you want a hard copy of the search results, see "Attribute Reports" on page 872.

Reopening an Attribute Search

Choosing **Attribute/Attribute Search** reopens the *Attribute Search* dialog box containing the search conditions from the most recent attribute search.

Attribute Reports

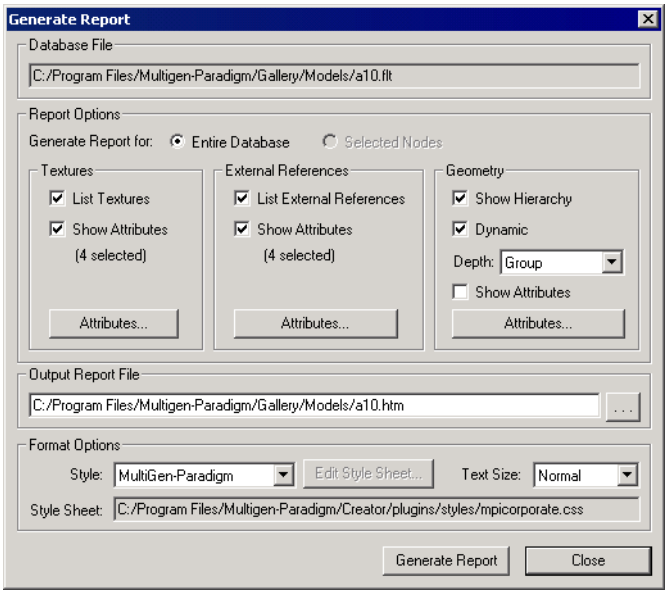
Attributes/Generate Report produces a report about your database, or selected nodes within your scene. The data is displayed in an HTML file which includes:

- Database Hierarchy – the database structure shown in a tree-like structure. You specify how deep within the hierachy to display information (group, object, face, etc.).
- Node Attributes – the selected attribute values for nodes. You specify which nodes are included and which attributes for each included node are displayed.
- Texture File Information – information on individual textures contained in the palette. You select which attributes of each texture are displayed.
- External References File Information – information on external references referenced by your database. You select which attributes of the external references are displayed.

To Generate a Report

Before launching the report generator, you should decide whether you want a report about specific nodes only, or for your entire database. If you want a report about specific nodes only, select those nodes in your database (in either the Graphics or the Hierarchy window), before launching the report generator. If you do not select any specific nodes, you receive a report on the entire database.

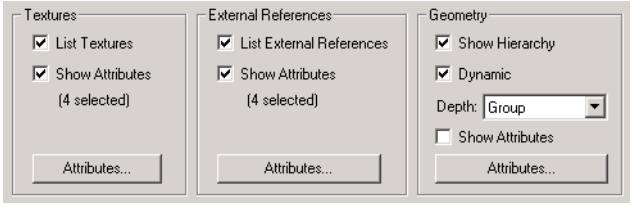
Step 1. Choose **Attributes/Generate Report** from the Menu bar. The *Generate Report* panel appears.



Step 2. Choose the scope of your report using the radio buttons: **Entire Database** or **Selected Nodes**. The latter option is available only if you selected particular nodes before you opened the *Generate Report* window.

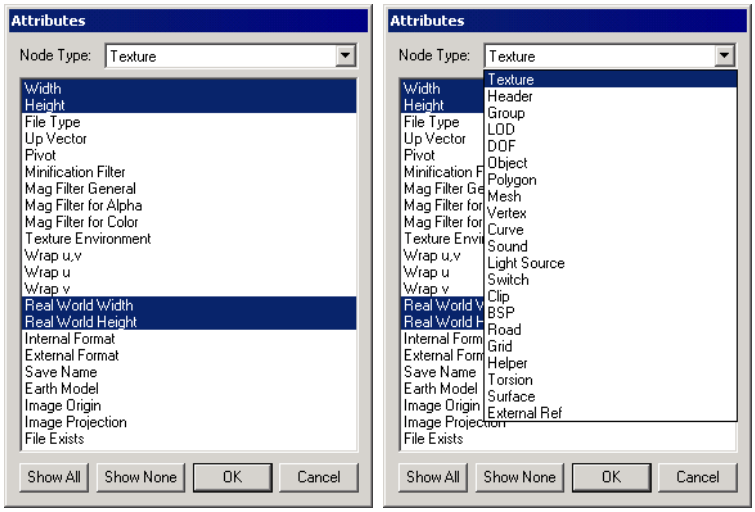


Step 3. Choose the items and attributes you wish to include in your report. The top checkbox in each column functions as an "on-off" switch for listing the items in that column (textures, external references, and geometry); the other checkboxes furnish additional data about the attributes of those items.



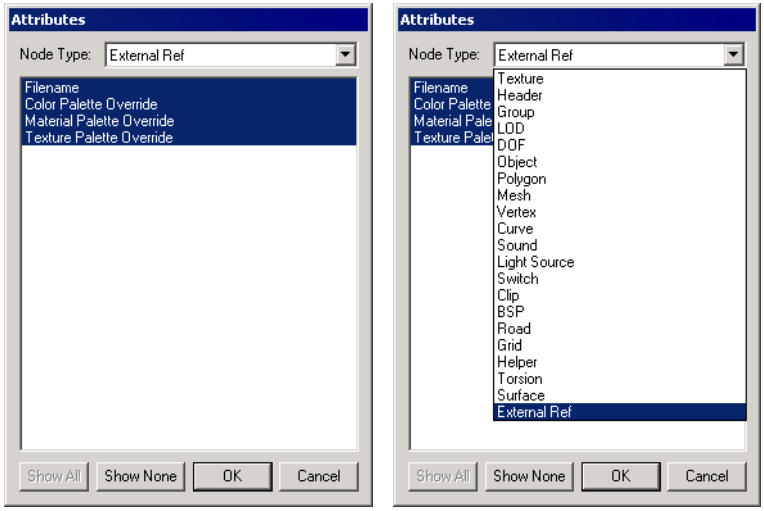
Your specific options include:

Textures – If you click the **Attributes** button within the **Textures** column, an **Attributes** window appears, allowing you to select which texture attributes are included in your report. By selecting different options from the **Node Type** list, you are able to choose specific attributes for each **Node Type**.

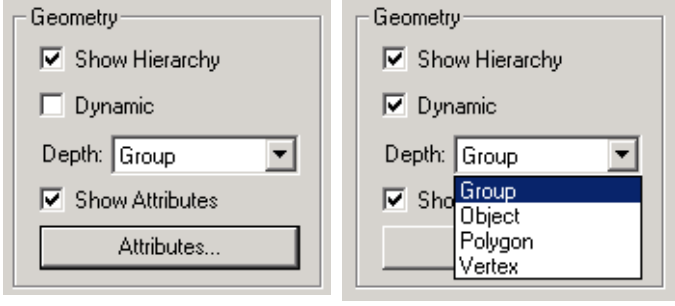


External References – If you click the **Attributes** button within the **External References** column, an **Attributes** window appears, allowing you to select which texture attributes are included in your report. By selecting

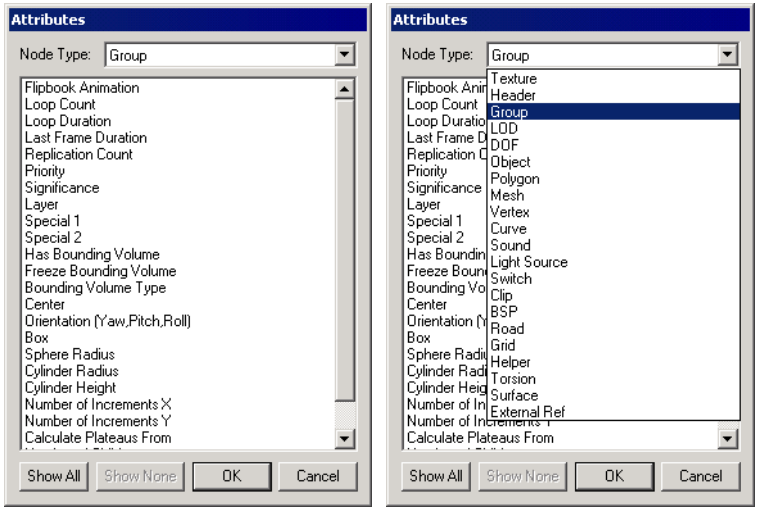
different options from the **Node Type** list, you are able to choose specific attributes for each **Node Type**.



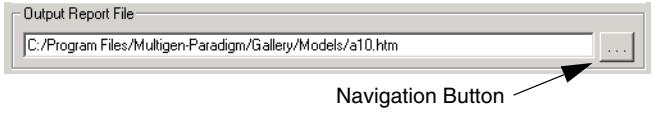
Geometry – If you choose **Show Hierarchy**, your report displays the geometry in a tree-like diagram. Selecting the **Dynamic** option produces an HTML file with a collapsing and expanding hierarchy. Using the **Depth** list, you choose the level of the geometry you want included in your report. You may choose from **Group**, **Object**, **Polygon**, and **Vertex**..



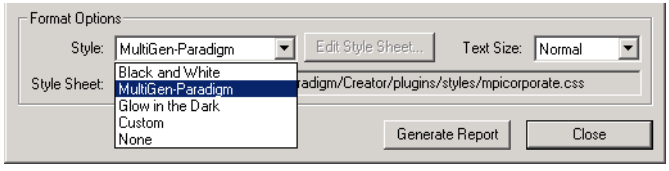
As with the previous columns, if you click on the **Attributes** button within the **Geometry** column, an **Attributes** window appears, allowing you to select which attributes are included in your report.



Step 4. If you want to specify a particular location for your report, click the navigation button on the right side of the the **Output Report File** section; a navigation window opens, and you select a destination. By default, your report is written into the directory with your database; if you wish for your report to remain adjacent to your database, skip this step.



Step 5. Choose the **Format Options** you prefer for your report. The report is written as an HTML file, so these controls allow you to modify the appearance of the information which is diplayed in your browser.



Style – There are three pre-defined styles: **Black and White** (the best choice if you plan to print your report), **MultiGen-Paradigm** (white background, with blue and gray shading in tables), and **Glow in the Dark** (black background with light lettering). If you choose **None**, the appearance of the report is governed by your default browser settings.

The different styles are produced by using Cascading Style Sheets (CSS) for the HTML output. If you prefer to construct your own CSS for reports, choose **Custom** from the list. The **Edit Style Sheet** button becomes active, and clicking the button opens the `custom.css` file in Notepad, where you can customize it.

Note: The CSS formatting is actually embedded within the HTML files produced by the report generator (internal CSS, in contrast to the external Cascading Style Sheets commonly used on web sites). Therefore, if you edit a `.css` file, your changes are not applied unless you regenerate the report.

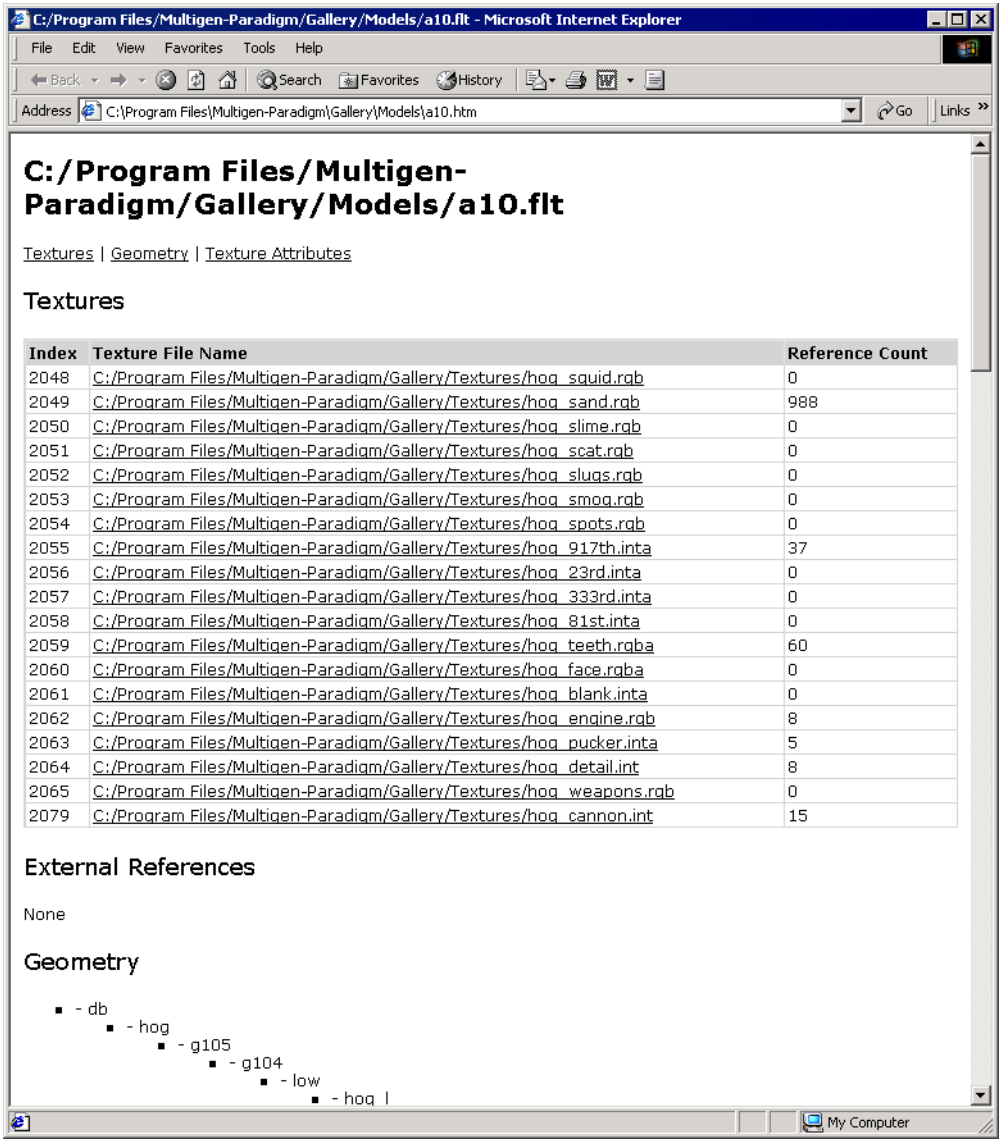
Note: If you are an IRIX user, and you wish to customize the Cascading Style Sheet, you need to open the the `.css` file in a text editor yourself. (Your text editor is not launched automatically.)

Text Size – In ordinary cases, keep the default setting of **Normal** from this list. You may adjust the the size of the text by choosing the **Smaller** or **Bigger** options.

Step 6. Click **Generate Report** when you are ready to view your report. The report is displayed automatically using your default browser (usually Internet Explorer).

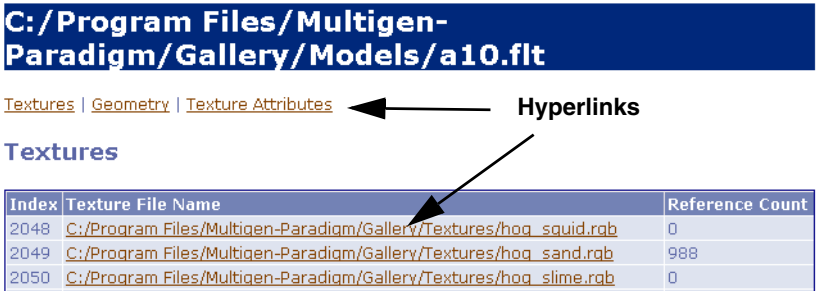
Note: If you are an IRIX user, you need to open the report file in your browser yourself; your browser is not launched automatically. Unless you have changed the destination for your report, it will be

found in the same directory as your model (with the same name, and an `.htm` extension).



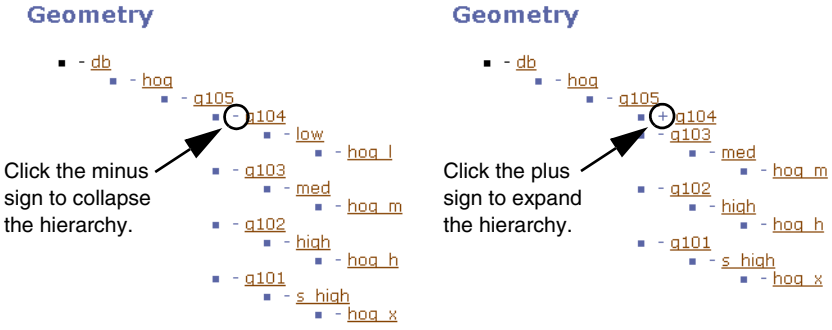
Report Example (Black and White Style)

Hyperlinks within the report help you navigate to detailed sections of the document.



Hyperlinks (MultiGen-Paradigm Style)

If you chose the **Dynamic** option in the **Geometry** section of the report generator, take a close look at the Geometry section in the report displayed in your browser window.



The tree is dynamically interactive, allowing you to collapse or expand various sections of the hierarchy. To collapse a section, position your cursor directly over the minus sign (-) next to an item and click it; to expand a section, position your cursor over the plus sign next to an item and click it.

Other sections of the report contain details of your database according to the options you selected in the report generator.

Attribute Resolver

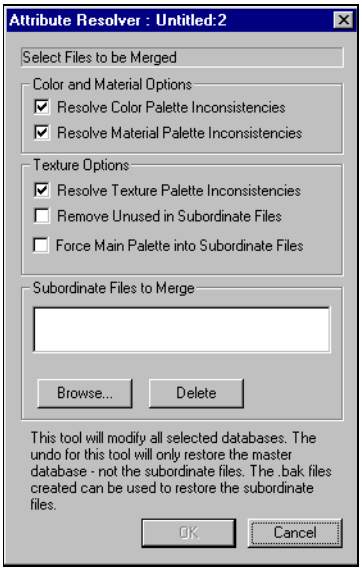
Every OpenFlight database has its own color, texture, and material palettes. Having different palettes may cause inconsistencies when you cut and paste from one database to another, or when you reference an external database without setting the palette **Override** checkboxes (see "External References" on page 370). The **Attribute Resolver** in the **Attributes** menu merges the palettes of subordinate databases into a selected master database, allowing all databases to be used together without inconsistencies.

To Resolve Palette Inconsistencies

Step 1. Open a database. This is the *master database* and will contain the merged contents of the palettes of the subordinate databases (selected below).

Note: Because **Attribute Resolver** makes a single pass through each database, the palettes of each selected database will be compatible with, but may differ from, the master database palettes. If you want each subordinate database to have the *same* set of palettes as the master database, run **Attribute Resolver** a second time.

Step 2. Choose **Attributes/Attribute Resolver** to open the *Attribute Resolver* window.



Step 3. Select the items to be resolved from the following:

Resolve Color Palette Inconsistencies replaces the color palette of the subordinate files with the color palette of the master database and changes the color of faces in the selected databases to the closest match in the new palette.

Resolve Material Palette Inconsistencies merges materials in each subordinate database into the master database and updates material assignments to match the new palette.

Resolve Texture Palette Inconsistencies merges the texture patterns in the subordinate files into the master database and updates texture assignments to match the new palette.

Remove Unused in Subordinate Files deletes textures that have not been applied to faces in each subordinate database before merging.

Force Main Palette into Subordinate Files assigns the texture palette of the master database to each subordinate database immediately after merging.

Step 4. Use the **Browse** button to select databases to merge into the master database. Each file you select is added to the **Subordinate Files to Merge** list at the bottom of the window.

Step 5. Click **OK** at the bottom of the *Attribute Resolver* window to merge the selected palettes in the subordinate files into the master database.

Exercise: Using Attribute Windows

This procedure demonstrates how to manage the attributes of groups and objects and how to navigate through the database from within an *Attributes* window. Before you begin, you may want to print this topic to make it easier to follow the instructions as you work with Creator.

Build a Sphere

In this section, you prepare for working with attributes by building a sphere.

Step 1. Choose **File/New** (Ctrl+N) to create a new file.



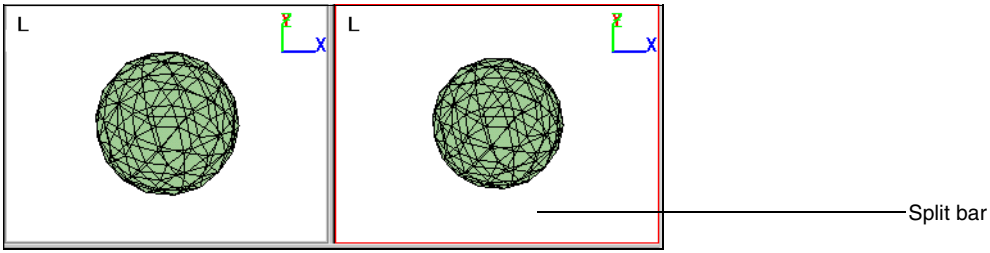
Step 2. Open the **Geometry** toolbox.



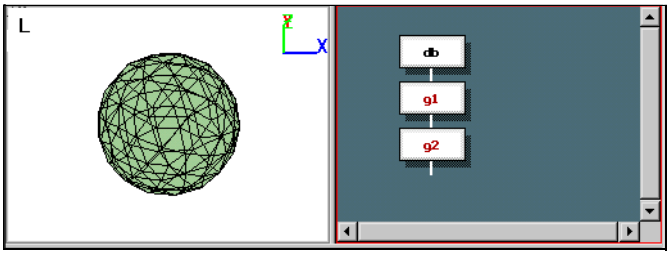
Step 3. Click **Sphere**.

Step 4. Create a sphere in the Graphics view.

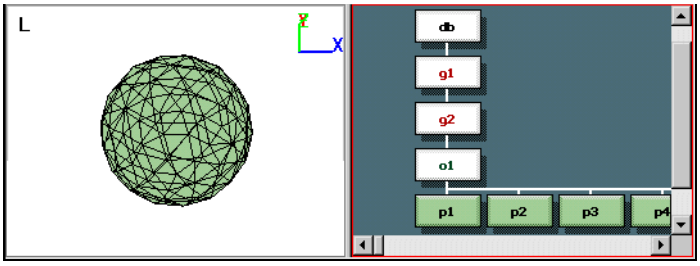
Step 5. Point to the split bar at the right side of the window and drag it left to split the Database window into side-by-side Graphics views.



Step 6. Click in the right Graphics view to make it active (the active view is outlined in red), and then press the keyboard shortcut S to change to a Hierarchy view of the sphere.



Step 7. Middle-click twice in the Hierarchy view to display the object and face nodes belonging to the sphere. You may need to drag down the right scroll bar to see nodes that are lower in the hierarchy.

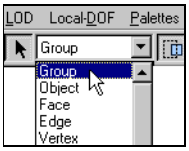


Step 8. Choose **File/Save** (Ctrl+S) to save your database.

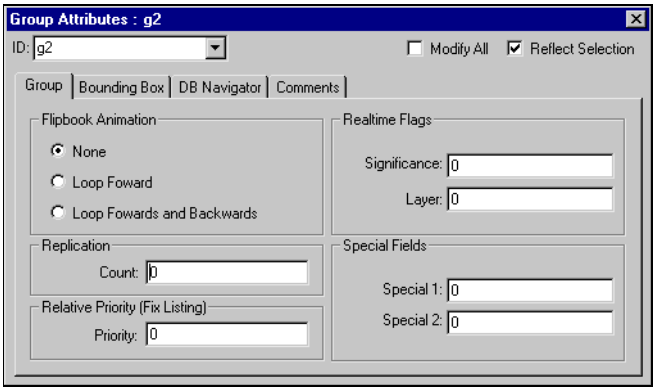
Open an Attributes Window

Double-clicking an item in the Graphics view or Hierarchy view selects the item and opens its Attributes window.

Step 1. Choose **Group** modeling mode in the Toolbar. In the Graphics view, the modeling mode determines the type of node you can select.

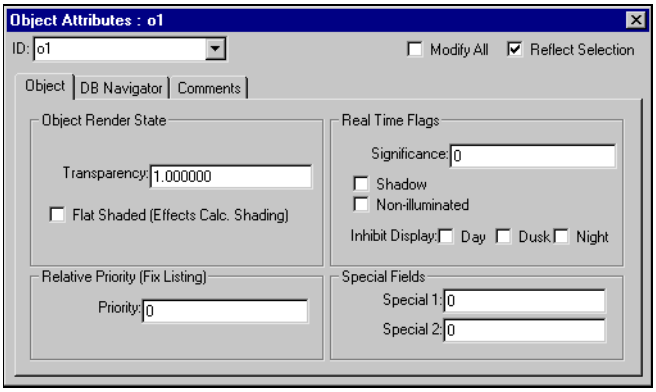


Step 2. In the Graphics view, double-click the sphere to open its *Group Attributes* window.



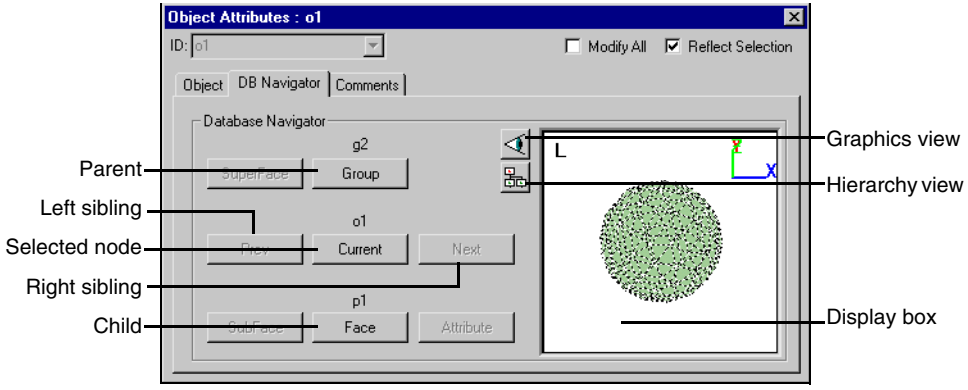
Step 3. Make sure the **Reflect Selection** checkbox is set. This tells the software to replace the current *Attributes* window each time a new window is opened. (If this checkbox is cleared, *Attributes* windows layer on top of one another.)

Step 4. Double-click the object node in the Hierarchy view to select it and open its attributes. In the Hierarchy view, you can select nodes and open their attributes without regard to the modeling mode.

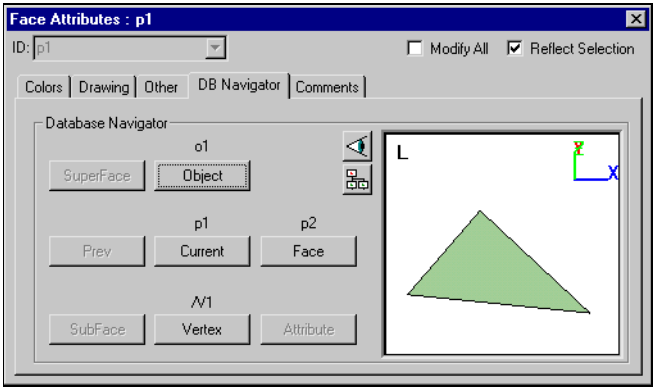


Navigate Through the Database

Step 1. In the *Object Attributes* window, click the **DB Navigator** tab. The DB Navigator panel appears in every *Attributes* window. It contains buttons for viewing the geometry and hierarchy attached to the selected node, and for selecting the parent, siblings, and children of this node.



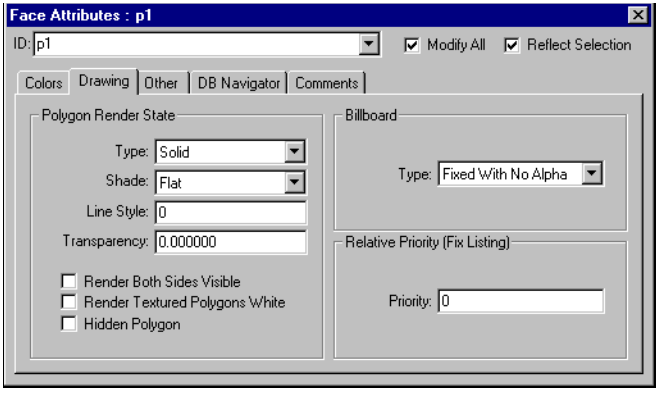
Step 2. Click the **Face** button to open the attributes of the first (left-most) child of the object.



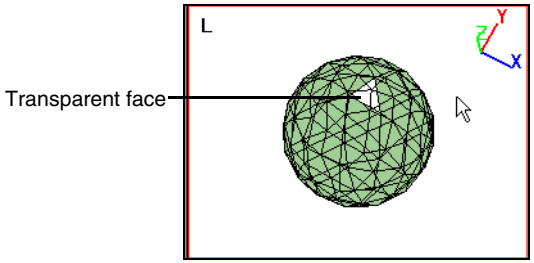
Edit Face Attributes

Step 1. Click the **Drawing** tab in the *Face Attributes* window. The Drawing panel contains attributes that control how the face is displayed.

Step 2. In the **Transparency** field, type 0 (zero). This makes the face invisible.



Step 3. Drag with the right mouse in the Graphics view to rotate the sphere. Observe that one of its faces is now invisible.



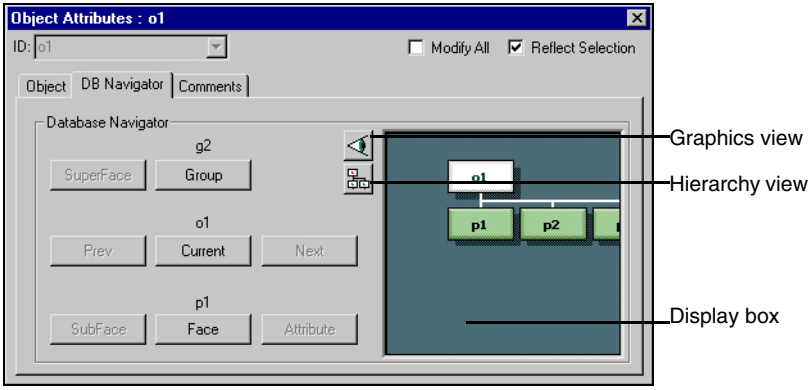
Navigate Using the Display Box

Step 1. Click the **DB Navigator** tab.

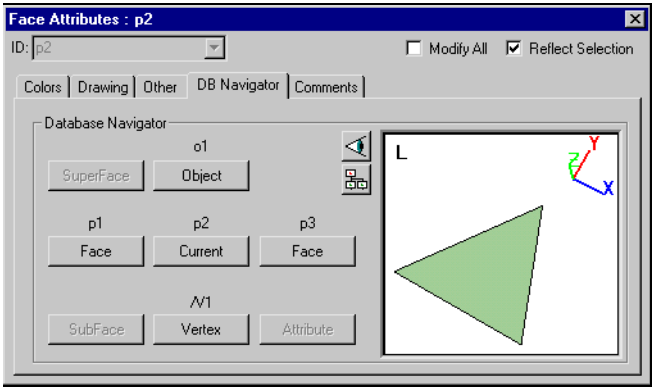
Step 2. Click the **Object** button to open the attributes of the parent of the transparent face. (The parent is the sphere object.)

Step 3. Click the **Hierarchy** button in the DB Navigator panel to view the hierarchy of the object node.





Step 4. In the **Display** box, click the second face from the left. Observe that this opens the corresponding *Face Attributes* window. As you see, you can navigate through the database hierarchy using the **Display** box.



Step 5. Click the **Close** button to close the *Face Attributes* window.

Step 6. Close the Database window and exit Creator.

24 Preference and Default Files

Creator stores application settings, terrain and feature generation settings, and default palettes in certain files. The following topics describe these files, where the files are located, and where Creator looks for these files when it needs to load them.

Preference and Shortcut Files

The following files contain application preferences and keyboard shortcut settings.

creator1.prefs

Stores the settings made in the Creator Preferences panels, the last entered information in the tool dialog boxes, and the index numbers of the current palette selections. When Creator launches it searches for creator1.prefs first in the user's home directory, and then in the Creator installation directory. If the file cannot be found, a new creator1.prefs file is created in the Creator installation directory using source definitions.

creator.ksc

Contains the map of default keyboard shortcuts. It is stored in the same directory as the Creator application. If Creator cannot find creator.ksc in this location, it loads a default keyboard map from source definitions.

custom.ksc

Contains the mapped keyboard shortcuts when you customize them using **Info/ Keyboard Shortcuts**. Custom.ksc is saved to the same directory as the Creator application. If Creator is unable to find this file, it searches for, and loads, creator.ksc. If it cannot find creator.ksc, it loads a default keyboard map from source definitions.

Palette Files

The following files store settings for the default Color, Material, Texture, and Light Source palettes. If you have customized a palette, you can set it to be used as the default palette every time you use Creator by saving the palette with the default palette name.

flt1.color

Stores the default Color palette. When Creator launches, it searches for the file flt1.color. If found, Creator loads this file as the default Color palette. Creator looks for flt1.color first in the directory specified by the MPI_CREATOR_RESOURCEPATH environment variable, then in the working directory. The working directory for Windows is the directory where Creator is installed, and for IRIX it is the directory where Creator was launched. If Creator cannot find flt1.color in either location it loads a standard palette of 1024 shaded color definitions.You can save custom Color palette files with other names, but must reload them each time Creator starts up.

flt1.materials

Stores the default Material palette. When Creator launches, it searches for the file flt1.materials. If found, Creator loads this file as the default Material palette. Creator looks for flt1.materials first in the directory specified by the MPI_CREATOR_RESOURCEPATH environment variable, and then in the working directory. The working directory for Windows is the directory where Creator is installed, and for IRIX it is the directory where Creator is launched. If Creator cannot find flt1.materials, it loads a standard palette of 64 material definitions. You can save custom Material palettes with other names, but you must reload them each time Creator starts up.

default.txt

Stores the default Texture palette. When Creator launches, it searches for the file default.txt. If found, Creator loads this file as the default Texture palette. Creator looks for default.txt first in the working directory, and then in the directory specified by the MPI_CREATOR_TXTPATH environment variable.The working directory for Windows is the directory where Creator is installed, and for IRIX it is the directory where Creator is launched. If Creator cannot find default.txt in either location, it loads a Texture palette that contains no texture patterns.You can save Texture palettes with other names, but you must reload them each time Creator starts up.

mg1.lts

Stores the default Light Source palette. When Creator launches, it searches for the file mg1.lts. If found, Creator loads this file as the default Light Source palette. Creator looks for mg1.lts first in the directory specified by the MPI_CREATOR_RESOURCEPATH environment variable, and then in the working directory. The working directory for Windows is the directory where Creator is installed, and for IRIX it is the directory where Creator is launched. If mg1.lts cannot be found in either location, Creator loads a standard Light Source palette with one light source defined.

Terrain and Feature Preferences

The following files store terrain processing preferences, batch terrain settings, and feature projection preferences. If you are using Basic Terrain Conversion you will have only the <ded>.prefs file.

<ded>.prefs

Stores the *Terrain* window settings for a .ded file. The default file name is the name of the elevation data (.ded) file with a .prefs extension. This information includes the **Gaming Area** selection, contour band sizes and colors, **Face Color From** setting, and LOD setup, texture, map and terrain preference settings. The file is stored in the same location as the .ded file. If Creator cannot find this file in that location, it will create a new one with Creator's default *Terrain* window settings.

<.ded>1.bpf

Stores the preferences set in the Batch panel of the *Terrain* window. These include the X and Y area block sizes, the byte length of each area block, the number of blocks in X and Y, the area block naming convention, whether shading is on or off, whether or not to process feature data, and whether or not to translate the SW origin to zero. This file also keeps track of which area blocks are selected for processing and which area blocks have already been processed.

The Batch panel preferences file name is <.ded file name>1.bpf. (This replaces the <.ded file name>1.bat file used by MultiGen Series II and previous versions of Creator.) It is saved to the same directory as the DED file each time you generate terrain or close the *Terrain* window. If a file of that name already exists, it is renamed <.ded file name>1.bak, and a new <.ded file name>1.bpf is created (giving you one backup before the settings are overwritten).

dfad.prefs

Contains settings defined in the Feature Preferences window that control the geometry and texturing of features as they are projected. The default location for this file is the directory, but you can specify a different directory path and name for this file in a Project file where Creator is installed. When Creator loads a project it will look for the location and file specified in the project file. If it cannot find the file, or if a project file has not been created, it will search for a dfad.prefs file in the Creator directory. If Creator cannot find the dfad.prefs file, it will create a new file set to default source definitions.

dfadbat1.prefs

Stores rules and actions settings for projecting features. These preferences are set up in the *Action List* and *LOD Rule List* windows. The default name of the file is dfadbat1.prefs and its default location is in the directory where Creator is installed; however, you can specify a different path and file name (the file must have a .prefs suffix) in a project file in the *Terrain* window Project panel. When Creator loads a project it will look for the location and file specified in the project file. If it cannot find the file, or if a project file has not been created, it will search for a dfadbat1.prefs file in the Creator directory. If Creator cannot find the dfadbat1.prefs file, it will create a new file set to default source definitions.

mgprop.dat

Contains setup parameters for light point palettes. The default location for this file is the directory specified by MPI_CREATOR_RESOURCEPATH environment variable. If MPI_CREATOR_RESOURCEPATH has not been set Creator saves light point preset files to the directory where Creator is installed. You can save light point palettes to a different directory and file name (the file must have a .dat suffix), but you must load the custom file by selecting Light Point Presets from the Attributes menu each time you restart Creator.

When Creator loads a project it will look for the location and file specified in the project file. If it cannot find the file, or if a project file has not been created, it will search for a mgprop.dat file in the directory specified by the MPI_CREATOR_RESOURCEPATH variable. If Creator cannot find the file in this location, or if MPI_CREATOR_RESOURCEPATH has not been defined, it will create a new file set to default source definitions in the Creator directory.

<filename>.prj

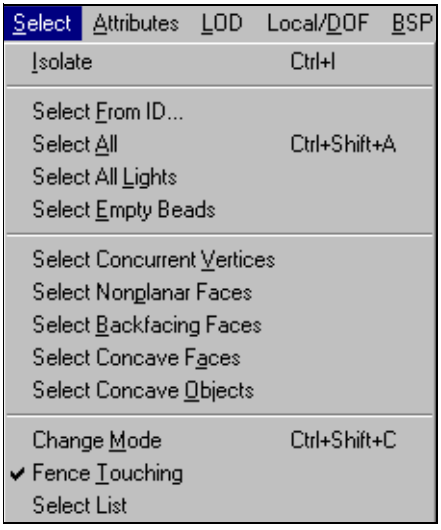
The project file specifies the .ded file, terrain preferences, dfad.prefs, dfadbat1.prefs. batch preferences, color, material, texture and light point preset files, and an output directory to be used for processing. When the project file is loaded, it automatically searches for and loads all the designated files and palettes. This allows you save and reuse color, LOD and property schemes and feature processing preferences whenever you need them. By default, project files are saved to the same directory as the .ded file, but they can be saved anywhere.

Road Definition File

default.rds.stores the default attributes for defining roads. Creator saves default.rds to the directory where Creator is installed, and searches for it in that same directory. You can save a road setup definition file with a different name (the file must have a .rds suffix) and in a different directory, but you must load it each time you restart Creator. (See “Applying and Saving a Road Definition” on page 1272 for details.)

25 Keyboard Shortcuts

Many Creator functions are activated with either the mouse or a keyboard shortcut. In pull-down menus, the keyboard shortcut that activates a function appears beside the function name.



Throughout the online help, keyboard shortcuts that activate functions appear in parentheses after the function name; for example, "**Select All** (Ctrl+Shift+A)." Press only the modifier (Shift, Ctrl, or Alt) that is explicitly called for; pressing a modifier that isn't called for may execute an entirely different shortcut.

> **Note:** The focus must be in the Graphics view in order for the keyboard shortcuts to work. If you have selected a node in the Hierarchy view, right click in the Graphics view. This sets the focus without deselecting the node in the Hierarchy view.

You can also display a menu's contents by pressing Alt + the underlined letter of the menu on the menu bar. For example, to open the **Palettes** menu, you would press Alt+P.

If you want to customize the keyboard shortcuts, see "Creating Keyboard Shortcuts" on page 903.

Function Key Definitions

Key	Action
F1	Help
F2	OK
F3	Next
F4	Cancel
F5	Display/Hide View Controls, Toolboxes

File Menu Shortcuts

Action	Shortcut
New File	Ctrl+N
Open File	Ctrl+O
Save File	Ctrl+S
Close Database	Ctrl+W
Exit MultiGen	Ctrl+Q

Edit Menu Shortcuts

Action	Shortcut
Undo	Ctrl+Z
Delete	Delete
Cut	Ctrl+X

Action	Shortcut
Copy	Ctrl+C
Paste	Ctrl+V
Insert Color	Ctrl+R
Insert Texture	Ctrl+T
Color From Face	Ctrl+Shift+R
Texture From Face	Ctrl+Shift+T

View Menu Shortcuts

Action	Shortcut
Toggle Graphics View/Hierarchy View	S
Reset View	U
Fit On Select	V
Fit On Grid	Ctrl+Shift+V
Center Select	C
Fit On Pick	Ctrl+F

Info Menu Shortcuts

Action	Shortcut
Preferences	Ctrl+Shift+P
Statistics Windows IRIX	 Ctrl+? Ctrl+K

Select Menu Shortcuts

Action	Shortcut
Isolate Select	Ctrl+I
Select All	Ctrl+A
Deselect All	Ctrl+D
Change Mode	Ctrl+Shift+C

Attributes Menu Shortcuts

Action	Shortcut
Modify Attributes	Ctrl+=
Rename Node	Ctrl+J
Animate	Ctrl+Shift+A
Calculate Shading	Ctrl+G

LOD Menu Shortcuts

Action	Shortcut
More Detail	Ctrl+M
Most Detail	Ctrl+Shift+M
Less Detail	Ctrl+L
Least Detail	Ctrl+Shift+L

Palettes Menu Shortcuts

Action	Shortcut
Color Palette	Ctrl+P

Window Menu Shortcuts

Action	Shortcut
Cascade	Shift+F5
Tile	Shift+F4
Zoom View	Shift+F6
Standard View	Shift+F9
Cad View	Shift+F1

Eyepoint and Tracking Plane Control Shortcuts

Action	Shortcut
Retrieve Eyepoint	Eyepoint Number (1-9) either number keys or keypad w/Num Lock
Retrieve Tracking Plane	Ctrl+Track number (1-9), either number keys or keypad w/Num Lock
Save Eyepoint	Shift+Eyepoint number (1-9)
Save Eyepoint and Tracking Plane	Ctrl+Shift+Eyepoint number (1-9)

View Control Shortcuts

Action	Shortcut
Draw Mode	D
Toggle Texture	T
Toggle Lighting On/Off	L
Toggle Grid	G
Toggle Draw Order (z-buffer on/off)	Z
Toggle View Panel and Toolbox Display On/off	F5
Shrink Mode	A
Perpendicular Grid	P
Trackplane From Face	F

Texture Mapping Shortcuts

Action	Shortcut
Apply the Current Texture Mapping	Ctrl+Insert Texture
Set current texture mapping to selected face	Alt+Get Texture From Face
Set current texture and current texture mapping to selected face	Ctrl+Get Texture From Face

Tool Shortcuts

Action	Shortcut
Group Mode	Shift+G
Object Mode	Shift+O
Face Mode	Shift+F
Edge Mode	Shift+E
Vertex Mode	Shift+V
LOD Mode	Shift+L
Create	Shift+C
Wall	Shift+W
Peak	Shift+P
Irregular Polygon	Shift+I
Put Texture (3 Point Put) Windows IRIX	 Shift+, Shift+X
Modify Vertex	Shift+M
Insert Color	Ctrl+R
Color From Face	Ctrl+Shift+R
Insert Texture	Ctrl+T
Texture From Face	Ctrl+Shift+T
Translate	Shift+T
Scale	Shift+S
Duplicate	Shift+D
Rotate About Point	Shift+R

Graphics View Shortcuts

Action	Shortcut
Change eyepoint	Right mouse (Right mouse+Middle mouse for continuous motion)
Pan image	Ctrl+Right mouse (Ctrl+Right mouse+Middle mouse for continuous motion)
Zoom image	Shift+Right mouse (Shift+Right mouse+Middle mouse for continuous motion)
Simultaneously select and make parent	Alt+Left mouse
Add to selected graphics	Shift+Left mouse
Hide (toggle off geometry	H

Hierarchy View Shortcuts

Action	Shortcut
Make node parent	Alt+Left mouse
Expand hierarchy	Middle mouse
Contract hierarchy	Alt+Middle mouse
Toggle node on/off	H

Customizing Keyboard Shortcuts

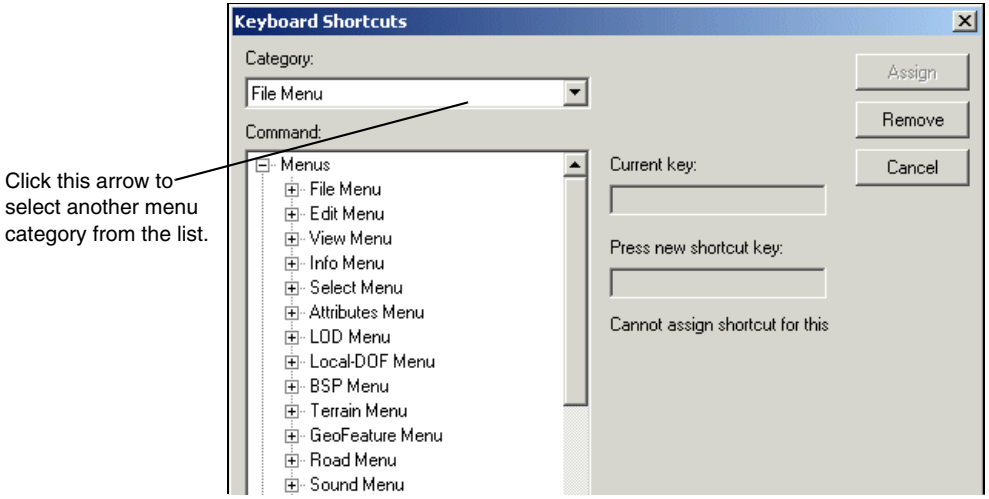
Creator has keyboard shortcuts for most menu commands and modeling operations, such as opening and saving files, changing the drawing mode, translating, rotating,

and scaling geometry, or changing the modeling mode. You can create custom keyboard shortcuts for any of these, define shortcuts for commands that do not currently have shortcuts defined, or delete shortcuts that you do not use.

Custom keyboard shortcuts are saved in the file `custom.ksc` in the `\MultiGen-Paradigm\Creator` directory.

Creating Keyboard Shortcuts

To display the list of keyboard shortcuts, select **Info/Keyboard Shortcuts**. The *Keyboard Shortcuts* window contains a list of every command for which you can define a shortcut, and its current shortcut, if any. The commands are listed by function categories. To view commands in another category, click the arrow to the right of the current category to display the contents of the list, and then select a new category. You can also click a category in the **Command** list.



To use a keyboard shortcut, select the item or items to be acted on, and press the shortcut keys.

To Create or Change a Keyboard Shortcut

Step 1. Choose **Info/Keyboard Shortcuts**.

Step 2. Choose the category that contains the shortcut you want to create or change.

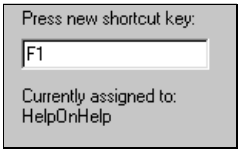
Step 3. Select the command from the **Command** list to display its current shortcut.

Step 4. Click within the **Press new shortcut key** field.

Step 5. Type the new shortcut.

Step 6. Click **Assign** to assign the new shortcut to the command.

Note: If the keyboard shortcut you entered has already been defined for another command, a message appears below the field.



Clicking **Assign** removes the keyboard shortcut from the command it is currently assigned to and assigns it to the new command.

When you are done assigning shortcuts, click **Close** to close the *Keyboard Shortcuts* window.

To Delete a Keyboard Shortcut

Step 1. Choose **Info/Keyboard Shortcuts**.

Step 2. Select the command to be modified. Its current keyboard shortcut displays in the **Current key:** field.

Step 3. Click **Remove** to unassign the shortcut.

To Restore the Default Keyboard Shortcuts

Step 1. Choose **Info/Keyboard Shortcuts**.

Step 2. Click **Restore Default Settings**.

26 Binary Separating Planes

Creator supports three rendering algorithms: fixed list (also called priority listing), in which the drawing priority of each item is determined by its position in the database hierarchy; Z-buffer, in which drawing priority is determined by drawing the items that are closest to the eyepoint last; and BSP, in which visual priority is determined by referencing each item's position relative to invisible planes inserted in the database.

BSP rendering is efficient because visual priority sorting is built into the structure of the database, requiring far fewer computations than either Z-buffering or fixed-list rendering. Subfaces do not present the problem they do in Z-buffered systems, because they are treated as children of the faces to which they are attached, rather than as coplanar faces.

BSP rendering is combined with fixed listing to establish drawing priorities in your database. Priority sorting by BSPs can resolve rendering problems that may be difficult or impossible to solve by fixed listing alone.

Designing Separable Databases

Given a properly constructed database, Creator can calculate and insert a nested series of BSPs that produce efficient rendering.

What Makes a Database Separable?

A database is separable if a plane can be inserted between each pair of nodes selected for separation without intersecting any geometry, and if none of the convex hulls around the items being separated are interpenetrating.

A *convex hull* is the smallest convex region that completely encloses all the vertices of each item selected for separation. Testing convex hulls is the polygonal equivalent of "shrink wrapping" each item and making sure none of the shrink wraps intersect one another.

BSP rendering requires some special considerations when modeling:

- All vertices in each face in your database must be coplanar.

- The polygons that represent BSPs cannot intersect any faces, so models must be constructed in such a way that BSPs can be inserted between nodes at significant points, without intersecting any of the geometry.
- Different models in the database must be separable from each other.

Once the above conditions are met and the database is separated, it can be rendered quickly and correctly from all viewing angles. In contrast, a fixed listed database can be drawn quickly, but there may not be a hierarchy that guarantees the database draws correctly, while a Z-buffered database draws correctly, but requires intensive computation and/or expensive graphics hardware to calculate relative priorities. The performance gains in rendering make the extra effort required to separate a database worthwhile.

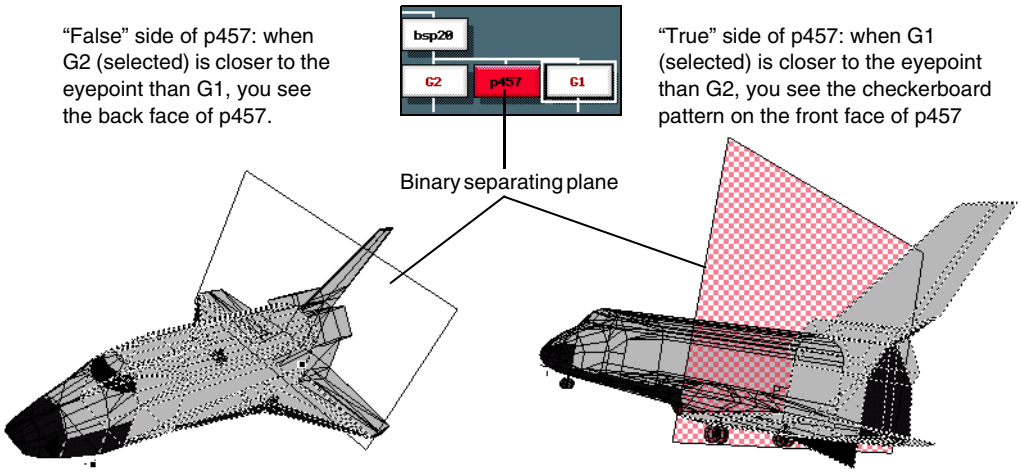
Combining BSPs and Fixed Listing

Because it is not practical to separate every face in a database with a BSP, any nodes that are not separated are fixed listed (prioritized by their positions in the database). This means you can separate a database at high levels of the hierarchy while relying on fixed listing to determine drawing priority at low levels. For example, it may be adequate to separate group nodes and rely on fixed listing to prioritize the objects attached to each group. The depth of separation necessary depends on the complexity of the geometry. The commands in the BSP menu calculate optimal BSP positions and place them in the database. Any geometry that is not separated by BSPs is fixed listed automatically.

Understanding BSP Trees

A binary separating plane tree consists of a left subtree, a polygon representing the binary separating plane, and a right subtree, all attached to a BSP node. The term *binary separating plane* refers to the polygon that separates the left and right subtrees of a BSP node. The BSP node is simply an attachment point for these three items.

The right subtree of a BSP node is sometimes referred to as the *true* side of the binary separating plane and is the side from which the plane is visible (filled with a checkerboard pattern) when the display of separating planes is enabled in the Graphics view. In this context, the terms *true* and *false* are synonyms for frontfacing and backfacing. To draw correctly, each binary separating plane must face its right sibling in the hierarchy. The Creator tools that create binary separating planes take care of this automatically.



Tips for Creating a Separable Database

Many of the techniques described in this section are good modeling practices for any system, but for BSP display they are essential.

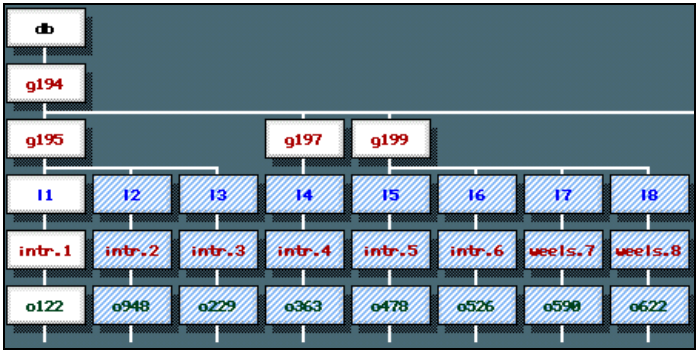
- Plan the major divisions of each model before you draw it. For example, if you want two or three major divisions in each direction, you must construct the model with many divisible sections. This requires more planning than if you decide that you just want one BSP down the middle of a model, and then rely on fixed listing for drawing the rest of the model.
- Always use the tracking plane when creating geometry. If you do not use the tracking plane when placing control points, the vertices of the polygon you construct may not be coplanar.
- Always use the tracking plane when performing a transformation. This ensures that the database is still separable when vertices are moved during a translate or scale operation.
- Be careful when performing operations such as snapping a vertex to an existing point. The polygon that results may not be coplanar.
- When modeling a symmetrical object, build one side, use **Check Separability** to make sure it is separable, and then Mirror the geometry across the tracking plane to build the other side (see "Mirror Tool" on page 170). This not only saves

drawing and correction time, it also ensures that Creator can insert a BSP down the middle of the model, providing efficient separation for the realtime system.

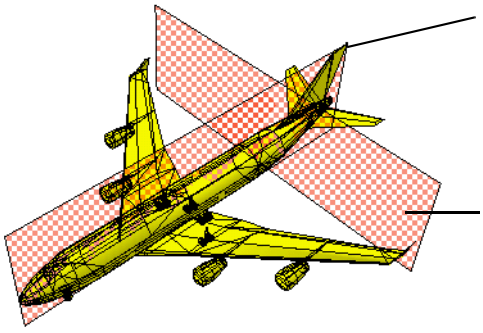
- Create tubular structures (such as airplane fuselages) with an even number of sides. This ensures that at least one BSP can produce a major division of the structure.

  Note: When tubular structures have an even number of sides, they can appear to be asymmetrical when rotated. On Z-buffered systems, you can specify an uneven number of sides for tubular structures, to compensate for this drawing artifact. For BSP rendering, the ability to place a major division in a large structure is more important. If you create even-sided cylinders with 8 or 10 sides, you can divide the structure with at least one BSP, and the rotational distortion with this number of sides is insignificant.

- Separate models individually, and then separate them from each other. Remember to separate each external reference in its own database.
- Pay attention to the hierarchy of your database. The Creator separation commands separate groups and objects (as well as LOD, DOF, and Switch nodes). Efficient grouping, especially in a large database, produces efficient separation, culling, and rendering.
- For example, the following illustration shows part of the database structure of a model included in the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory. Each LOD is built from a single object, which has as its children every polygon that will be visible for that LOD. There are no objects or groups to separate from each other. This model could not be used in a BSP system without extensive rearrangement of the database, inserting objects, and perhaps groups, in the hierarchy of each LOD.

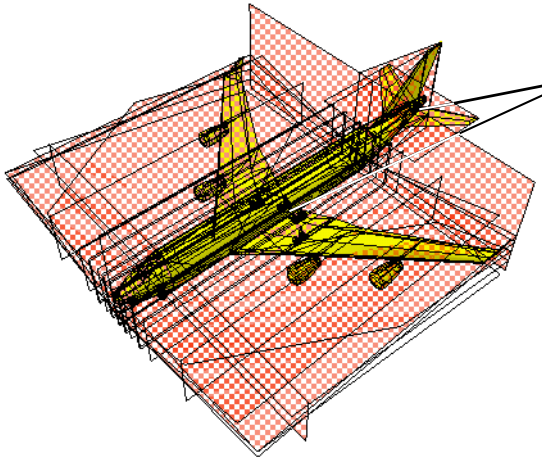


- The following figures show the major BSP divisions and the full separation of an airplane that was constructed for display in a BSP system.



The fuselage has an even number of sides, and the tail section is divisible along the same pathway, so a BSP can be inserted lengthwise, separating the left and right sides of the plane.

The wings do not extend down to the tail, so a BSP can be inserted through a cross-section of the fuselage, dividing the front and back of the plane.



The wings and tail fins must be connected to the fuselage without using interpenetrating hulls

Careful construction produces a model that can be separated by many planes in different directions. This maximizes the efficiency of the rendering system.

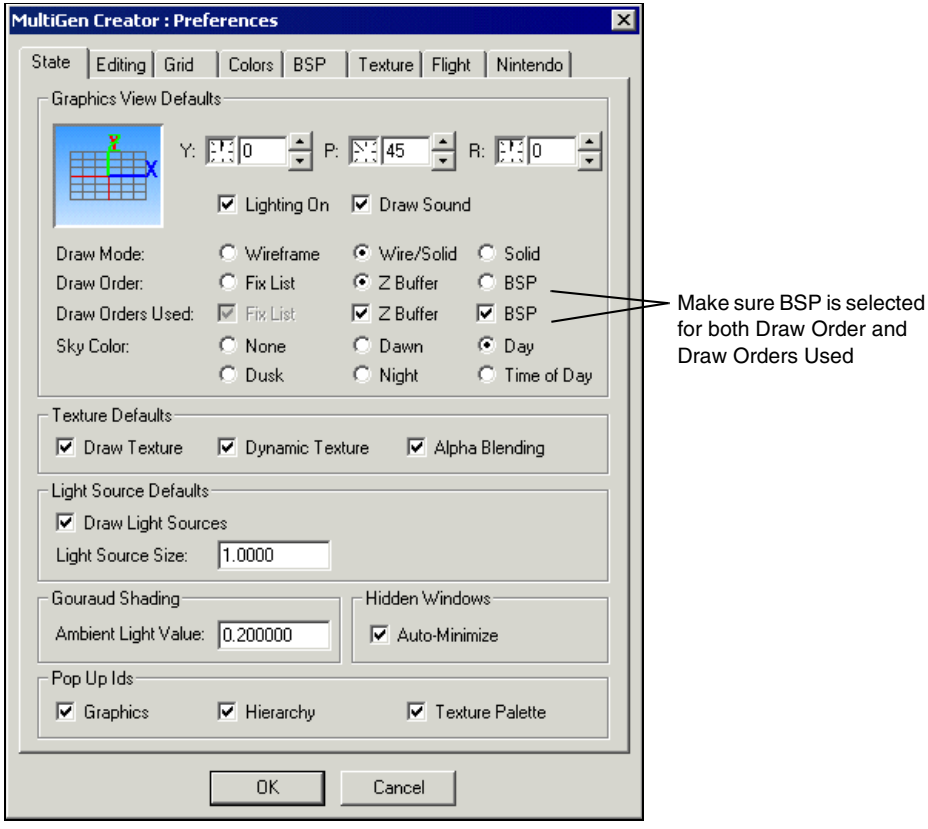
Enabling BSP Draw Order

To view the effects of BSP drawing priority, you must enable BSP drawing order. To do this, choose **BSP** from the **Draw Order** menu in the View panel, or press the Z key repeatedly to toggle between draw orders in the Graphics view. Before you can select BSP draw order with the Z key; however, you must enable BSP draw order in the State Preferences panel. To change the preference setting:

Step 1. Choose **Info/Preferences** and click the **State** tab. The State preferences appear.

Step 2. Under **Draw Order**, enable **BSP**.

Step 3. Under **Draw Orders Used**, make sure **BSP** is checked, and then click **OK**.



Automatically Separating a Database

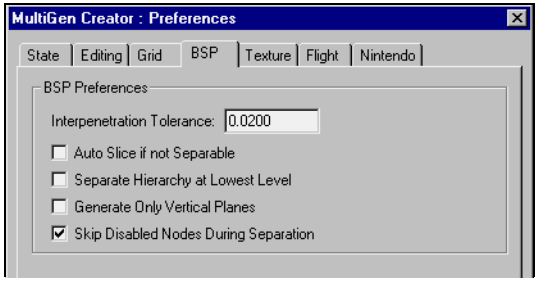
The BSP menu contains several functions that automatically separate the selected portion of the database hierarchy.

- **Check Separability** creates a temporary separation of the database, allowing you to preview the effect of **Separate Selected**.
- **Separate Selected** separates the database at any hierarchy level, discarding groups and other components of the hierarchy, and replacing them with BSP nodes. Because **Separate Selected** can rearrange the hierarchy, you should not use it when a structural change will disrupt functionality (with DOF nodes, for example).
- **Separate Children** separates the children of each selected group, LOD, switch, or DOF node without regard to the node type of the children or to the selected modeling mode. Unlike **Separate Selected,** this function preserves intermediate levels of hierarchy in the separated subtrees.
- **Recursive Separation** traverses the selected portion of the hierarchy and separates parent nodes and their children all the way down each selected subtree, preserving intermediate levels of the hierarchy in the process. You must be in group, LOD, DOF, switch, or BSP mode to perform a **Recursive Separation**.

Setting BSP Preferences

BSP preference settings affect the results you get when you separate a database. For this reason, you should set BSP preferences *before* you attempt to separate a database.

To set BSP preferences, choose **Info/Preferences**, and then click the **BSP** tab.



BSPs have the following preferences:

Interpenetration Tolerance

Specifies the minimum distance between two convex hulls that are considered to be interpenetrating, giving you the ability to separate elements that are slightly overlapping. The **Interpenetration Tolerance** you set here should match the setting on your realtime system; if you set a greater **Interpenetration Tolerance** than the one allowed by your realtime system, flickering will occur when you run the database in realtime.

Auto Slice if not Separable

Slices the database geometry as necessary to insert separating planes during **Check Separability** (see "Using Auto Slice to Insert BSPs" on page 928).

Separate Hierarchy at Lowest Level

Controls whether Creator searches the database from the top of the selected tree(s) to the bottom for nodes that match the current modeling mode. This preference affects only **Separate Selected** and **Check Separability**. If this preference is set, Creator separates the lowest occurrence of the selected modeling mode in the hierarchy. If this preference is cleared, the hierarchy is separated at the highest possible level.

Caution! Separating a database with **Separate Selected** and **Separate Hierarchy at Lowest Level** may disrupt level-of-detail, clipping, and other nodes that interact with the hierarchy. It is safer to select the nodes you want to separate directly, or to use **Recursive Separation** (see "Using Recursive Separation" on page 920).

Generate Only Vertical Planes

Constrains the automatic generation of separating planes to vertical planes only. This is useful in separating terrain areas or in separating objects positioned on terrain.

Skip Disabled Nodes During Separation

Ignores nodes that are not visible in the Graphics view during automatic separation. This is useful for turning off areas of the database that separated successfully, so you can work on the areas that did not.



To turn the display of nodes on or off, select the nodes, and then click **Toggle Display** in the **Hierarchy** toolbox.

Using Check Separability

Check Separability creates temporary planes in the Graphics view of the database, showing the areas where a database will separate or fail to separate. No BSP nodes or planes are inserted in the database hierarchy. This allows you to troubleshoot and correct portions of the database before inserting BSP structure.

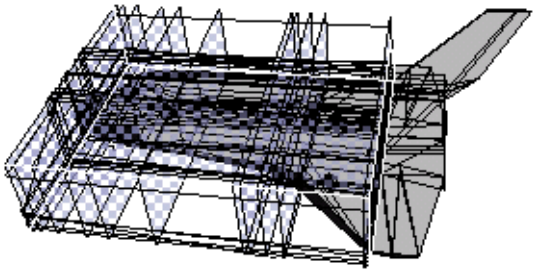
To separate two nodes in the database, Creator must define a plane that does not intersect the geometry attached to either node. To create a preliminary separation:

Step 1. Set the modeling mode to the type of node you want to separate.

Step 2. Select the portion of the database you want to separate (or select the entire database).

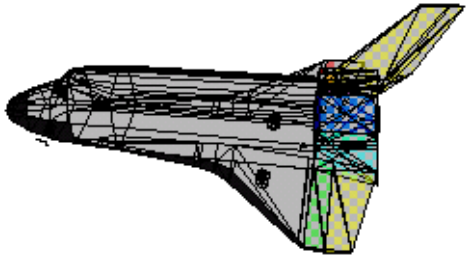
Step 3. Choose **BSP/Check Separability**.

If the separation succeeds, a set of preliminary separating planes appears in the Graphics view, drawn in the current color with a transparent checkerboard pattern. These planes are temporary and are not saved with the database. To make the separation permanent, reselect the part of the database on which you just performed **Check Separability**, and then choose **Separate Selected**.



Temporary planes show how this model can be successfully separated

If separation is unsuccessful, the areas causing problems are indicated by checkerboard patterns in different colors.



The objects that could not be separated in the tail section of this model are highlighted by patterns in different colors

To fix a database that fails to separate, see “Correcting Inseparable Databases” on page 926.

Removing Temporary Planes and Hulls

Temporary planes are created in the Graphics view when you use **Check Separability**, and separation is successful. Temporary hulls are created around elements (groups or objects, for example) when you use **Check Separability**, **Separate Selected**, **Separate Children**, or **Recursive Separation**, and separation is unsuccessful. To remove these temporary planes or hulls, choose **BSP/Remove Plane/Hulls**.

Using Separate Selected

Separate Selected separates the selected items, rebuilds the database hierarchy, and attaches each newly separated subtree to the *lowest common ancestor* (the first node, as you travel back up through the database hierarchy, that is an ancestor of each selected node).

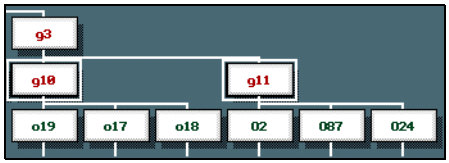
Separate Selected does not use the current database parent to determine what to separate. The type of node that is separated (group, object, and so forth) is determined by the current modeling mode. For example, in Group mode, only the selected groups are separated. In Object mode, selected objects are removed from their groups, placed under a BSP node, and then separated from each other.

The following illustrations show the effect of modeling mode on **Separate Selected**.

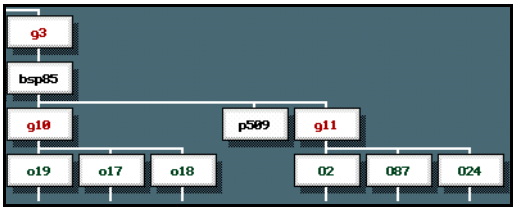
When you choose **BSP/Separate Selected** in Group mode, the selected groups are separated by BSPs. The objects remain as children of their original parent groups.



Group mode enabled



Original hierarchy

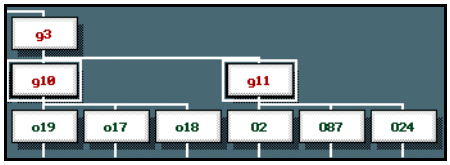


Hierarchy produced by Separate Selected

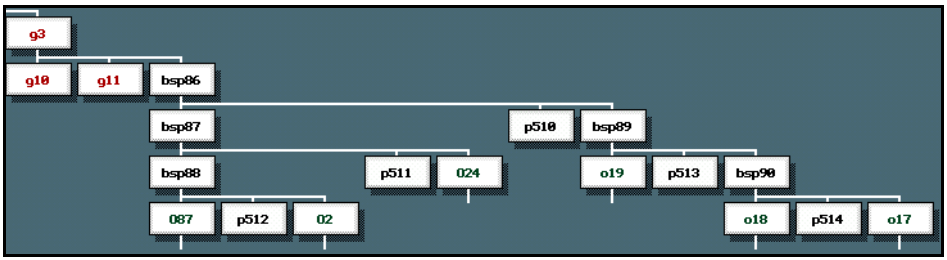
In Object mode, when you select groups and choose **BSP/Separate Selected**, the groups are not separated. The objects that are their children, however, are removed, placed under a BSP node, and then separated. After the separation, the original parent groups are empty.



Object mode enabled



Original hierarchy



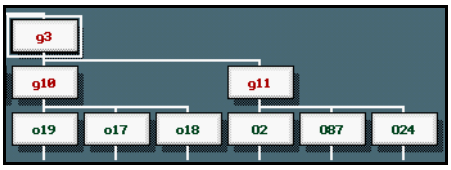
Hierarchy produced by Separate Selected

Separate Selected can radically alter database hierarchy. If your database contains LOD, clipping, or other nodes that make use of the database hierarchy, use **Separate Children** or **Recursive Separation** instead of **Separate Selected**.

To Use Separate Selected

Step 1. Select the modeling mode from the list in the Toolbar.

Step 2. Select the portion of the hierarchy you want to separate.



Selecting a parent also selects the children. In Group mode, if you selected g3, separation would fail, however, because there is no other group at the same level as g3 (the first group encountered). In Object mode, separation would succeed because there are other object nodes at the same level as the first object node encountered (O19).

Step 3. Select **BSP/Separate Selected**.

A node that matches the current modeling mode does not need to be at the top of the selected tree, but when Creator finds the first instance of this node type in the hierarchy, there must be at least one other node of the same type on the same level. For example, if Group mode is enabled, Creator will traverse the selected tree. When it finds the first Group node, there must be at least one more Group node at that level. If there is not, you see the error message "Less than 2 convex hulls found. Separation aborted."

It is not unusual for a database to fail to separate on the first attempt. If this happens, Creator issues an error message and generates a report of the problems that were encountered (see "Correcting Inseparable Databases" on page 926).

Using Separate Children

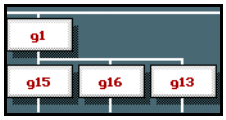
Separate Children separates the children of the highest group, LOD, switch, or DOF nodes in the selected portion of the hierarchy. The parent node type is determined by the modeling mode.

Separate Children adds BSP nodes to the hierarchy without changing the relative positions of the separated children. This, combined with the ability of **Separate Children** to process different (sibling) node types, makes it easy to mix fixed listing and binary separation in the same database.

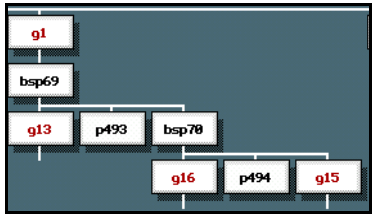
In the following example, **Separate Children** separates the children of the selected group.



Group mode enabled



Original hierarchy



Hierarchy produced by Separate Children

To Use Separate Children

Step 1. Set the modeling mode from the Toolbar.

Step 2. Select the portion of the hierarchy you want to separate.

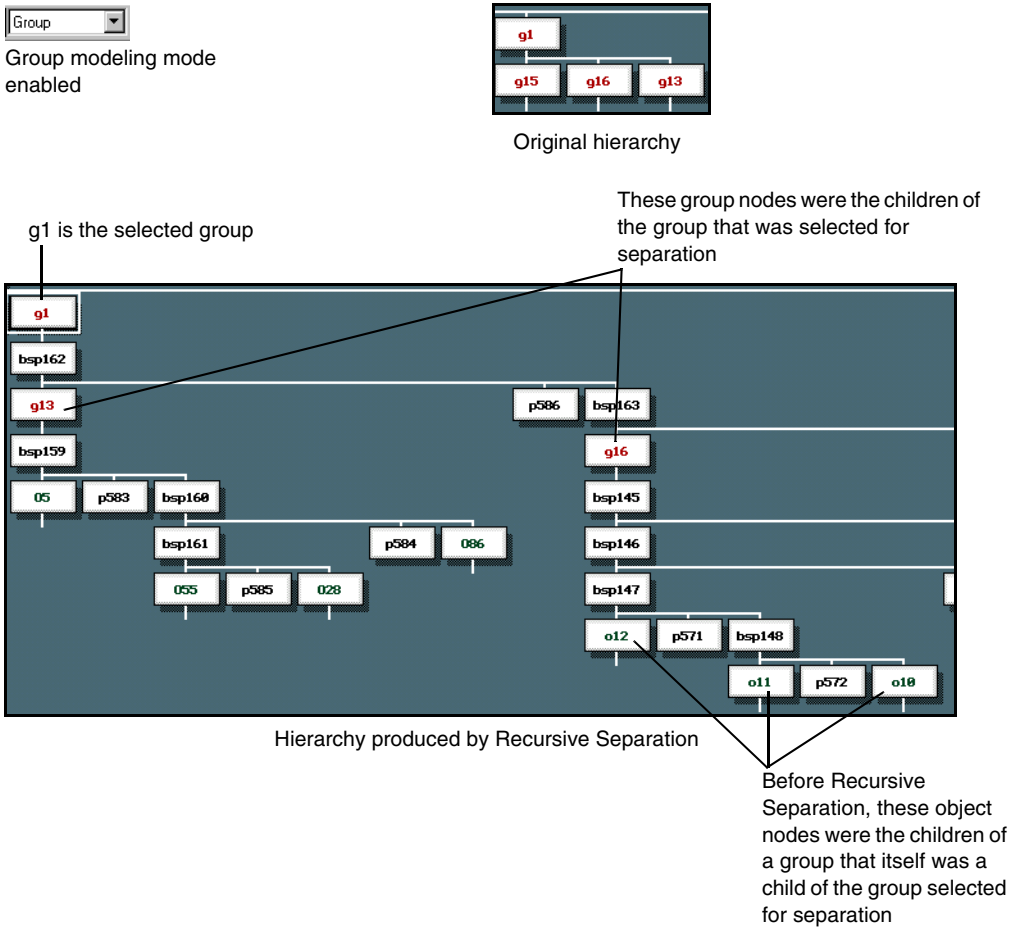
At least two nodes that match the modeling mode must be at the top of the selected tree.

Step 3. Select **BSP/Separate Children**. If the separation fails, see "Correcting Inseparable Databases" on page 926.

Using Recursive Separation

Recursive Separation applies the **Separate Children** function all the way down the database tree, separating all child nodes in the selected portion of the hierarchy that match the current modeling mode (group, object, LOD, or DOF) and *their* children.

In the following example, **Recursive Separation** separates the groups that are the children of the selected group and the objects that are the children of the child groups.



When a model is properly constructed, **Recursive Separation** builds an efficient structure for rendering. LODs are also taken into account and separated without disrupting the LOD structure. Databases with animated groups or DOF nodes should not be separated by **Recursive Separation**, however, because the structural changes that occur when BSP nodes are inserted may introduce errors.

To Use Recursive Separation

Step 1. Select **Group**, **Object**, **LOD**, or **DOF** modeling mode from the Toolbar. The modeling mode determines which nodes and their children are separated.

Step 2. Select a node at the top of the subtree to be separated.

Step 3. Choose **BSP/Recursive Separation**.

If separation fails, see "Correcting Inseparable Databases" on page 926.

Manually Separating a Database

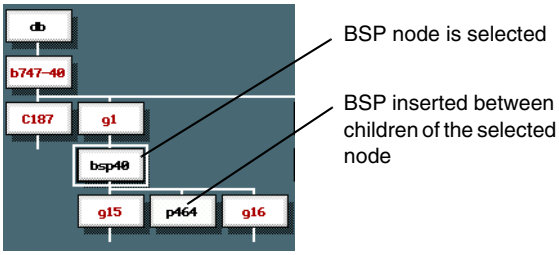
The separation commands in the BSP menu automatically create BSP nodes and planes at logical positions in the database.You can also use the **Create BSP** tool and the **BSP/Create BSP Plane** command to create and place BSPs manually, to control their exact positions in the database, or to add BSPs to a database that already has been separated.

Creating a BSP Node

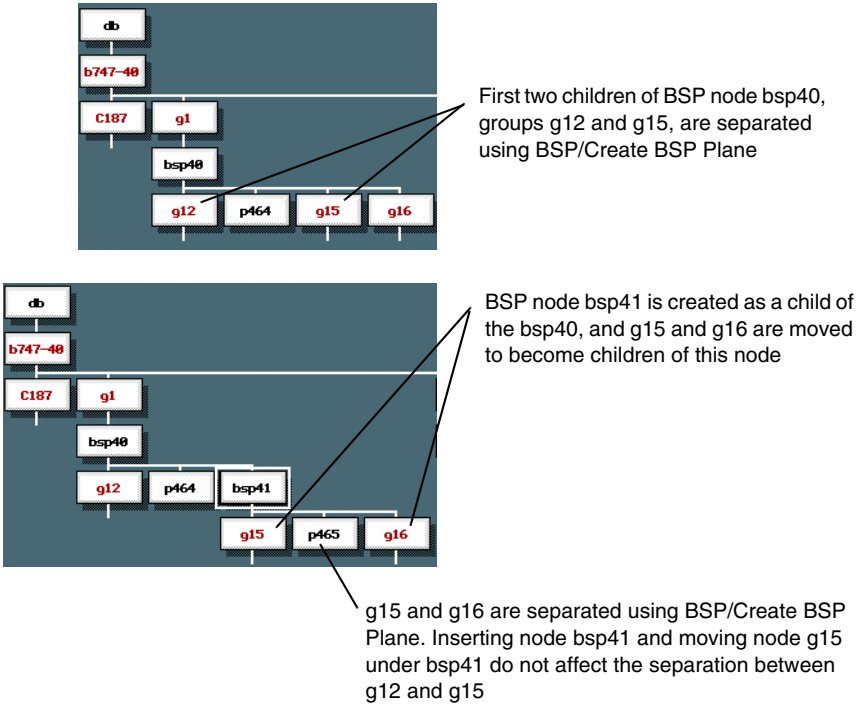
The **Create BSP** tool inserts a BSP node under the current parent. The groups or objects you want to separate must be made children of this node, and then separated using **BSP/Create BSP Plane**. (See "Inserting a BSP Manually" on page 923.)

Inserting a BSP in the Database

BSP/Create BSP Plane inserts a binary separating plane between the children of a selected BSP node, at the current tracking plane position.



If the BSP node has more than two children, the BSP is inserted between the first two children, from left to right. If you want to separate more than two children, you must nest the BSP nodes in the hierarchy, as shown in the following figure.



Inserting a BSP Manually

To insert a BSP manually:

Step 1. Select the parent of the nodes you want to separate. The parent must be a group, LOD, DOF, or switch node.



Step 2. Click the **Parent** button at the lower-left corner of the Creator window.



Step 3. Open the **Create** toolbox.

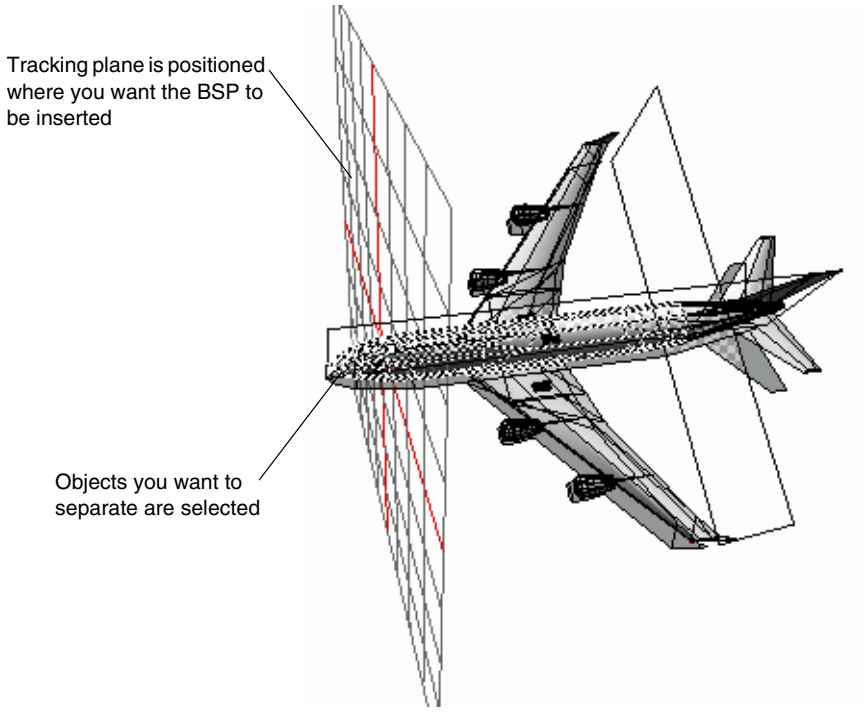


Step 4. Click **Create BSP**. A new BSP node is attached to the current parent.

Step 5. Select the new BSP node and click the **Parent** button to set this as the new parent.



Step 6. Select the objects or groups you want to separate.





Step 7. Open the **Hierarchy** toolbox.

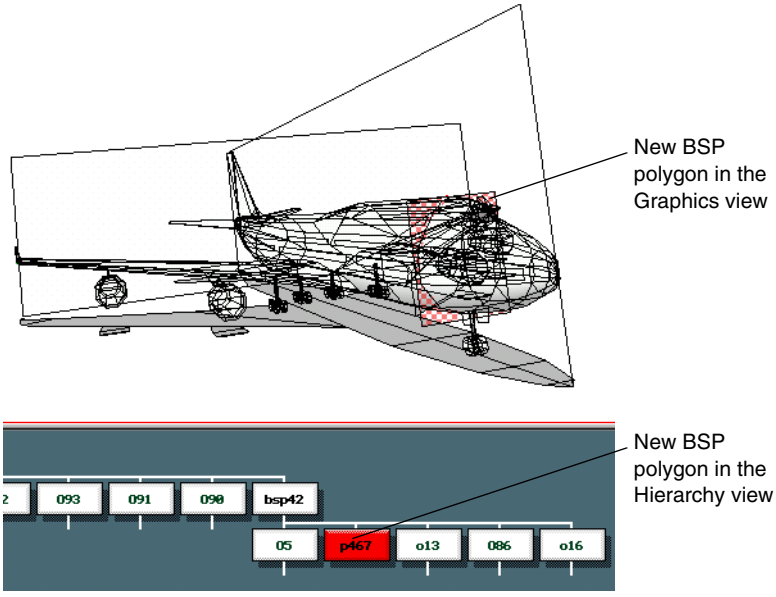
Step 8. Click **Attach**. The items you want to separate are now children of the BSP node.



Step 9. Place the tracking plane in the position and orientation you want to assign to the new BSP.

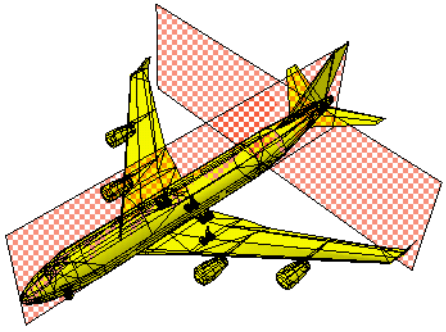
Step 10. In the Hierarchy view, select the BSP node.

Step 11. Select **BSP/Create BSP Plane**. A new BSP separates the selected objects.



Displaying Binary Separating Planes

Separating planes appear in the Graphics view as rectangles filled with a transparent checkerboard pattern. The color of each plane is the current color when the plane is generated.



Using Display Planes

To toggle the display of binary separating planes in the Graphics view, choose **BSP/Display Planes**. A check appears next to this menu item when the display is enabled. Disabling the display of BSPs allows you to get a clearer view of the geometry of the database.

Using Select Planes

BSP/Select Planes selects only the BSPs that are children of the selected node. If no node is selected, **Select Planes** selects all BSP polygons in the database. When **BSP/Display Planes** is enabled, you can use **Select Planes** to display or hide selected BSP planes. For example, you can select a parent node in a tree, select **BSP/Select Planes**, then use the **Toggle Display** tool in the **Hierarchy** toolbox (Shift-3) to disable only the selected BSP planes.

Collapsing a BSP Structure

There are three ways to remove BSPs:

- To remove the temporary BSPs that appear when you choose **BSP/Check Separability**, choose **BSP/Remove Plane/Hulls**. This command works only on temporary BSPs.
- To remove BSPs immediately after you perform a separation, choose **Edit/Undo**.
- To remove BSPs at any other time, select the portion of the database tree you want to change, and choose **BSP/Collapse BSP**. This removes the BSP nodes and binary separating planes in the selected subtree. The separated children are attached to the parent of the BSP node, producing a hierarchy that is fixed listed.

  Note: Do not choose **BSP/Select Planes** and press **Delete** to eliminate BSPs. This deletes the BSP polygons, but does not remove the BSP nodes.

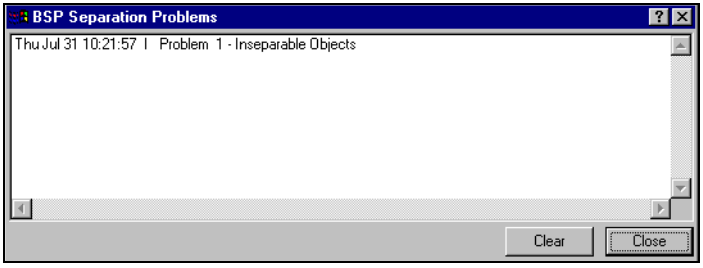
Correcting Inseparable Databases

A database fails to separate if the convex hulls formed around the items being separated are interpenetrating, or if Creator is unable to find a plane that does not intersect each pair of items being separated. In either case, Creator issues an error

message and highlights the offending portions of the database by enclosing them in patterned hulls of different colors.

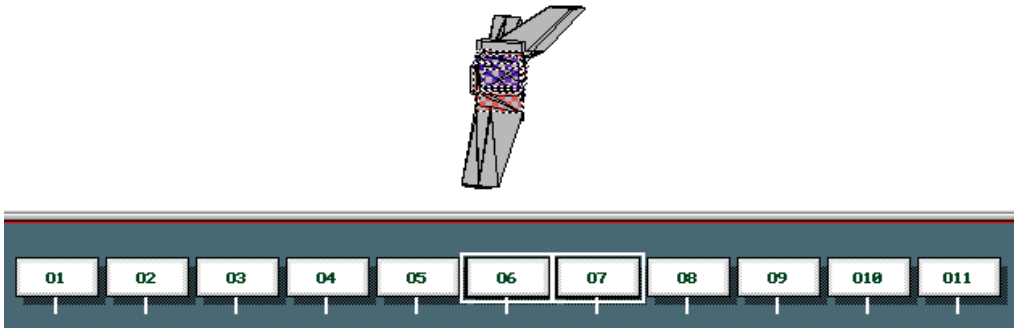
Using Diagnosis

You can view individual problems by choosing **BSP/Diagnosis**. The **BSP Separation Problems** dialog box appears.



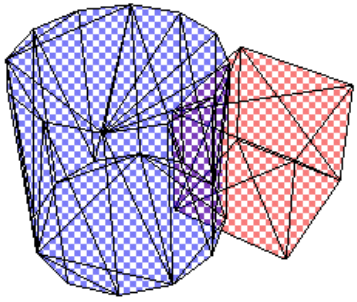
To select the geometry that caused a problem, select the problem from the list in this dialog box. Once you select the problem area, you can isolate the geometry by choosing **Select/Isolate** (Ctrl+I). In the isolated view, you can test each portion of the isolated geometry to find where the separation problem occurs.

In the following example, a part of a model that failed to separate was isolated, and the separability between objects was tested by selecting the object nodes in pairs and then using **BSP/Check Separability**. The checkerboard patterns on objects **06** and **07** indicate that they cannot be separated without some correction to the geometry.



Correcting Interpenetrating Hulls

If Creator finds interpenetrating hulls, a checkered bounding box is displayed around the hulls of each of the interpenetrating items. Before proceeding, you must correct the geometry manually (by moving the elements so they no longer intersect, or by removing the part of the geometry in one element that interpenetrates the other, for example).

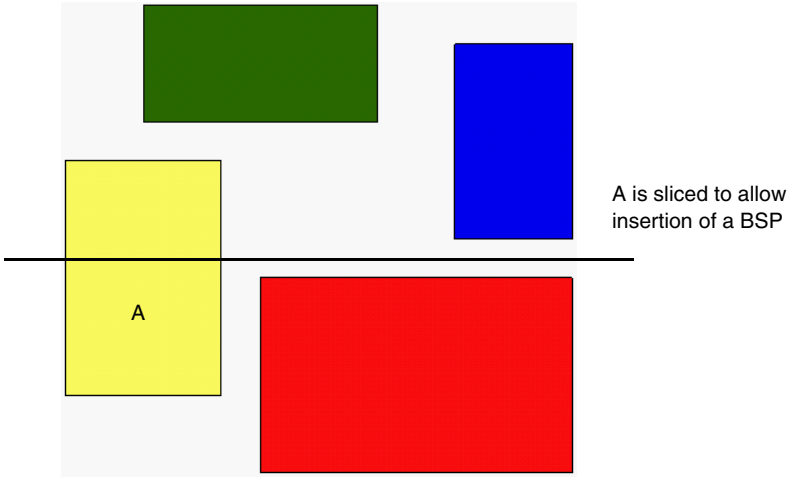


Interpenetrating hulls must be corrected either by separating the objects entirely (if they are separate models), or by slicing at the point where they intersect and removing the interior portions. A BSP can then be inserted between the objects. Some cleanup work will need to be done to adjacent polygons, and these will be rendered with fixed listing.

Using Auto Slice to Insert BSPs

If Creator finds a pair of items that cannot be separated by inserting a plane, you can either correct the geometry manually or turn on the BSP preference **Auto Slice if Not Separable**. When this preference is on, Creator attempts to slice inseparable items to

allow for the insertion of a separating plane. The two new items that result from each slice are given new node identifiers and are attached to the original parent.

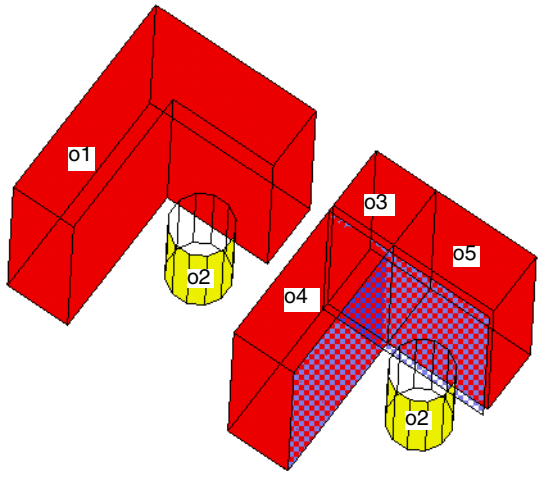


Note: The auto-slice function cannot resolve cases of interpenetrating hulls.

Separating Concave Objects

For best results, avoid the use of concave objects when building a database that is to be separated. To search a database for concave objects, choose **Select Concave Objects** from the **Select** menu. The **Slice** tool is useful for dividing concave objects into separable objects (see “Slice Tool” on page 174).

Because o1 is concave, o1 and o2 cannot be separated at Object level



Breaking o1 into three objects (using the Slice tool) makes it separable at Object level

BSP Separation and Isolated Views

A common practice when resolving separation problems is to select the components that will not separate, isolate them (using **Select/Isolate**), make the corrections, and then perform the separation.

Sometimes when you perform a separation in an isolated view, you see the message "Separation succeeded" at the bottom of the window, but all items in the isolated view disappear. This is because the display in the isolated view is built from the highest-priority node that was selected when you chose **Select/Isolate**. If, during separation, a BSP node is inserted as the parent of the highest-priority node, its position changes to the next-lowest level in the tree, and Creator can no longer find it to build the list. If you performed a separation that did *not* insert a BSP above this node (**Separate Children**, for example), the isolated view displays the successful separation of the items.

Check Separability does not change the database geometry, so you can use this command successfully in an isolated view. If you want to perform your actual separations on isolated portions of the database, however, a better method to use is to select the components you *do not* want to work on, and disable their display by pressing Shift+3 or by using the **Toggle Display** tool in the **Hierarchy** toolbox.

Exercise: Separating a Database

This exercise shows you how to separate a database using different commands in the BSP menu. To begin the exercise:

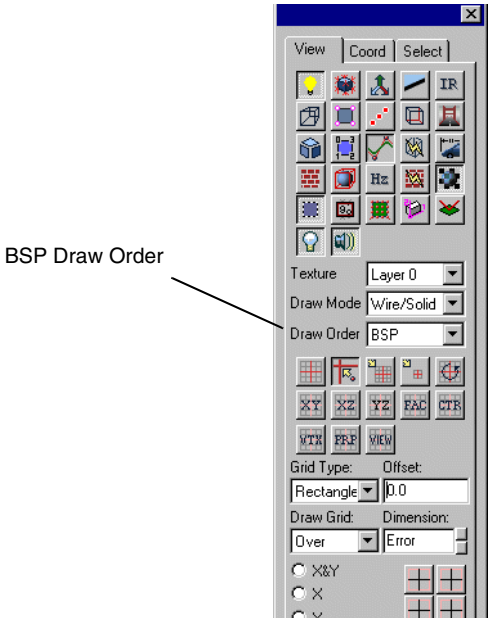
Step 1. Open `b747_400.flt` in the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory.

Note: If you did not install the exercise files when you installed Creator, you can open this file directly from the Creator installation CD.



Step 2. Click the **View Panel** button in the Toolbar to open the View panel.

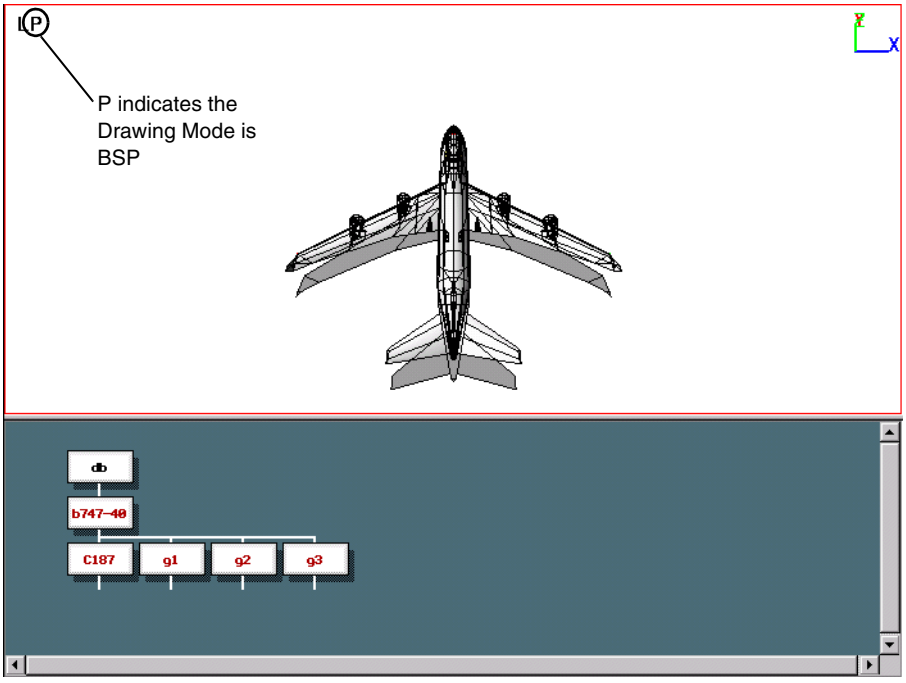
Step 3. Choose **BSP Draw Order**.



Step 4. Click the **Grid On/Off** button to turn off the grid (this makes the BSPs easier to see) and then close the View panel.



Step 5. Split the window into Graphics and Hierarchy views. The image on your screen should look similar to the following figure.



Note: You can use the **Browse** buttons in the Help window to move from one part of the exercise to the next.

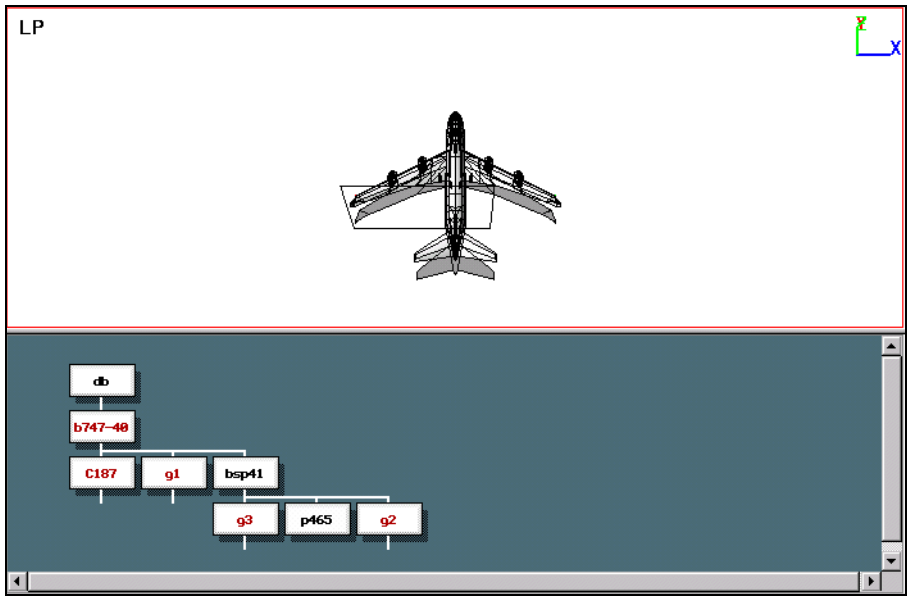


Using Separate Selected

Step 1. In the Hierarchy view, select the group nodes **g2** and **g3**. These contain the tail and the left half of the airplane

Step 2. Make sure you are in Group mode, and then choose **BSP/Separate Selected**. Creator inserts a BSP node above both groups in the hierarchy, and inserts

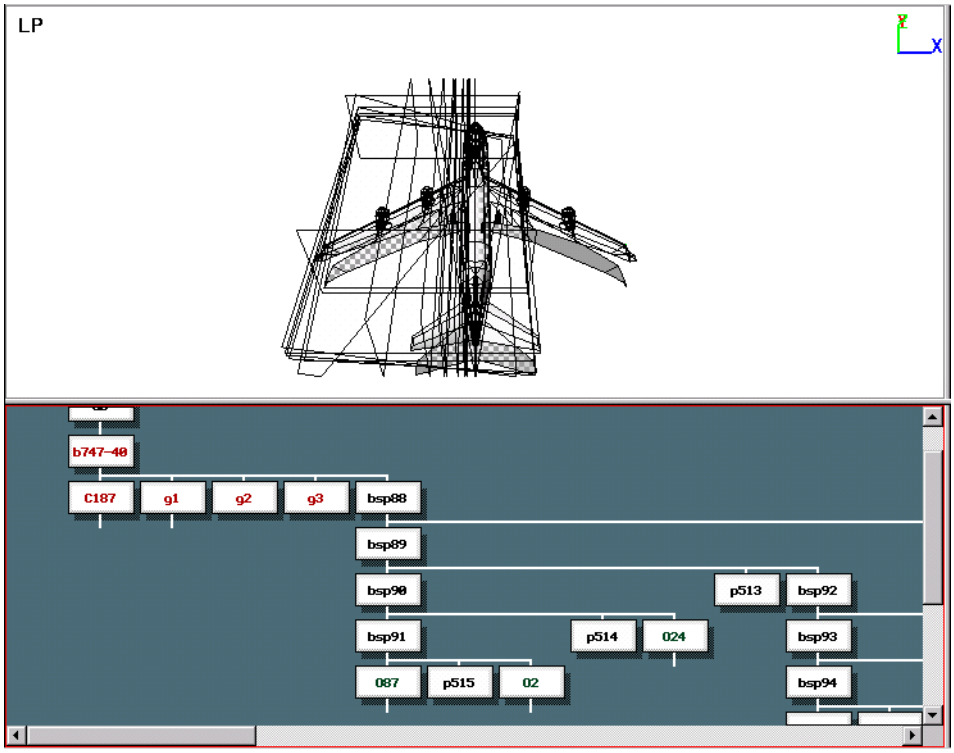
a plane between the two groups. You can change the eyepoint with the right mouse get a better view of the BSP placement.



Step 3. Select **Edit/Undo** to remove the plane.

Step 4. Select group nodes **g2** and **g3** again.

Step 5. Select Object modeling mode from the list on the Toolbar, and then choose **BSP/Separate Selected**. Notice that you get different results.



Instead of separating the groups, Creator separates only the objects below the group nodes, because the separation was performed in Object mode. Notice that a BSP node (**bsp521**) is inserted at the same level as the group. The objects that previously were children of **g2** and **g3** are moved below **bsp521**, and are then separated from each other.

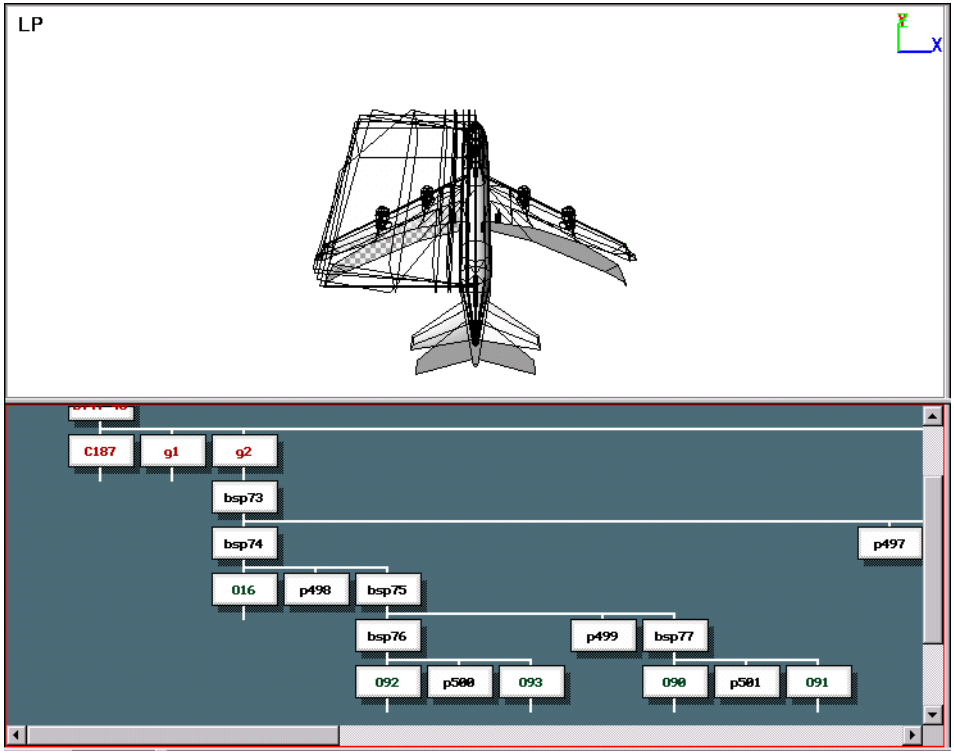
To preserve the tree structure as much as possible when using **Separated Selected**, set the modeling mode to match your top level selection. If you select group nodes, for example, make sure you are in Group mode; if you select objects, make sure you are in Object mode, and so forth.

Comparing the Effect of Mode on Separate Selected

Step 1. Select **Edit/Undo** to clear the BSPs. Hold down the Alt key and middle-click **g2** or **g3** to collapse the Hierarchy structure back to the original four group nodes.

Step 2. Select **g2**.

Step 3. Make sure you are in Object mode, and then choose **BSP/Separate Selected**. Notice that the *objects* under the selected group are separated.



Step 4. Select **Edit/Undo**.

Step 5. Select **g3**.

Step 6. Change to Group mode, and then choose **BSP/Separate Selected**. Notice that this time the status line displays the error message "Less than 2 convex

hulls found-separation aborted." This is because Creator is looking for at least two groups to separate, and only one group (**g3**) is selected.

If **g3** had two groups as its children, you would still get this error message. When using **Separate Selected** Creator looks for at least two items to separate at the top level. If these are separable, BSPs are inserted, and separation moves to the next level. In this example, only one group is available at the top of the selection tree, so separation fails immediately, and the other levels are not evaluated for separation.

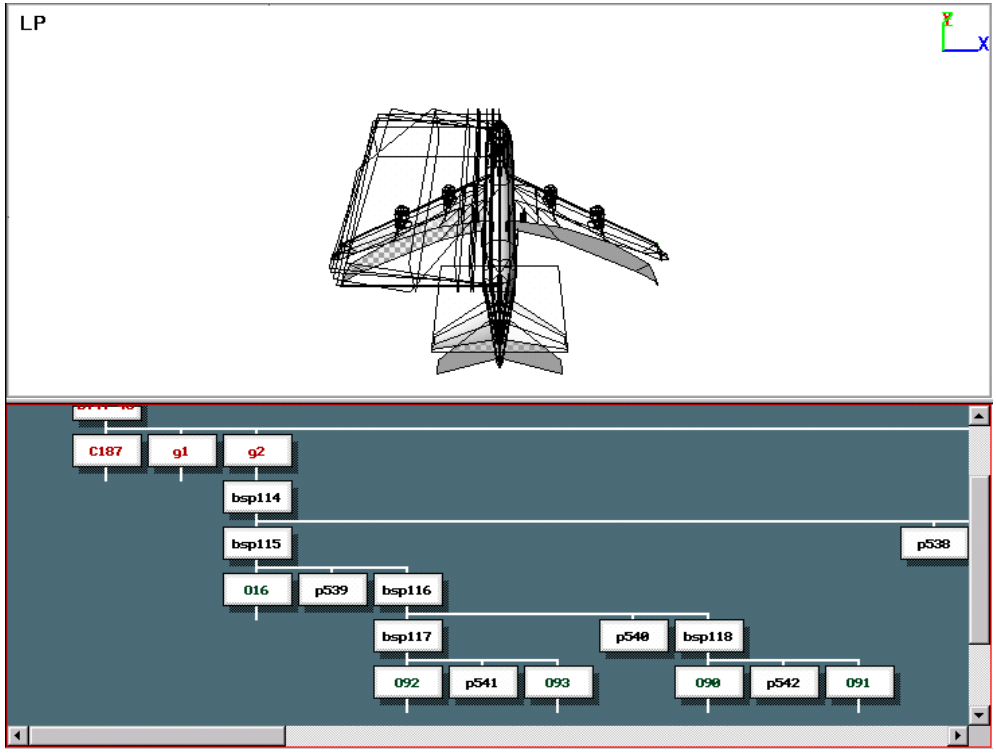
Using Separate Children

Step 1. Select **Edit/Undo** to clear the BSP planes you created with **Separate Selected**. Hold down the Alt key and middle-click **g2** or **g3** to collapse the Hierarchy display back to the original four group nodes.

Step 2. Select the group nodes **g2** and **g3** again.

Step 3. Select **Group** mode from the **Mode Selection** list on the Toolbar.

Step 4. Choose **BSP/Separate Children**. Creator inserts BSPs between all objects that are children of **g2** and **g3**.



Step 5. Use the slider under the Hierarchy view to see more of the structure.

Step 6. Notice that no BSP is inserted between the groups themselves. The children of **g2** and **g3** (the objects below the group nodes) are separated, but they remain under their original parent nodes.

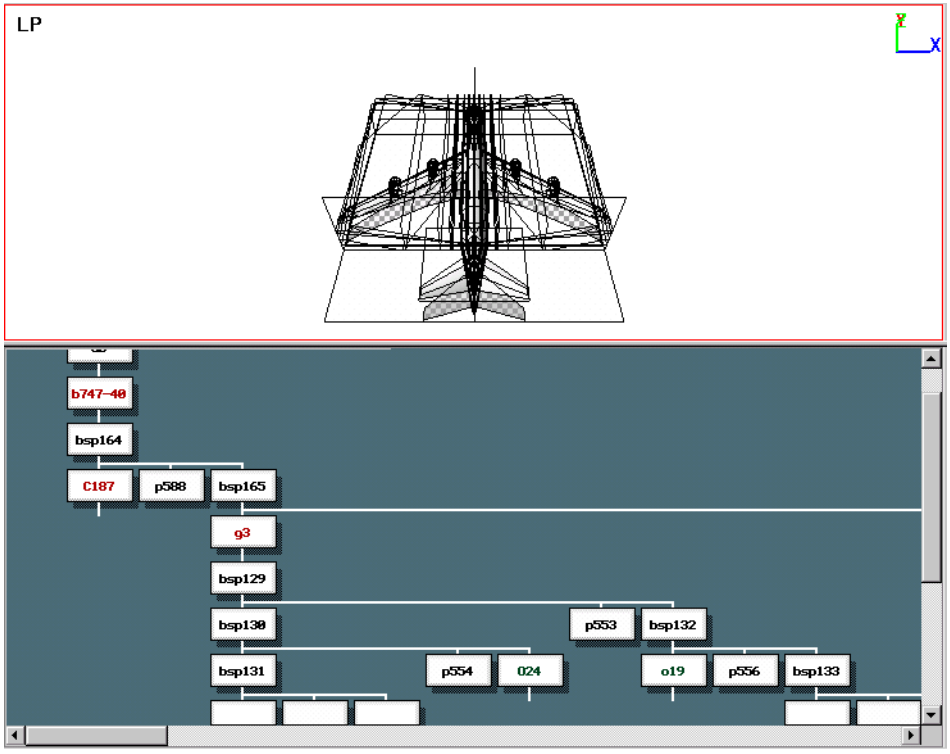
Separate Children is good to use when you want to preserve the original tree structure. It is also good for working on individual trees.

Using Recursive Separation

Step 1. Select **Edit/Undo** to remove the planes. Hold down the Alt key and middle-click **g2** or **g3** to collapse the Hierarchy display back to the original four group nodes.

Step 2. Make sure the modeling mode is set to Group.

Step 3. Select group node B747-400 and choose **BSP/Recursive Separation**. Notice that all groups and objects all the way down the hierarchy tree are separated by BSPs.



27 Basic Terrain Conversion

The Basic Terrain option converts real-world geographic data into a polygonal database, which you can use to create realistic virtual worlds. Basic Terrain uses the Delaunay method to interpret your geographic data and convert it to a Creator database that is made up entirely of triangular polygons. You control the process using these basic steps:

1. Convert the geographic data into DED format, using the conversion utilities provided with Creator, or a third-party conversion program such as Okino PolyTrans.
2. Import the DED file into the Creator *Terrain* window, and select a **Gaming Area**.
3. Set parameters that control the way the DED elevation data are converted to triangular polygons.
4. Set properties that control how contours are displayed.
5. Define LODs.
6. Generate the polygonal terrain and display it in the *Database* window.
7. Save the database.
8. Test the database in your realtime system, and make adjustments, if necessary.
9. You may need to re-import the DED file, adjust parameters in the *Terrain* window, and regenerate the terrain to optimize it for your realtime system.

Importing Elevation Data

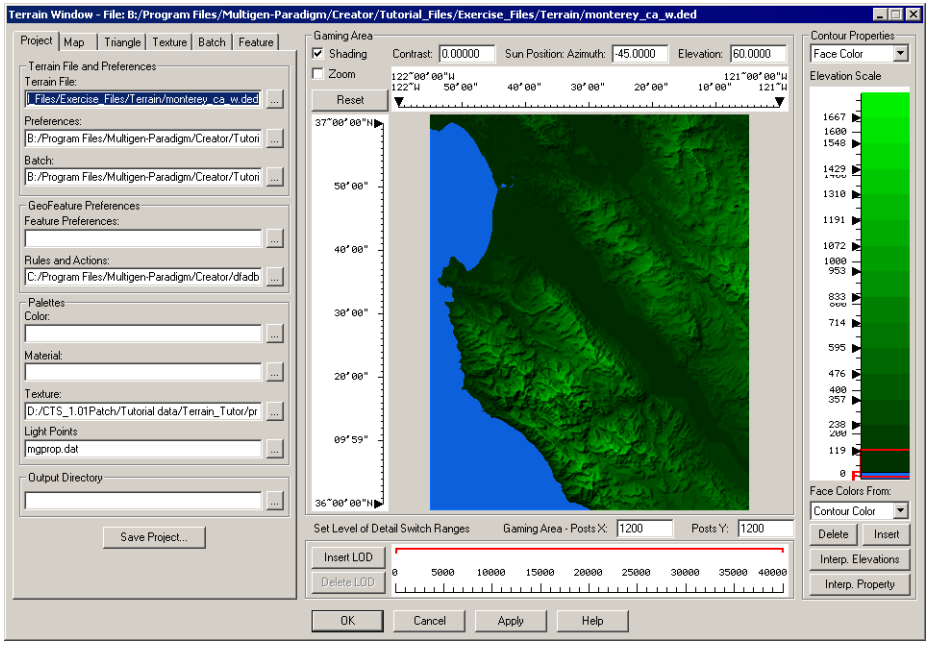
Elevation data comes in many different formats, such as USGS DEM and NIMA DTED. These original source data formats represent elevation maps for specific geographic regions, but with different format specifications. The Digital Elevation Data (DED) format is a uniform elevation map format that MultiGen-Paradigm developed for the Creator terrain generation tools to read. Because the terrain generation tools do not read source data in their original format, the original source data must first be converted to the DED format. (For additional information about converting files to DED format, see “Reading and Importing Elevation Data” on page 1063.)

DED files contain a matrix of elevation values and header information about the number of points in the latitudinal and longitudinal directions. DED files can describe any size area on the Earth. Utility programs are included with Creator to convert NIMA DTED and USGS DEM data into Digital Elevation Data (DED). The DED Builder in Creator uses the functionalities of these utility programs as one tool to convert elevation data into DED files. See"Reading and Importing Elevation Data" on page 1063 for instructions on how to use these utilities and the DED Builder.

Once your elevation data is in DED format, you can import the file into the Creator *Terrain* window. To do this:

Step 1. Select **Terrain/New Project**. A file-selection dialog box prompts you to select a DED file.

Step 2. Select the DED file you want to use. The *Terrain* window appears.



From this window you select all of the parameters required to generate a terrain database.

Selecting a Gaming Area

The *Terrain* window displays your imported elevation data in the **Gaming Area** as a map that you can use to select only the portion of the data you want. You do not need to use all of the data contained in the DED file you imported.

To select terrain data for processing, drag a fence around the portion in the **Gaming Area** that you want to include in your terrain database, or drag the arrows in the **Latitude** and **Longitude** scales to select the area. The selected area becomes the new **Gaming Area**.

To display an enlarged view of the selected area, set the **Zoom** checkbox. To re-select the entire file, click **Reset**.

Choosing Triangulation Parameters

The Basic Terrain option uses the Delaunay triangulation algorithm, which extract a subset of the elevation values in the DED file to convert into the vertices of the triangles that make up the final terrain database. **Triangulation Parameters** work together to give greater priority in the selection process to certain post values. The settings you choose for these parameters give you control over the triangulation process.

The following steps give you specific instructions on how to set triangulation parameters in Creator. For a more general discussion about how to choose the best parameter settings for your realtime system, see "Design Tips for Building Terrain" on page 956.

Step 1. Set the group size by choosing a **Group Size in Latitude Posts (X)** and **Group Size in Longitude Posts (Y**). These two setting determine how much area is incorporated into each terrain group. The two **Group Size** parameters should be equal, unless you have a specific reason for making them different.

The group size directly impacts LOD switching and the size of the bounding boxes used to determine what will be drawn to the screen and what will be culled by your realtime application (see "Balancing Culling and Drawing" on page 957).

Step 2. Set a **Z Scale Multiplier** if you want to exaggerate the difference between elevations to give a special effect (to make a mountainous terrain from data derived from foothills, for example). A **Z Scale Multiplier** of 1 preserves the original elevations.

Step 3. Set **Land Posts Color Threshold** to specify a minimum percentage of non-zero posts that must be present for the terrain color to be assigned to a polygon.

If the number of non-zero posts is less than this value, the Fixed Elevation color (usually blue) is assigned to the polygon. If the number of non-zero posts is greater than this value, the normal terrain color is assigned. This control is useful for substituting the Fixed Elevation color for polygons that are mostly water.

Step 4. Set **Disable Edge Matching** to disable edge matching through all levels of detail. Delaunay triangulation forces vertices on edges in levels of detail to match for smooth LOD switching. If **Disable Edge Matching** is set, edges can be matched by altitude priority, for example, instead of by vertices.

If **Disable Edge Matching** is cleared, vertices are not added in higher levels of detail if they are not on an edge in the lowest level of detail.

Step 5. Set **Quad Tree LOD's** to control the hierarchy of each square of terrain in different levels of detail. If **Quad Tree LOD's** is set, every higher resolution LOD has four groups that represent the area covered by a single group in the next lower resolution LOD.

If **Quad Tree LOD's** is cleared, each square area of terrain is represented by a single group in each LOD. In this case, the amount of resolution is determined by **Maximum Faces/Group**.

Step 6. Set **Delaunay Error Log** to create a text file listing the name of each database created in a batch with the Delaunay algorithm, followed by the maximum and average Delaunay error for each group in each LOD. The error log has the same name and directory path as the original DED file, plus the extension `.log`.

Step 7. Set **Calc. RMS Error** to calculate the Root Mean Square distance from the elevation post data to the polygons for each group and store that information in the group comments. RMS error is a measure of precision

between the generated polygons and source data. The smaller the RMS error, the more precise the match between the elevation post data and the polygons is.

Step 8. If your terrain area contains mountainous or hilly areas, you can enable **Ridge and Valley Detection** to give preference in the triangulation to elevation posts that fall along the high and low points of these areas. Leave the **Ridge and Valley Tolerance** at the default setting (see "Ridge and Valley Detection and Tolerance" on page 945).

Unless you want to emphasize dramatic changes in elevation, it is better to clear **Ridge and Valley Detection** and set **Use Post Weighting Multipliers** instead.

Step 9. Enable **Use Post Weighting Multipliers** to give greater weight to posts that vary the most from the average elevations in a particular area. Leave the **Terrain Accuracy Tolerance** at the default setting (see "Use Post Weighting Multipliers and Terrain Accuracy Tolerance" on page 946).

If you set **Use Post Weighting Multipliers**, disable **Ridge and Valley Detection**. The calculations for **Ridge and Valley Detection** usually create enough polygons to reach the **Maximum Faces Per Group**, so **Use Post Weighting Multipliers** has no effect.

Step 10. Set **Preserve Coastline** if you want to give preferential treatment to elevation posts that fall on a coastline.

When **Preserve Coastline** is enabled, Creator tests the edge of each new polygon for intersection with the coastline and adds coastal posts to the triangulation (see "Preserve Coastline" on page 945).

If your **Gaming Area** includes a coastline, you should always enable this parameter to ensure that the coastline is represented with some accuracy. Selecting this parameter causes other areas in the terrain to be represented by fewer polygons, however; you may need to increase the **Maximum Faces Per Group** to get acceptable results for all areas.

Step 11. Select an **LOD Number** for setup. The displayed values of **Maximum Faces/ Group** and **Terrain Accuracy Tolerance** apply only to the selected LOD.

Triangulation Parameters

The following parameters control the Delaunay triangulation process:

Group Size in Latitude Posts (X) and Group Size in Longitude Posts (Y)

Define the height and width of each rectangular terrain group in the highest LOD. For example, if **Group Size in Latitude Posts (X)** = 8 and **Group Size in Longitude Posts (Y)** = 8, each group in the highest LOD corresponds to an 8 x 8 area of posts.

The group size directly affects the culling process your realtime system uses to determine what needs to be drawn. If the group size is too large, no culling will be done, and the realtime system will spend all its time drawing, decreasing performance. If the group size is too small, too many culling groups may be generated for the realtime system to process, and performance also will be adversely affected. You may need to generate the terrain using different group sizes and then test the results on your realtime system to determine the optimal group size (see "Balancing Culling and Drawing" on page 957).

Small group sizes can result in many more polygons than large groups because each group's corner posts are included in the triangulation, and because the limit imposed by **Maximum Faces Per Group** applies equally to all groups, regardless of their physical size. When you decrease the group size, more groups are needed to cover the terrain area.

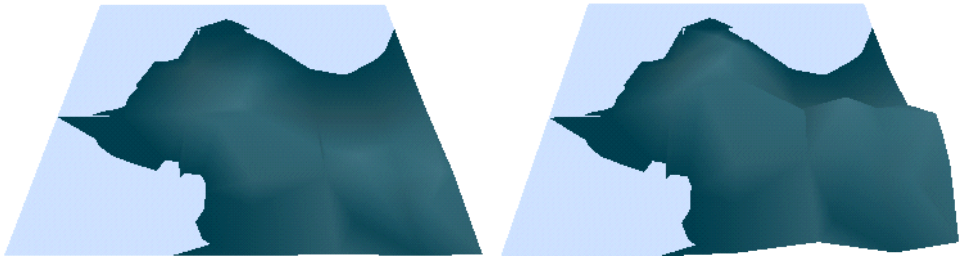
If only one level of detail is being created, the area is not chopped into rectangular groups. In this case, polygons are sorted into groups based on the center of each polygon, which reduces the resulting number of polygons.

Maximum Faces Per Group

Defines the maximum number of polygons in each group for every level of detail. This value sets an upper limit on the number of triangular polygons generated by the ridge and valley detection, coastline detection, and error correction algorithms.

Z Scale Multiplier

Multiplies each post value as it is converted to a database coordinate. A **Z Scale Multiplier** of 10, for example, changes the *z* component of each vertex to ten times its original elevation, exaggerating the elevation differences in the original terrain.

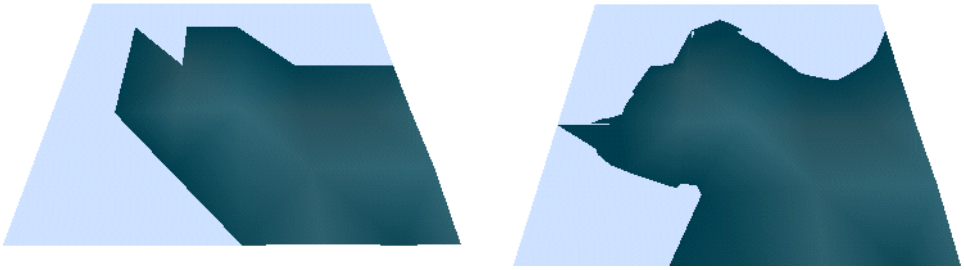


Z-scale Factor = 1

Z-scale Factor = 10

Preserve Coastline

Gives greater priority to coastline elevation posts during the triangulation process. A coastline post is one at zero elevation (sea level) that is adjacent to posts that are not at zero elevation.



Terrain generated with no correction factors

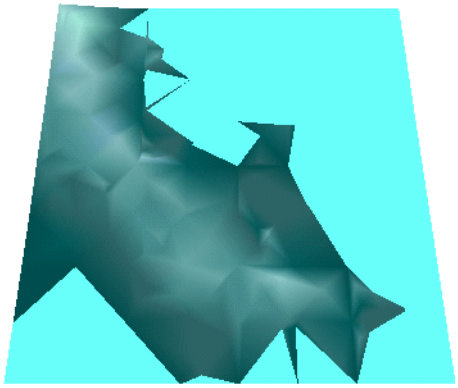
The same terrain generated with Preserve Coastline

Ridge and Valley Detection and Tolerance

Compares and analyzes differences in elevation between adjacent posts to determine which posts belong to ridges and valleys, and then adds sets of posts to the triangulation that follow the ridges and valleys.

The elevations of unused posts are then compared to the elevations of posts that are already included in the triangulation. Posts along the longest edges of existing triangles are considered first, and unused posts on the group boundary have higher priority than those on interior edges. Posts are added as vertices that create new terrain polygons until the difference between the elevation of each unused post and the average elevation of its neighbors is less than the **Ridge and Valley Tolerance** limit you

set, or until the **Maximum Faces per Group** is reached. The default **Ridge and Valley Tolerance** is the best setting for most terrain data.



Terrain generated with
Ridge and Valley Detection



Terrain generated with
Ridge and Valley Detection
and Preserve Coastline

Use **Ridge and Valley Detection** when you have terrain with mountainous or hilly terrain areas that are areas of special interest (areas where you expect to place detailed cultural features, for example). Areas with less dramatic changes in elevation receive a smaller allotment of polygons, so these areas are rendered with less detail.

To force a more balanced evaluation of terrain data, clear **Ridge and Valley Detection** and set **Use Post Weighting Multipliers**.

Use Post Weighting Multipliers and Terrain Accuracy Tolerance

Use Post Weighting Multipliers, computes the distance from each post that is not included in the triangulation to the edge of the corresponding polygon, and adds the post with the greatest distance to the triangulation.

The analysis and addition of triangular polygons is repeated until the average distance between each unused post and the corresponding polygon is less than the **Terrain Accuracy Tolerance** or until Creator reaches the limit of **Maximum Faces Per Group**.



Terrain generated with Error Correction



Terrain generated with Error Correction and Preserve Coastline

If you set **Use Post Weighting Multipliers**, clear **Ridge and Valley Detection**. Creator calculates **Ridge and Valley Detection** first, which often creates enough polygons to meet the **Maximum Faces per Group**.

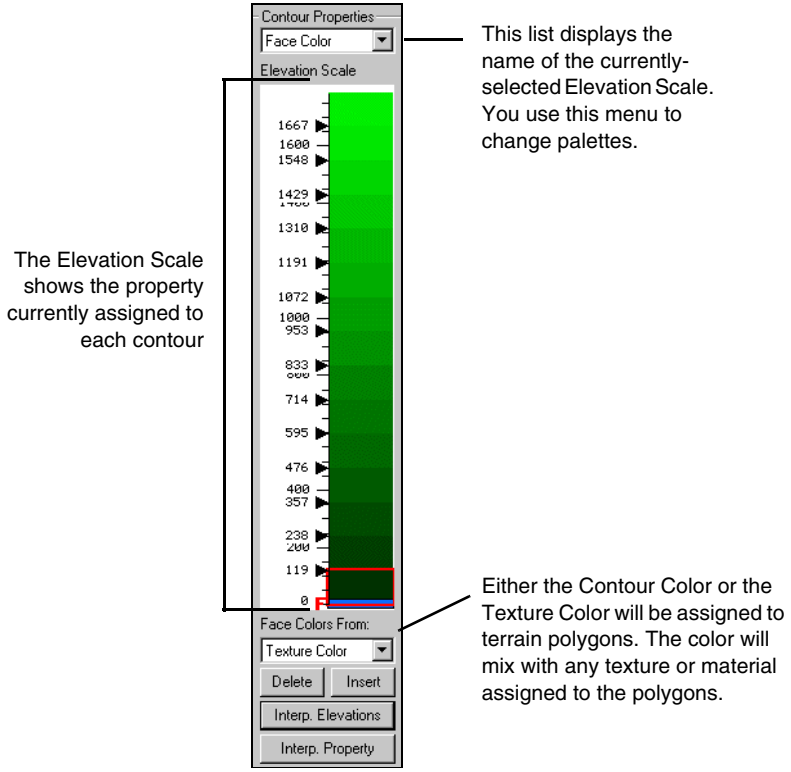
Changing Contours and Their Properties

A *contour line* is a line on a map that joins points of equal elevation. A *contour* is an area between two contour lines that represents a range of elevations. When you import elevation data, the map in the **Gaming Area** displays color-coded contours calculated from the data. You can change the range of elevations represented by any contour, add or delete contours, and change **Contour Properties**.

Assigning Properties to Contours

Contour Properties determine the contour or texture color, texture, and material assigned to polygons in the new terrain database. As each polygon is generated, its color, texture, and material are taken from the contour that corresponds to the polygon's median elevation.

You can assign a contour color, texture color, texture pattern, and material to any or all contours using the Elevation Scale in the *Terrain* window.



The properties you assign have the following priorities:

- The **Face Colors From** setting chooses whether the **Face Color** shown in the Elevation Scale or the **Texture Color** is applied to the polygons when the terrain database is generated. If you choose **Contour Color**, the **Texture Color** is ignored; if you choose **Texture Color**, the **Face Color** setting is ignored.
- If you assign a texture or material to a contour, the face or texture color mixes with the material or texture pattern. One way to avoid this is to assign a **Texture Color** of white to every contour and select **Face Colors From: Texture Color**. The default assignment for **Texture Color** is white.
- If you set **Indirect Texture** in the **Texture** panel, the terrain is textured according to the pattern of a color-keyed image and its corresponding text file. In this case, all other **Contour Properties** are ignored (see “Using Indirect Texture” on page 1185).

To Assign Properties to a Contour

Step 1. From the pull-down menu, select the Elevation Scale (**Face Color**, **Texture Color**, **Material**, or **Texture**) you want to change.

Step 2. Double-click a contour to open the corresponding Color, Texture Color, Texture, or Material palette.

Step 3. Select a color, texture color, texture, or material. The item you select in the palette is immediately assigned to the selected contour.

Step 4. To change another contour, select it, and then click another item in the palette. Repeat this step for every contour you want to change, or use interpolation to fill in the other entries in the palette.

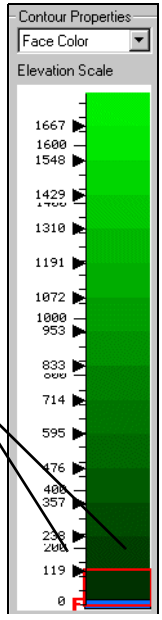
Note: You do not have to make assignments to every contour. If a texture or material entry is blank, a value of -1 (no texture/material) is assigned to faces in the corresponding range of elevations. Default colors and texture colors appear in the **Contour Color** and **Texture Color** Elevation Scales.

Adjusting Contours

You can set up as many (or as few) terrain contours as you like. To change the contour, drag it or click it to edit its value. Note that this changes the properties assigned to the range of elevations represented by the contour; it does *not* change the elevation data itself.

To Select and Resize a Contour

To select a contour, click it. A red outline appears around the contour. To change the size of a contour, you can drag the upper or lower boundary to the desired elevation or click it and edit its value. The other contours resize to accommodate the change.



A red rectangle highlights the selected contour. Drag the arrow or line to change the upper or lower boundary of the contour.

To Add a Contour

To add a contour, select a contour and click **Insert**. The selected band splits in half. If no contour is selected, the lowest one splits in half.

To Delete a Contour

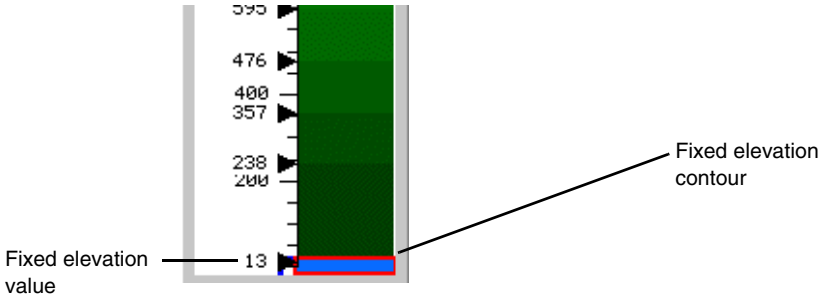
To delete a contour, click it, then click **Delete**. If no contour is selected, the lowest one is deleted. The elevations represented by the deleted contour are added to the next highest contour.

To Create Uniform Contours

To create contours of equal size, click **Interp. Elevation**. Existing contours are resized until each spans an equal range of elevations. If you find that you have too many or too few bands, use **Insert** or **Delete** to add or eliminate selected contours, and then click **Interp. Elevation** to redistribute the elevations assigned to each contour.

Changing the Fixed Elevation Contour

The *Fixed Elevation Contour* (labeled **F** in the display) assigns a color, texture, and material to one specific elevation. This is typically used to make sea-level polygons blue, but you can change the fixed elevation by dragging the **F** up or down or by editing the **Fixed Elevation** value. Assignments made to the Fixed Elevation Contour are not affected by **Interp. Elevation** or **Interp. Property**.



Interpolating a Palette

Interpolation fills in the current Elevation Scale using the contents of the lowest and highest contours. To interpolate a palette:

- In the **Contour Color** or **Texture Color** Elevation Scale, assign colors to the lowest and highest contours, and then click **Interp. Property** to fill all remaining contours with a smooth gradient of these two colors.
- In the **Texture** or **Material** Elevation Scale, assign a texture pattern or material to the lowest contour, and then click **Interp. Property** to copy this texture or material to all other contours. Texture patterns or materials assigned to higher contours are removed from the Elevation Scale (but remain in the Texture or Material palette).

Creating Terrain LODs

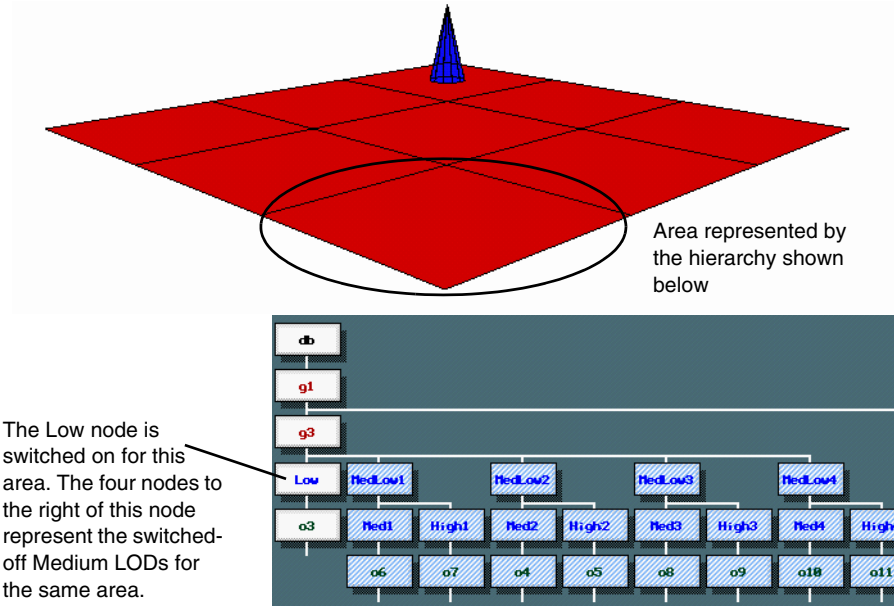
LODs (Levels of Detail) display the terrain at different levels of complexity. The highest LOD is displayed closest to the eyepoint; it contains the greatest number of polygons. Lower LODs, with lower polygon counts, may be displayed for areas farther away from the eyepoint.

Creator automatically creates quad tree LODs for Basic Terrain elevation data when you add LODs in the *Terrain* window. You can add, delete, or change the ranges for these LODs; the quad tree LOD calculations control the number of polygons in each LOD.

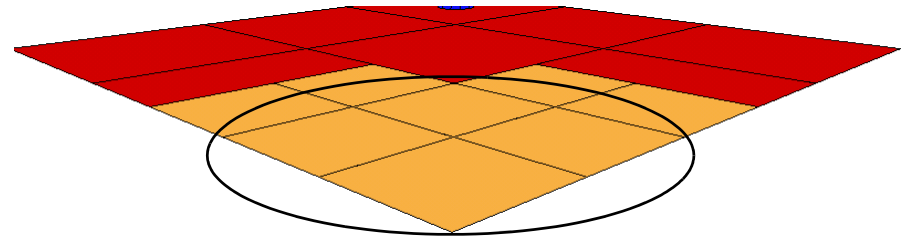
About Quad Tree LODs

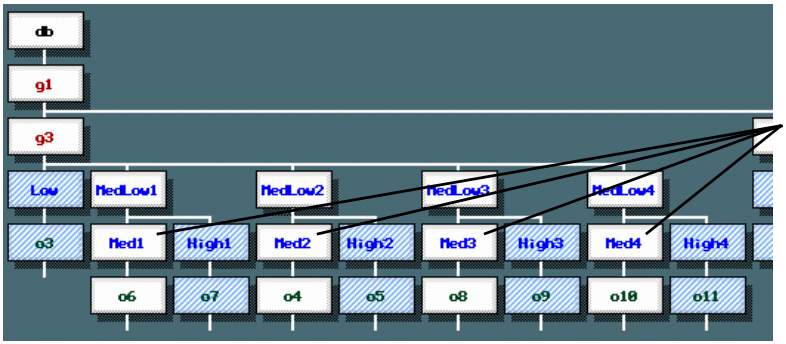
The quad tree method of calculating LODs for Basic Terrain uses the group size defined by **Group Size in Latitude Posts (X)** and **Group Size in Longitude Posts (Y)** to calculate the number of groups in the highest LOD. The number of polygons in each group of the highest LOD is controlled by **Maximum Faces per Group**. In the second-highest LOD, four of the high LOD groups are represented by a single group, so the resolution (and the number of polygons in the new group) is one-fourth that of the highest LOD. In the third-highest LOD, four groups from the second-highest LOD are represented by a single group, and the resolution is one-fourth of the next highest LOD, and one-sixteenth of the highest LOD (so the number of polygons per group is one-sixteenth that of the highest LOD). Additional LODs are calculated in the same way.

In this example, the terrain area is represented by nine low-resolution groups.



Moving closer to the object, each low-resolution group in the foreground switches to four medium-resolution groups.

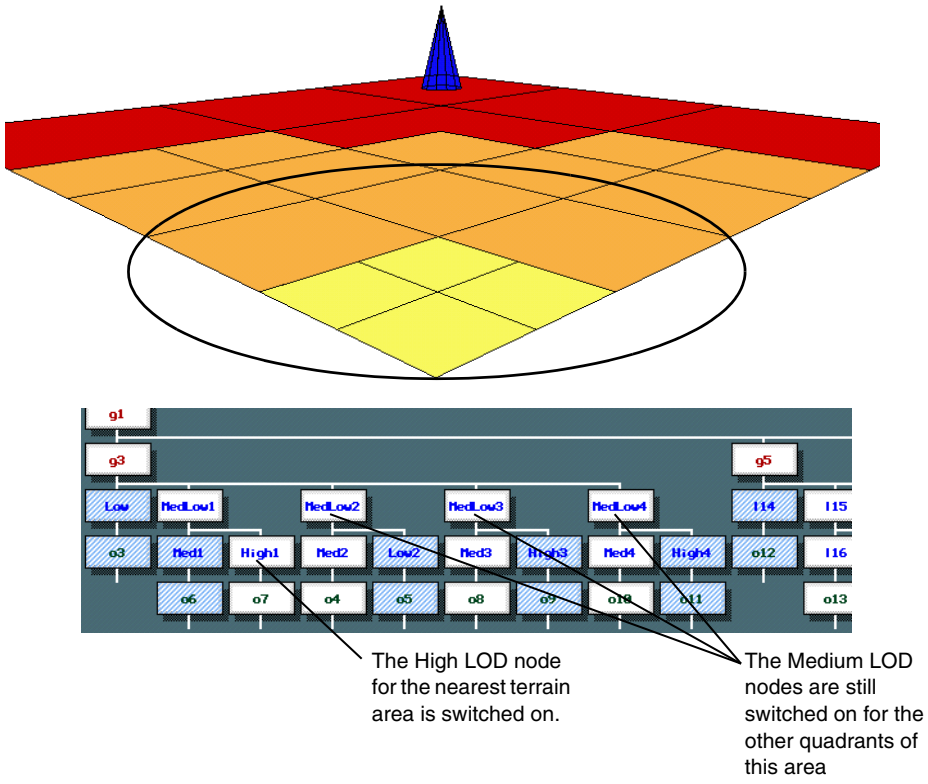




The Medium LOD nodes are switched on

High LOD nodes for this area appear to the right of each Medium LOD node

Moving even closer, the nearest medium-resolution group is replaced by four high-resolution groups.



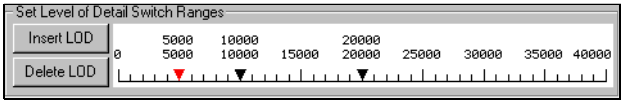
Setting LOD Switch Ranges

You create LODs by setting switch ranges. A *switch range* is the range of distances from the eyepoint at which a particular LOD is displayed. The boundaries of the switch range are the *switch-in* and *switch-out* distances (see "Levels of Detail" on page 707).

To Set Switch Ranges

You set the switch range for each level of detail using the LOD scale in the *Terrain* window. In this scale, each interval between triangular sliders (or between a slider and the end of the scale) corresponds to one LOD, with the lowest LOD on the right.

To set a switch-in distance, drag a slider to the desired setting on the LOD scale, or click it and edit its value. The active slider appears in red.



You should set the switch-in distance of the lowest LOD so that it falls beyond the boundary of the terrain; otherwise the terrain will appear to pop in and out of the horizon. By default, the lowest LOD switches in at 10,000,000; the highest LOD switches out at 0. Dragging a slider past the right end of the scale increases the range of available switching distances displayed in the control.

The switch-out setting for the LOD is automatically set when you choose a switch-in setting for the next highest LOD. The highest LOD is displayed right up to the eyepoint (0 on the scale).

To Add a Level of Detail

To add an LOD, click **Insert LOD**. If a slider is selected when you click this button, the LOD on its left is split in two. If nothing is selected, the highest LOD is split in two.

To Delete a Level of Detail

To delete an LOD, select a slider and click **Delete LOD**. If nothing is selected when you click this button, the highest LOD is deleted.

Design Tips for Building Terrain

The same DED file can produce many different terrain databases, depending on the settings you choose in the *Terrain* window. Many times, the best results can be achieved only by generating a series of trial databases, and then running them on the realtime system. The information in this section can help narrow the scope of your trial data.

- *Know your hardware.*

You must know the number of polygons, textures, light sources, and so forth, that can be supported by the target hardware platform running the realtime simulation.

- *Know your realtime/rendering software.*

  How much time, in milliseconds, is required for applications processing, collision detection, culling, and rendering. *Applications processing* includes calculating the x,y,z and yaw, pitch, roll of the eyepoint, and the flight/driving model acceleration. *Collision detection* calculates when the bounding volumes of models intersect, indicating that a moving model has collided with a stationary object or terrain polygon, or with another moving model. *Culling* is the process of determining what needs to be drawn, based on the eyepoint and viewing frustum. *Rendering* involves displaying the results to the screen. Can your realtime system render, at 60 Hz, the number of textured anti-aliased polygons you have specified? If not, you need to simplify your terrain.

- *Determine your scenario requirements.*

  An airplane flying at 40,000 feet has a very large frustum, but the terrain can be less accurate than when viewed closer to the ground.

  A helicopter flying at 2000 feet has a smaller viewing frustum, but the terrain will be more visible, so it must be more accurate, and culture elements, such as tree tops, must be visible.

  A vehicle on the ground has a much smaller viewing frustum than either of the previous two cases (you cannot see as far from the interior of a car than from an airplane or helicopter, for example), but the terrain and culture that is visible from the car must be very accurate.

Balancing Culling and Drawing

Your realtime system should spend approximately the same amount of time culling the data to be viewed as it does to render the result to the screen.

When switching LODs, the bigger the group size, the bigger the terrain area that changes. When the group size is too large, however, the result is LOD "popping."

Most realtime systems give statistics for culling, drawing, application processing, and collision detection. Once you create the terrain and models, you should do simulations on your realtime system and make adjustments to the terrain's group size based on the results displayed by the statistics. In Creator, this means adjusting the Triangulation

Parameters **Group Size in Latitude Posts (X)** and **Group Size in Longitude Posts (Y)** in the *Terrain* window.

When your realtime application is determining what should be culled and what should be drawn to the screen, an entire group is either culled or drawn. Small groups result in longer culling time (more groups must be evaluated to decide what must be drawn or discarded); large groups result in longer drawing time (if even a small area of the group is within the field of view, the entire group must be drawn).

- If the system spends too much time culling, increase the culling (group) size.
- If the system spends too much time drawing, decrease the culling (group) size.

Texture Considerations

When you add texture, you must check the texture memory and pixel fill rate of the realtime system. The *pixel fill rate* is the number of textured, anti-aliased pixels per second that can be supported by your hardware. (This statistic should be listed in the hardware specification for your system.) *Texture memory* is the amount of memory available for loading and storing textures. In some systems, this memory is specialized; in other systems, texture is loaded into the general system memory. You must take into account not only the number of textured polygons, but also the memory used by mipmaps, if present. (*Mipmaps* are a series of low-resolution versions of the base image.)

Choosing LOD Ranges

In order to make your scene realistic when moving through it, accurate switching distances are important. You must also consider the amount of terrain that will be switched in as the eyepoint moves, whether there are cultural features on the terrain, and how much detail your users need to see at each level of detail. Total polygon count is also an important factor when planning LODs.

If you do not have definite ideas about how to manage LODs, you should start by dividing the *viewing volume* (maximum viewing distance) into thirds to produce a high, medium, and low LOD. The high LOD is visible in the nearest third of the viewing volume; the medium LOD is visible in the second third of the viewing volume and the low LOD covers the last third of the viewing volume.

A vehicle on the ground might have a viewing volume of 6000 meters, for example, so the high LOD would switch in at 2000 meters, and the medium LOD would switch in at 4000 meters and switch out at 2000 meters. For an airplane with a viewing volume of 30,000 meters, the terrain would have high and medium switch-in distances of 10,000 and 20,000 meters, respectively.

The entire low LOD must switch in outside of the field of view to avoid "popping" (the sudden appearance of terrain or features) on the horizon (see "Understanding the Viewing Volume" on page 263).

You can use the following tips as guidelines when working with LODs.

- The total polygon count between the highest and lowest resolution LOD should not differ by more than approximately 30%.
- Set the initial switching distance to a range where the minimum dimension of the model is approximately 2 pixels.
- When figuring transition distances between LODs, the dimensional change in the geometry between the lower resolution LOD and the next higher one should be approximately two pixels.
- Use texture for lower resolutions of a feature, or for features that are not viewed closely, to save on polygons. For example, use a building facade texture representing a building in a lower resolution LOD that is modeled with polygons in a higher resolution LOD.
- Use textures to substitute for polygon detail. For example, a picket fence using an intensity alpha texture to simulate the spaces between pickets.

After you generate the terrain and models, test the database on your realtime system. If too much time is spent drawing, try making the switch range for the highest LOD shorter. When the switch range for the highest LOD is shorter, more terrain is displayed in the medium and low LODs, reducing the overall polygon count.

Make sure you test the worst case for your scenario. If you are flying an airplane, for example, the average polygon count takes sky areas into account; the sky generally has little detail, so it can be represented by a relatively small number of large polygons. If the plane if flying straight down towards the ground, the polygon count can be considerably higher than the average, because you are looking exclusively at terrain, and the terrain is close to the eyepoint.

An alternative to adjusting the high LOD switching distance is to decrease the overall number of polygons in the database.

Modify Delaunay

Modify Delaunay imports the post format file originally used to create a terrain database and lets you manually retriangulate the terrain by adding new points that form the vertices of new terrain polygons. It is especially useful for adding points to eliminate LOD "popping" or to enhance areas of high detail or interest.

When changes are made to a terrain database using **Modify Delaunay**, Creator compares the vertex coordinates of the new geometry with those of the old geometry. When all three vertices of a given face match, properties can be transferred from the existing face to the new one, or existing subfaces can be transferred.

If a three-vertex match is not found, a search is made for a two-vertex edge match, and the texture is regenerated based on the texture mapping of the original face. If there are no two- vertex or three-vertex matches, then the polygon is retextured as if it were created as new terrain using the last preferences for the DED file written to disk.

Exercise: Using Modify Delaunay

This exercise adds additional post points to enhance the detail of a valley and its surrounding ridges. The valley will contain an airstrip, and the enhancements ensure that the approach to the airstrip is clearly visible in all LODs. If you did not install the exercise files, you must copy the `modify_delaunay` example directory from the Creator CD-ROM to a directory on your hard drive, so that Creator can write the terrain preferences files into that directory (files cannot be written to a CD-ROM).

Getting Started

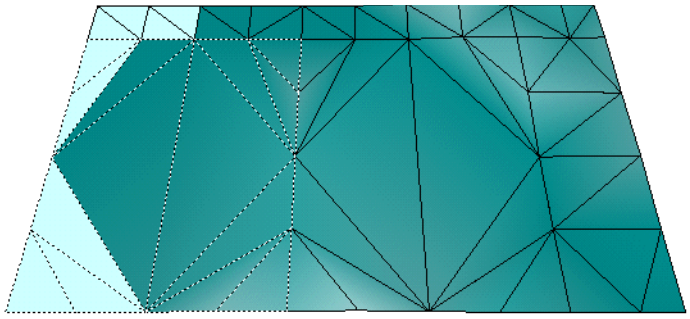
Step 1. Open `monterey.flt` in the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory.

Step 2. Set the modeling mode to **Group** (Shift-G). You must be in **Group** mode to use **Modify Delaunay**.

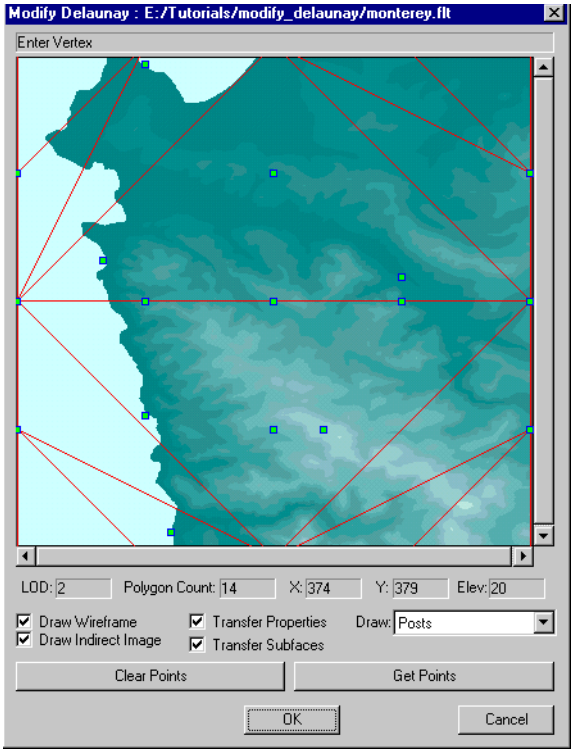
Add Points to the Lowest LOD

Step 1. Select **LOD/Least Detail**.

Step 2. Select the group where the airstrip will go.

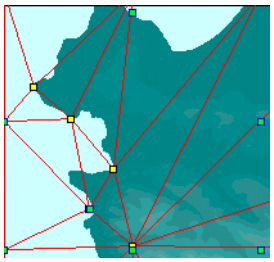


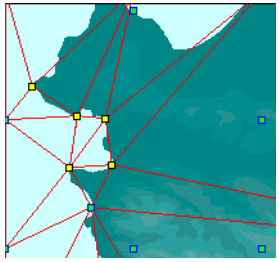
Step 3. Select **Terrain/Modify Delaunay**. The *Modify Delaunay* window appears, showing the original contours from the *Terrain* window. The lines show the edges of the current triangulation. The points displayed in cyan are the vertices from the next highest LOD. You can click these to add them to the low LOD (they are not added automatically), but in this exercise we will add new points to the low LOD and transfer them to the higher LODs.



Step 4. Click points along the valley, and the coastline at the mouth of the valley, to enhance the detail. Pay attention to the triangles you are creating; you do not want to place points such that a triangle covers an equal land and water area, for example.

Poor triangulation: because individual triangles contain a mix of land and water, the final terrain will have a poorly- defined coastline

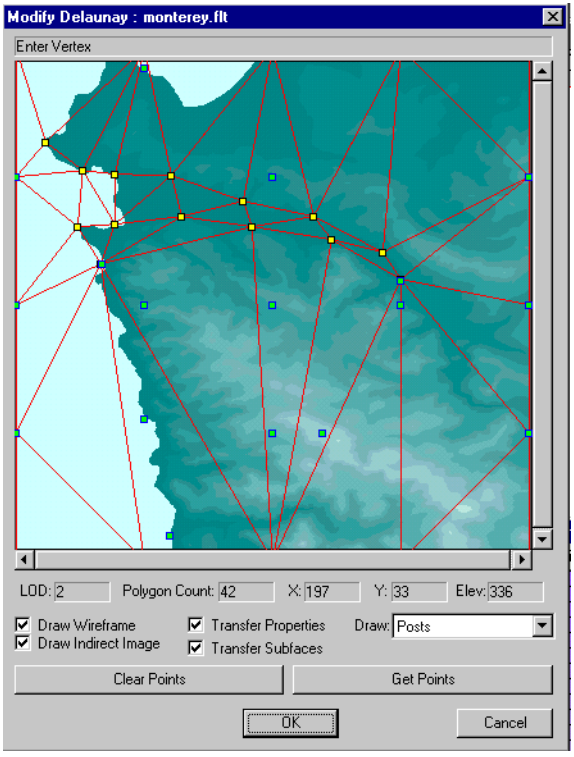




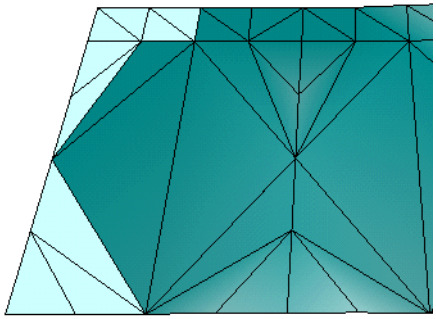
Good triangulation: each triangle is composed mostly of land or water, producing a better definition of the coast in this low LOD triangulation

If you do not like where you placed a point, right-click to eliminate it. Notice how the **Polygon Count** increases as you add points. The increased

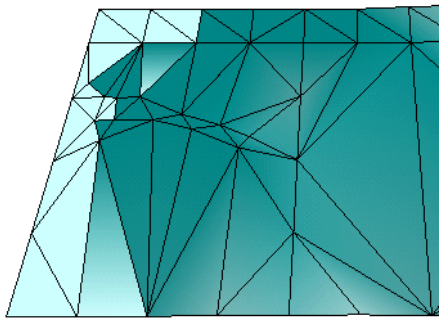
polygon count in each LOD may affect the performance of your realtime system.



Step 5. Click **OK**. The Graphics view shows the new triangulation of the low LOD.



Original low LOD triangulation: polygon count for selected group = 14

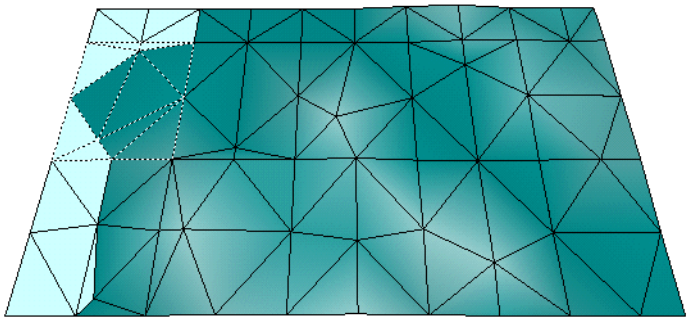


Modified low LOD triangulation: polygon count for selected group = 40

Add Points to the Medium LOD

Step 1. Select **LOD/More Detail** (Ctrl+M) to display the medium LOD.

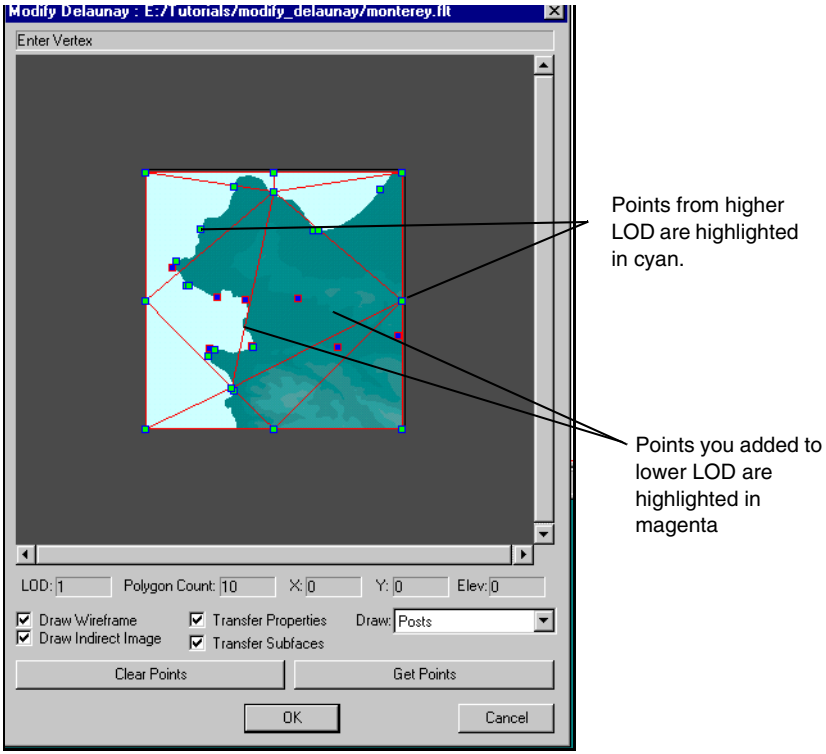
Step 2. Select the group that contains the valley.



Note: In the quad tree LODs generated by Creator, one low LOD group covers the same area as four groups of the next higher LOD (see "About Quad Tree LODs" on page 952). **Modify Delaunay** works on only one group at a time. If the features you want to enhance cover a large area, you may need to select and modify up to four different

groups in the medium LOD to enhance the area covered by the low LOD. In this exercise, we will modify only one group in each LOD.

Step 3. Select **Terrain/Modify Delaunay**. The contours of the medium LOD appear in the *Modify Delaunay* window. The points from the next highest LOD are highlighted in cyan. The points you added to the lower LOD are highlighted in magenta.



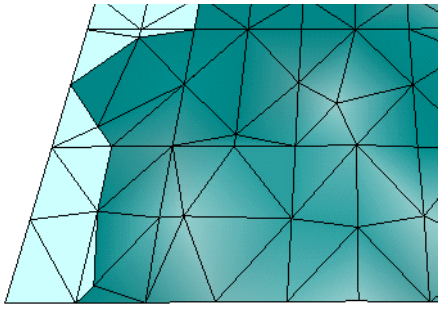
Step 4. Click **Get Points**. The *Input Value* dialog box appears.



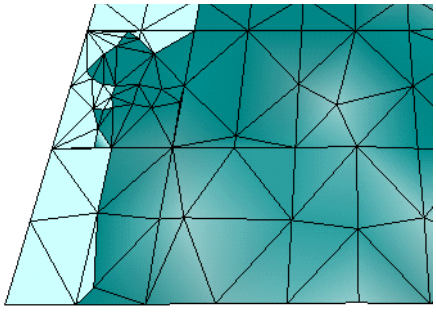
This dialog box prompts you to enter the level number of the LOD that is the source of the points. The highest LOD is level 0 (zero), and each lower-resolution LOD gets a higher number, in order. Because `monterey.flt` has three LODs, you would enter 0 for the highest LOD, 1 for the medium LOD, and 2 for the lowest LOD.

Step 5. Enter 2 and click **OK** to add the points you entered in the lowest LOD to the triangulation for the medium LOD. The points are no longer highlighted in magenta. These common points will ensure a smoother transition when the realtime system switches between LODs.

Step 6. Click the other locations on the map where you want to add points to the medium LOD (make sure you keep track of the polygon count), and then click **OK**.



Original triangulation for the medium LOD group: 10 polygons

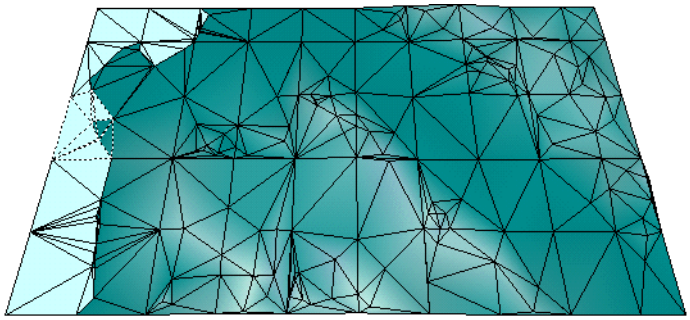


Modified triangulation for the medium LOD group: 50 polygons

Add Points to the Highest LOD

Step 1. Select **LOD/More Detail** (Ctrl+M) to switch to the next higher LOD, which in this case is the highest LOD.

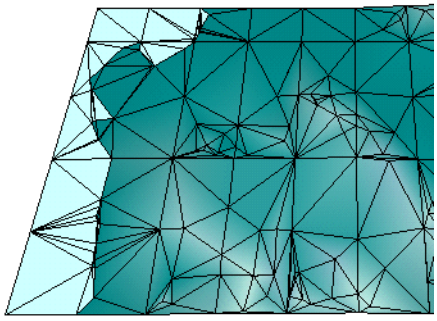
Step 2. Select the group that contains the valley, and then select **Terrain/Modify Delaunay**.



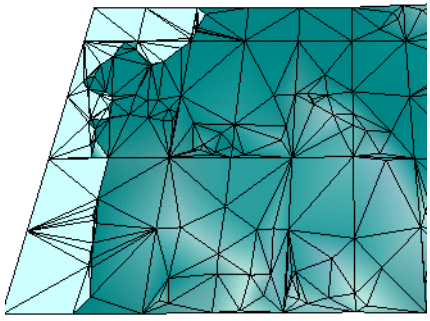
Step 3. Click **Get Points**. The *Input Values* dialog box appears.

Step 4. Enter 1 and click **OK** to add the points from the medium LOD to the triangulation for the high LOD.

Step 5. Click the map to add points to the high LOD, and then click **OK**.



Original triangulation for the high LOD group: 34 polygons



Modified triangulation for the high LOD group: 98 polygons

Note: Instead of selecting each LOD to which to add a vertex, you can add a vertex to all LODs at the same time by holding down the **Ctrl** button while you click the location of the vertex.

Modify Delaunay Requirements

To use **Modify Delaunay**, a terrain database must meet the following requirements:

- The database must have been triangulated using version 14.2 or later of the MultiGen terrain conversion option.
- The original gridded terrain post (DED) file must be available and must be located in the same directory path as the database.
- Post data stored in the header and group node **Comments** attributes must be intact. These attributes contain information that correlates each group with an LOD and an area of the DED file.

To Modify a Delaunay Triangulation

Step 1. Open a database created with Delaunay triangulation.

Step 2. Select a group to edit. The selected group cannot be part of an externally referenced file.

Step 3. Choose **Terrain/Modify Delaunay**. Creator reopens the original DED file and displays the selected group on top of gridded terrain post data in the *Modify Delaunay* window.

Step 4. Edit the vertices in the group. The group is automatically retriangulated as you add or delete each vertex.

- To *add* a vertex to the triangulation, click the left mouse button.
- To *delete* the vertex closest to the pointer, click the middle mouse button.

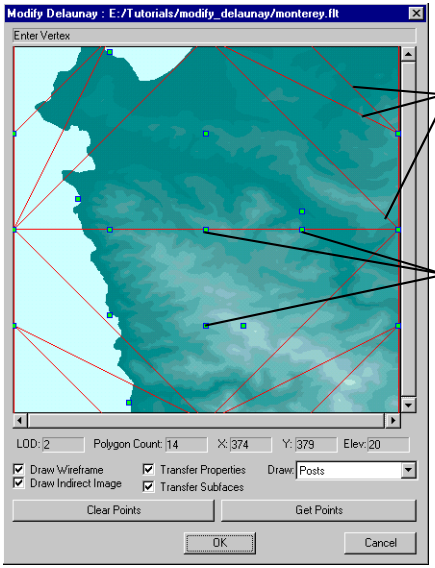
If you change a vertex along an outside (boundary) edge of a group, you should make the same change in the corresponding edges of the higher and lower resolution levels of detail, to preserve edge matching between levels of detail. However, you do not need to manually adjust group edges within the same level of detail; Creator automatically matches and re-triangulates adjacent groups within the same LOD.

The Modify Delaunay Window

The *Modify Delaunay* window displays the following information:

- The gridded terrain post area represented by the selected group.
- The triangulation (faces) of the group.
- The vertices of the next higher LOD, displayed in cyan. (This color may vary, depending on your system's color scheme.)
- The vertices of the next lower LOD, displayed in blue. (This color may vary, depending on your system's color scheme.)

  Because Delaunay triangulation includes the vertices of each group in the next higher LOD, vertices from lower resolution LODs are often hidden under the vertices displayed for higher resolution LODs.



Modify Delaunay Preferences

The following parameters appear at the bottom of the *Modify Delaunay* window.

LOD

Displays the LOD of the group you are editing. Level-of-detail (LOD) numbers increase as the amount of resolution decreases. LOD 0 is the highest resolution Level-of-detail, LOD 1 is the next highest, and so on.

Polygon Count

Displays the number of polygons in the current triangulation of the group.

X and Y

Displays the offset of the terrain post under the pointer, relative to the lower-left corner of the original DED file.

Elev

Displays the elevation of the terrain post under the pointer.

Draw Wireframe

Controls the display of wires (edges) around each face in the *Modify Delaunay* window.

Transfer Properties

Copies the color, texture, material, vertex *u* and *v* values, vertex colors, and vertex normals from the existing face to the new face.

Draw

Sets the polygon display mode in the *Modify Delaunay* window.

- **Posts** displays a contour map of the gridded terrain post data. In this mode, pressing the right mouse button temporarily displays a flat shaded (polygonal) view of the group.
- **Flat Shaded** displays an unshaded (polygonal) view of the group. In this mode, pressing the right mouse button temporarily displays a contour view of the terrain.
- **Gouraud Shaded** displays a Gouraud shaded view of the group. In this mode, pressing the right mouse button temporarily displays a contour view of the terrain.

Draw Indirect Image

Displays any indirect texture assigned to the group, rather than the contour display.

Transfer Subfaces

Transfers any existing subfaces to the new geometry.

Clear Points

Removes the interior vertices of this group. Vertices along the group boundary are not affected. The interior vertices of other levels of detail are not affected by this function, and remain visible in the window.

Get Points

Opens the *Input Value* dialog box, which prompts for the number of another LOD, and then copies the vertices from that LOD into the current LOD.

Exercise: Generating a Terrain Database

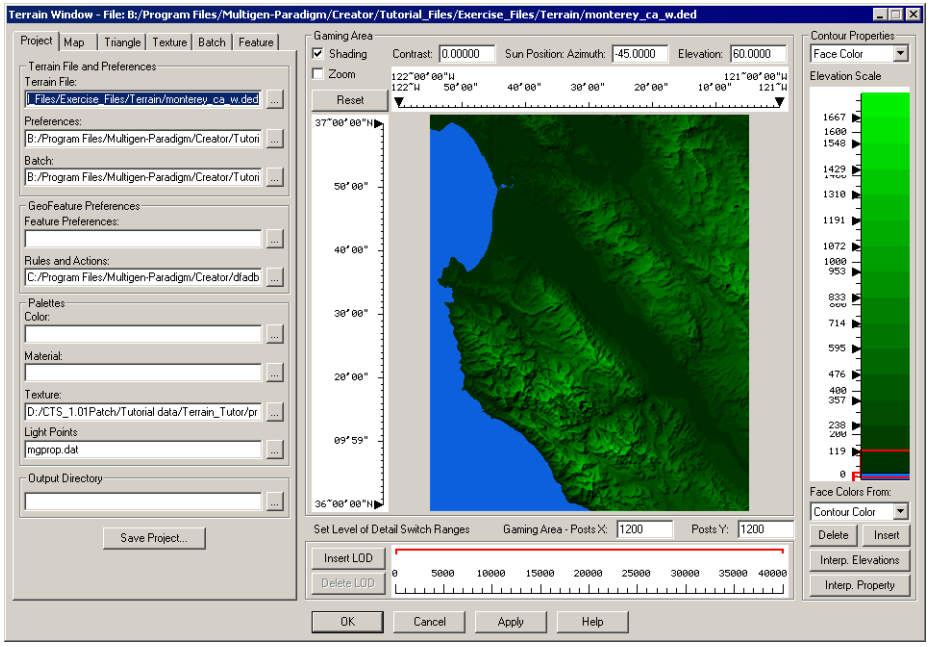
Import a DED File

Creator includes utilities for converting NIMA DTED and USGS DEM data into the Digital Elevation Data (DED) format understood by Creator. This exercise uses a DED file that already has been converted from USGS DEM format. You must copy this example file from the Creator CD-ROM to a directory on your hard drive, so that Creator can write the terrain preferences files to the same directory (files cannot be written to a CD-ROM).

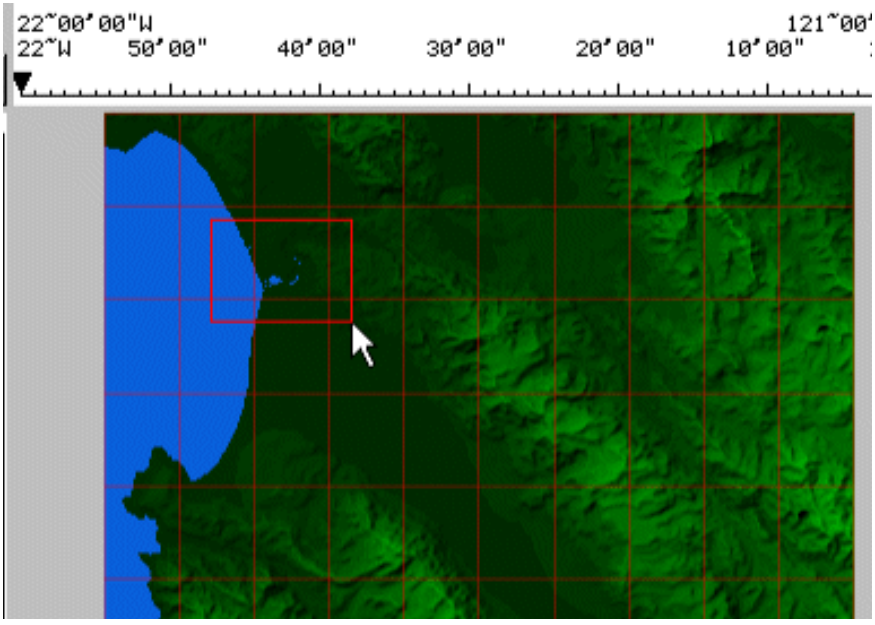
Step 1. Choose **File/Open** (Ctrl+O) to open a new, untitled Creator database.

Step 2. Choose **Terrain/New Project**. A file-selection dialog box appears, prompting you to select a DED file.

Step 3. Choose `monterey_ca_w.ded` from the `\MultiGen-Paradigm\resources\tutorials\creator\exercise_files` (Windows) or `/usr/local/MultiGen-Paradigm/resources/tutorials/creator/exercise_files` (IRIX) directory, and click **OK**. The *Terrain* window appears.



Step 4. In the **Gaming Area**, drag a fence around the semicircular lake area.



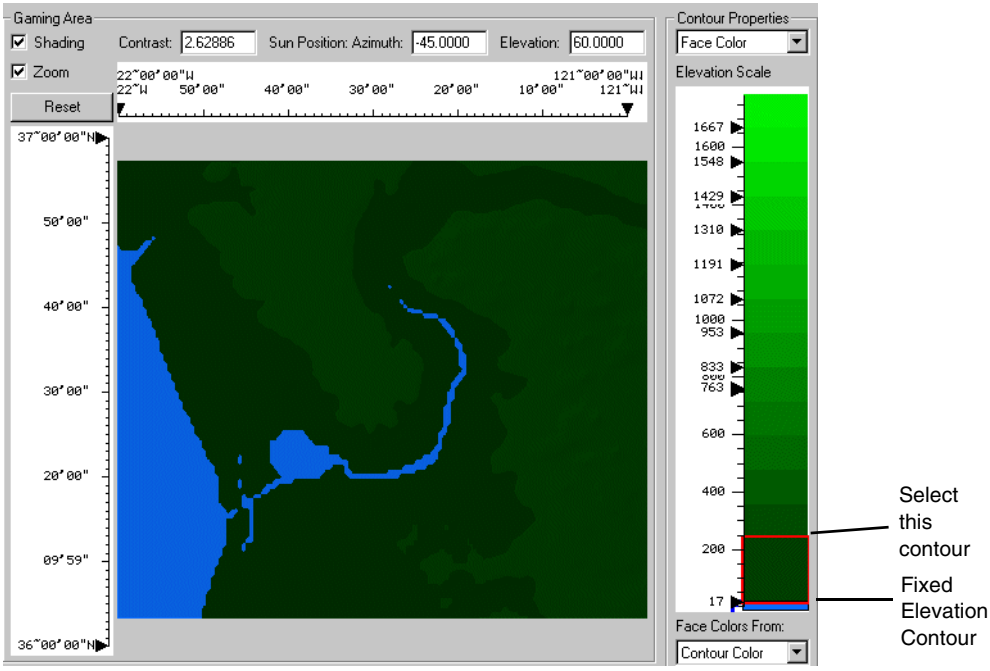
Step 5. Click **Zoom**. The **Gaming Area** zooms in on the selected area.



Change Contours and Properties

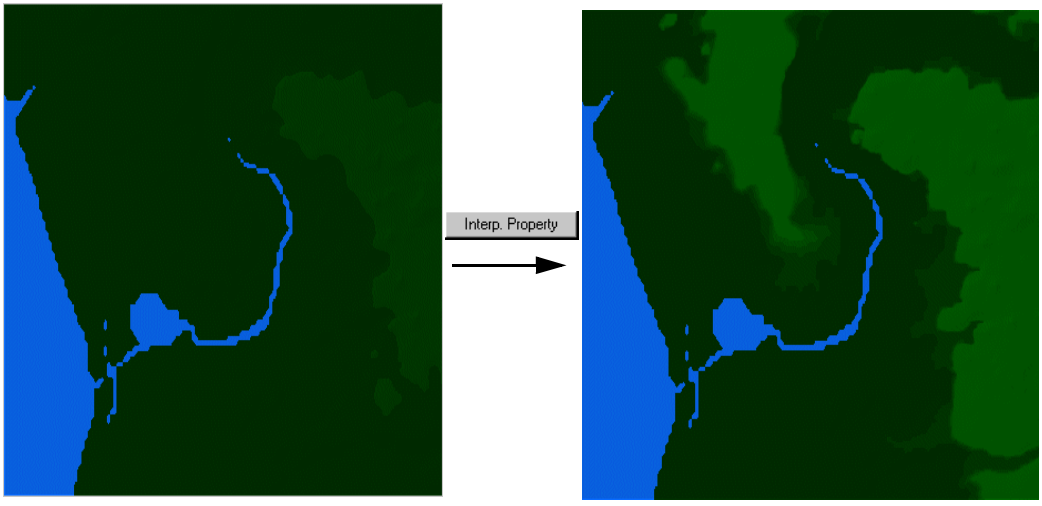
The selected area is almost all the same color because its range of elevations fall within the same contour band. To make this area more visually interesting, you can divide the existing contour bands into smaller bands, and have Creator recalculate the color assignments for each band.

Step 1. If it is not already selected, select the contour band that is just above the **Fixed Contour** in the **Elevation Contours** scale. You can see from the color that this band represents most of the elevations in the **Gaming Area**.

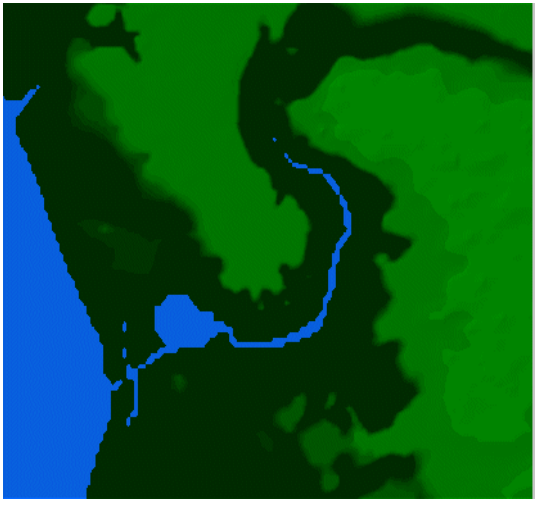


Step 2. Click **Insert**. The selected contour is divided into two equal elevation bands, and the lower band is selected.

Step 3. Click **Interp. Property**. Creator recalculates the colors in the **Elevation Contours** scale, giving each contour band its own color. Notice that certain elevations in the **Gaming Area** become lighter. These are the areas that used to be in the lowest contour band, but are now in the second-lowest band.



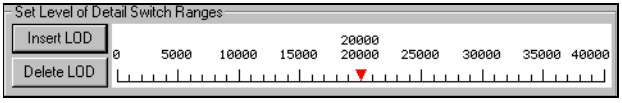
Step 4. Click **Insert**, and then click **Interp. Property** three more times. When you are finished, the area around the inlet has several distinct contour bands.



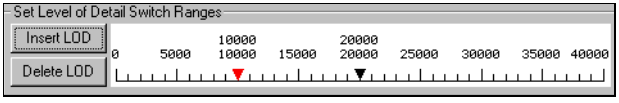
Create LODs

LODs provide a way to reduce the polygon count by displaying simpler representations of terrain that is far from the eyepoint and more complex representations of terrain that is close to the eyepoint. The Basic Terrain option creates *quad tree LODs*, where each LOD is four times more detailed than the next lower LOD.

Step 1. At the lower-left corner of the *Terrain* window is the LOD scale. Click **Insert LOD**. A red marker appears at 20,000 on the scale. You now have two LODs calculated for your terrain: a high LOD that switches in at the eyepoint and switches out at 20,000 database units, and a low LOD that switches in at 20,000 database units and switches out at 40,000 units (the current end of the scale).

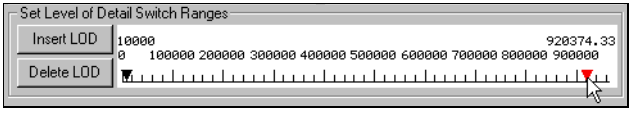


Step 2. Click **Insert LOD** again. A new red marker appears at 10,000 units.

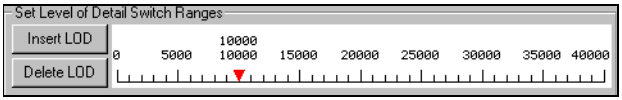


Step 3. Click the marker at 20,000 units. It turns red, indicating that it is the current LOD. The marker at 10,000 turns black.

Step 4. Drag the red marker to the right, beyond the 40,000 mark. Notice that the scale expands to accommodate higher values. The units on the scale also become larger, so that all LODs can be displayed on the scale.

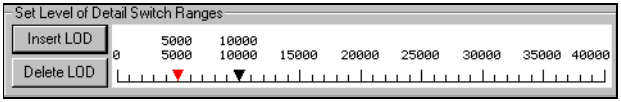


Step 5. Click **Delete LOD**. The LOD that switches in at 10,000 units is now the current LOD.



Step 6. Click **Insert LOD**. The switch-in point for the new LOD is inserted half-way between the eyepoint and the switch-in point for the current LOD. You should now have two markers that define three LODs: a high LOD switches in at the eyepoint and switches out at 5,000 units, a medium LOD switches in at 5,000 units and switches out at 10,000 units, and a low LOD that switches in at 10,000 and switches out at 40,000.

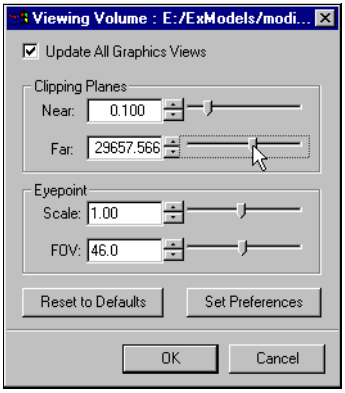
Note: You can change an existing LOD by dragging it to a new location on the scale or by clicking on the number and editing its values directly.



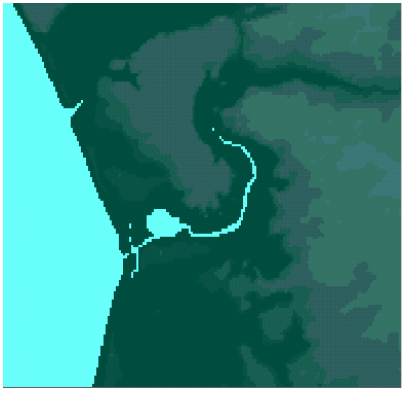
Generate a Test Terrain

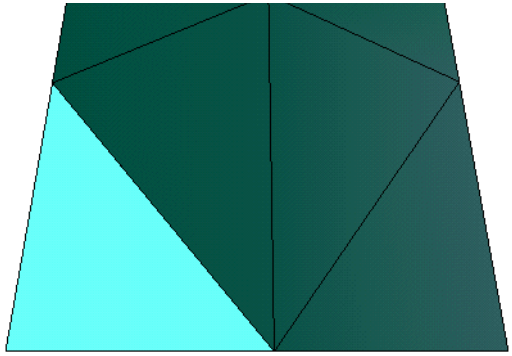
Step 1. Click **OK** to generate terrain at the default setting.

Step 2. If nothing is visible when you generate the terrain, choose **View/Viewing Volume** and move the Far Clipping Plane slider to the right until you see the terrain database.



Notice that the generated terrain bears little resemblance to the **Gaming Area** display in the *Terrain* window. This is because the selected area is almost flat, the flat area is almost at sea level, and no preference has been given to the coastline that defines the water area. We need to adjust the settings and regenerate the terrain.





Step 3. Close the database file without saving and opening a new file.

When Creator generates terrain, it does not overwrite what exists in the database; it covers existing elements at the same longitude and latitude with the newly-generated terrain. To avoid generating a second layer of terrain, you must start a new file each time you make a test terrain or delete the old terrain manually.

Set Triangulation Parameters

Only a small subset of the elevation points contained in the DED file can become the vertices of the triangular polygons that will form the terrain in your database. The polygons are divided into groups by geographical area, to help the realtime software process the terrain efficiently. Triangulation parameters control the number of polygons and groups that are formed, and help determine which elevation points become vertices.

Step 1. Choose **Terrain/New Project**. The DED file `monterey_ca_w.ded` is already selected. Click **OK**.

The *Terrain* window opens with the **Gaming Area** and settings that were chosen earlier in this exercise. This is because each time you generate terrain, Creator creates a preferences file that stores the settings.

Step 2. Enter a **Group Size in Latitude Posts (X)** and **Group Size in Longitude Posts (Y)** of 30 each.

The selected **Gaming Area** was small enough to fit into a single group at the default group size of 100 x 100 posts. Reducing the group size to 30 x 30 posts divides the terrain into smaller groups that can be analyzed independently by the terrain conversion software (and processed independently by the realtime software). It also increases the maximum number of polygons that can be generated, because the **Maximum Polygons per Group** limit applies to each of the smaller groups.

Step 3. Set **Ridge and Valley Detection** to give preference in the triangulation to elevation posts that fall along the high and low points of these areas.

Step 4. Set **Use Post Weighting Multipliers** to tell Creator to perform an analysis of each elevation point and its surrounding points, to find the areas with the greatest variation. Leave **Terrain Accuracy Tolerance** at the default setting, because that setting is ideal for most terrain generation.

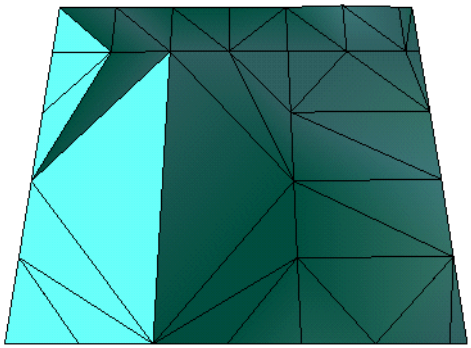
Step 5. Set **Preserve Coastline** to give greater priority in the triangulation to elevation points along the coast.

Step 6. Click **OK** to generate the terrain.

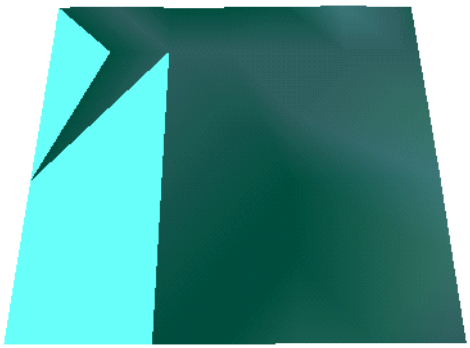
View the Terrain

Use commands in the LOD menu to view the terrain at different LODs.

Step 1. Choose **LOD/Least Detail**. The low LOD view of the terrain appears.

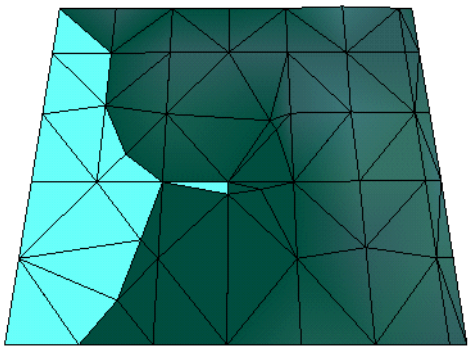


Low LOD: wireframe solid view

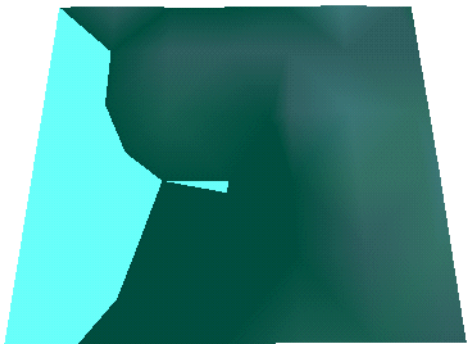


Low LOD: solid view

Step 2. Choose **LOD/More Detail**. The medium LOD appears.

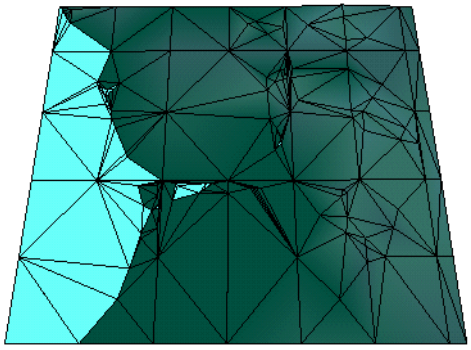


Medium LOD: wireframe solid view

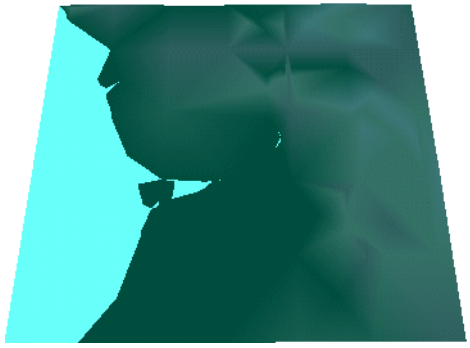


Medium LOD: solid view

Step 3. Choose **LOD/Most Detail**. The high LOD terrain appears.



High LOD: wireframe solid view



High LOD: solid view

Once you generate terrain, you should test it on your realtime system, evaluating appearance and performance. It may take several tries to generate the ideal terrain database for your system. You can also use **Modify Delaunay** to make fine adjustments to your terrain (see “Modify Delaunay” on page 960).

This completes the Terrain exercise.

28 Pro Terrain Conversion

Creator's terrain algorithms provide different methods for converting elevation data to the polygons that form your terrain database. During conversion, only a very small percentage of the elevation posts in your data become the vertices of the polygons that form your final terrain.You choose the algorithm and settings that will give you the best terrain representation that will run efficiently on your target realtime system.

There are four terrain conversion algorithms available with the CreatorPro and Creator TerrainPro options: Polymesh, Delaunay, TCT, and CAT. Each algorithm has its advantages:

- *Polymesh* creates a terrain database consisting of a uniform, rectilinear grid of triangles. This algorithm is ideal for image generators that use Binary Separating Planes (BSPs).
- *Delaunay* creates a terrain database of triangular polygons. This method is best suited for hardware Z-buffering because you can control polygon count and maximize terrain accuracy by creating more polygons in rough terrain areas and fewer in flatter areas.
- *Terrain Culture Triangulation* (TCT) allows pre-projection of feature data. Features such as roads, rivers, lakes, and streams, become a part of the terrain skin, rather than lying on top of it.
- *Continuous Adaptive Terrain* (CAT) provides solutions to some of the traditional problems of terrain LOD management, by performing smooth transitions without popping. The CAT hierarchy is based on a triangular area which results in more efficient culling than with Delaunay or Polymesh; however, it runs only on systems that are running SGI Performer.

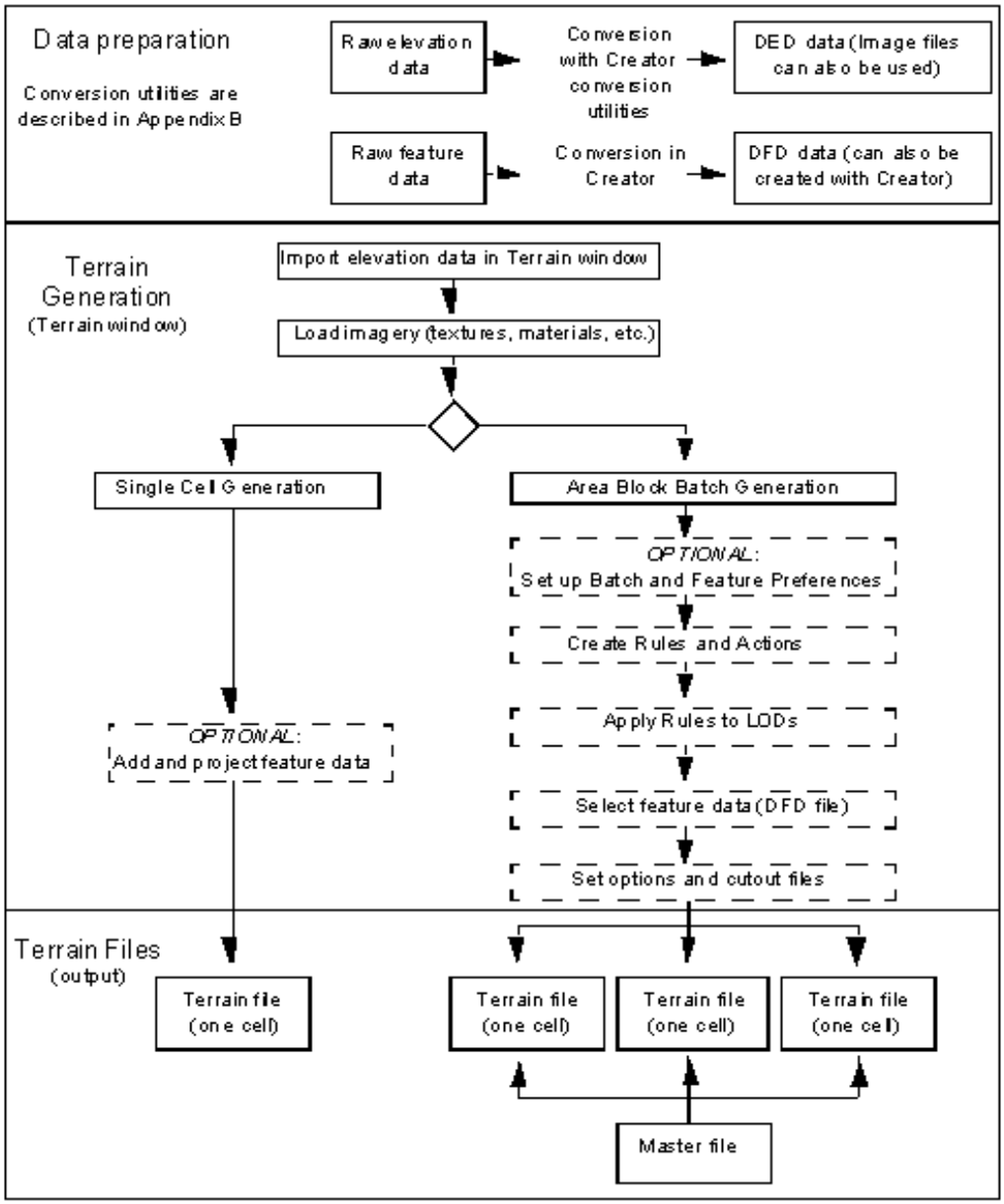
In addition to meeting the requirements of your simulation, the conversion algorithm you choose must be compatible with the type of hardware your database will be running on. Factors to be considered include processing speed, type of edge matching supported, tri-stripping ability, polygon limitations, and BSP or Z-buffer compatibility.

If you are unfamiliar with creating terrain databases, you should read *Creating Terrain for Simulations* to understand the tasks involved in creating a terrain project. This book, which is provided in printed form and as an Acrobat (.pdf) file on the Creator CD, contains a table that lists advantages and limitations of each conversion method and includes descriptions of each algorithm.

The following topics provide more information on CreatorPro and TerrainPro conversions. If you purchased the Basic Terrain option, instead, see “Basic Terrain Conversion” on page 939.

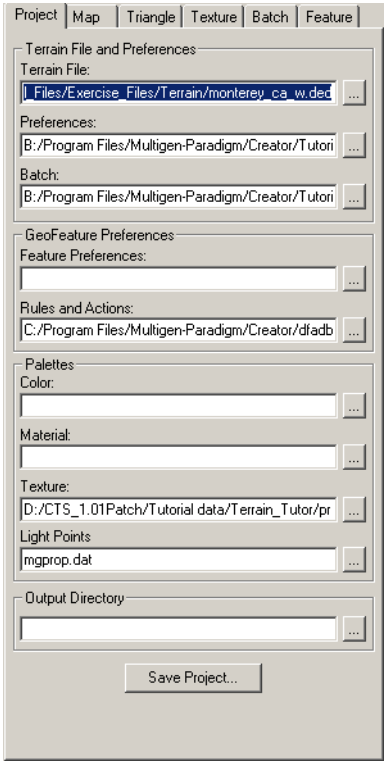
Terrain Process Flowchart

The process of creating terrain files is shown below.


Data preparation
Conversion utilities are described in Appendix B
Raw elevation data
Conversion with Creator conversion utilities
DED data (Image files can also be used)
Raw feature data
Conversion in Creator
DFD data (can also be created with Creator)
Terrain Generation
(Terrain window)
Import elevation data in Terrain window
Load imagery (textures, materials, etc.)
Single Cell Generation
Area Block Batch Generation
OPTIONAL:
Set up Batch and Feature Preferences
Create Rules and Actions
Apply Rules to LODs
Select feature data (DFD file)
Set options and cutout files
OPTIONAL:
Add and project feature data
Terrain Files
(output)
Terrain file (one cell)
Terrain file (one cell)
Terrain file (one cell)
Terrain file (one cell)
Master file

Creating Projects

Projects save terrain and feature processing preferences in a file that can be used any time you want to use a certain set of preferences. Project files specify *Terrain* window settings, feature preferences, default palettes, batch preferences, feature rules and actions, light point preferences, and the output directory to be used for processing terrain. You set up these project preferences in the *Terrain* window's **Project** panel.



You can load a different DED file from the Project panel by specifying it in the Terrain File field or clicking the browse button to the right of the field and choosing the file you want. This eliminates the need to close the *Terrain* window and start a new project in the **Terrain** menu.

You can load existing preference, and batch preference files, or specify new names in which to save your settings. Feature preferences, rules and actions, and light point

palette files must be existing files. Default color, material, and texture palette files must first be created and saved in their respective palettes.

The output directory must be an existing directory. If Creator cannot find the specified directory, it will save files to the directory that contains the post elevation data file (.ded).

Project files can be loaded whenever you wish to process terrain with those settings. When the project file is loaded, it will automatically load the preference files and palettes associated with it. You can simply load a project, click the Ded file button in the Project panel, and choose a different DED file, or process the same DED file and perhaps choose a different set of area blocks to process, or reprocess the same terrain and settings after having corrected feature errors.

To Create a New Project

Step 1. Choose **Terrain/New Project**. The *Open Terrain File* dialog box opens.

Step 2. Select a post elevation (.ded) file, and click **Open**. The post data is loaded into the *Terrain* window.

Step 3. In the **Project** panel, a name and path have already been specified for the terrain preference file, based on the path and name of the elevation data (.ded) file you chose in the previous step. You can specify a different path and name, if you want. For a description of what is stored in this file, see "<ded>.prefs" on page 891.

Step 4. If you are using the batch process, enter a pathname for your batch preference file, or specify the name of an existing batch preference file. For a description of what is stored in this file, see "<.ded>1.bpf" on page 891.

Step 5. Specify the feature preference file (default is dfad.prefs) that you want to use. For a description of what is stored in this file, see "dfad.prefs" on page 891.

Step 6. Type a new name for the light point properties palette file to use. For a description of what is stored in this file, see "mgprop.dat" on page 892.

Step 7. Specify the dfadbat1.prefs file to be used for Rules and Actions. For a description of what is stored in this file, see "dfadbat1.prefs" on page 892.

Step 8. Specify the material, color, and texture palette files you want to use. If you are processing terrain with texture files, either in the contour palette, with indirect texture or geospecific texture, these texture patterns will be loaded and available when the terrain is processed.

Step 9. Specify the path and directory where you want your terrain database to be saved.

Step 10. Click **Save Project**. The *Write Project File* dialog box opens.

Step 11. Type a file name in the **File name** field, set the directory, if necessary, and then click **Save**. The project file is saved to the output directory unless another path and directory is specified. For a description of what is stored in this file, see "<filename>.prj" on page 892.

To Load a Project

Step 1. Click **Terrain/Open Project**. The *Open Project File* dialog box opens.

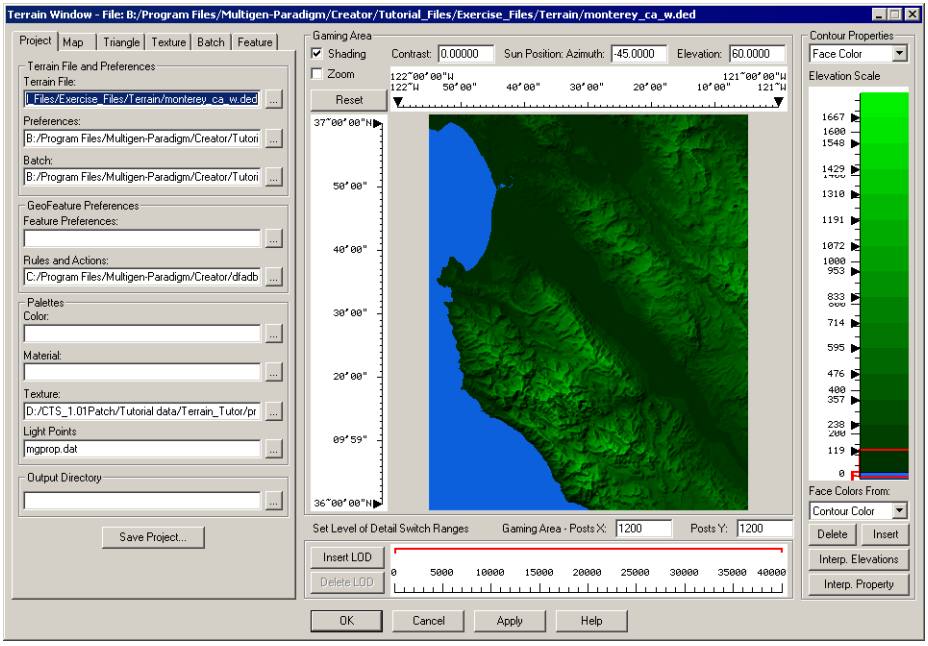
Step 2. Select a project (.prj) file, and click **Open**. The files specified in the project file are automatically loaded and the post elevation (.ded) file is imported into the *Terrain* window.

Note: You can also double-click a project file. Creator will automatically launch and open the *Terrain* window.

Step 3. Process your terrain file as desired.

The Terrain Window

The *Terrain* window contains a contour map of the imported data and controls to generate the terrain database. The controls set up database colors and textures, define terrain conversion preferences, set levels of detail, and add cultural features to the database when it is generated.

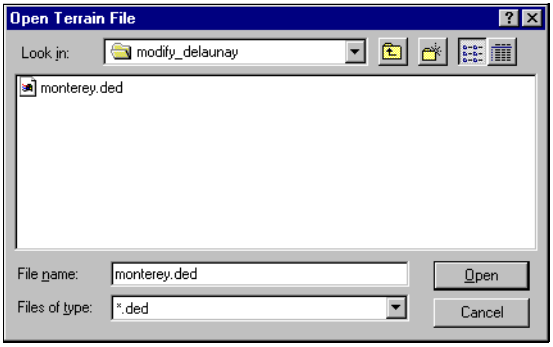


Opening the Terrain Window

To generate a terrain database you must import a file in Digital Elevation Data (DED) format into Creator. Creator opens the DED file in the *Terrain* window, where you can preview the data and generate a polygonal terrain database.

To Import a DED File

Step 1. Choose **Terrain/New Project**. The *Open Terrain File* dialog box appears.



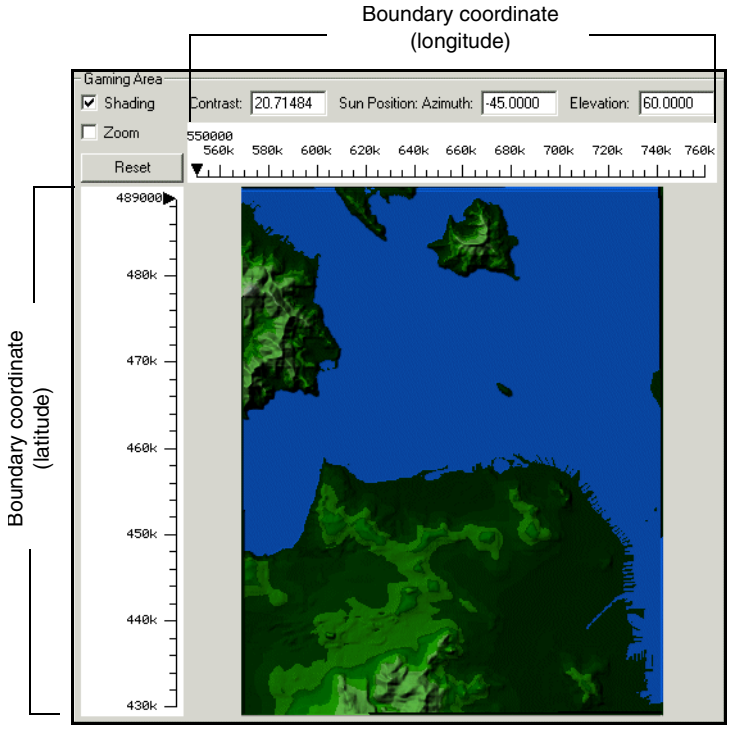
Step 2. Select a DED file, and then click **OK**. The *Terrain* window opens displaying the selected file.

Note: You can specify a default path for the *Open Terrain File* dialog box by setting the MPI_CREATOR_TERPATH environment variable before starting Creator.

For more information about DED files, see "Converting Raw Elevation Data" on page 1084.

Selecting an Area for Processing

Step 1. The imported elevation data is displayed in the **Gaming Area** as a map. You select only the portion of data you want to process. You do not need to use all of the data contained in the .ded file you imported.



Step 2. You can specify whether you want the contour map to appear as a flat display or as a shaded relief. When **Shading** is set, you can set the **Sun Position: Azimuth** and **Elevation** values to determine the height and angle of the light illuminating the terrain. These settings have no effect on the generated terrain. However, using the controls in the **Texture** panel, you can save the contour map as a georeferenced image, which can then be applied as a geospecific or indirect texture (see "Texture Panel Preferences" on page 1047).

Step 3. Drag a fence around the area that you want to include in your terrain database (holding down Shift in batch mode), or drag the arrows in the Latitude and Longitude scales to select the area. The selected area becomes the new **Gaming Area**. Alternatively, you can enter the number of posts for longitude in the **Gaming Area- Posts X** field and latitude in the **Gaming Area- Posts Y** field.

Step 4. To display an enlarged view of the selected area, set the **Zoom** checkbox. To reselect the entire file, click **Reset**. You can also zoom in and out using key-and-mouse combinations. The results depend on the direction in which you drag the mouse:

- Hold down Shift and the right mouse button and drag towards the top of the gaming area to zoom in, in Y; drag towards the bottom of the Gaming Area to zoom out.
- Hold down Shift and the right mouse button and drag to the right in the Gaming Area to zoom in, in X; drag to the left to zoom out.

If you find that you want to see an area that is just out of view, you can hold down Ctrl and drag with the right mouse to pan in the Gaming Area.

Gaming Area Parameters

You can set the following parameters in the *Terrain* window's **Gaming Area**.

Shading

When this checkbox is set, the contour map displays as a shaded relief. When it is cleared, the contour map displays as a flat image.

Zoom

Magnifies the selected area of the contour map so that it fills the viewing area.

Reset

Returns the display of the contour map to its default setting.

Contrast

Controls the amount of lightness and darkness from the sun. A larger number produces more contrast.

Sun Position: Azimuth

Sets the position of the "sun" as it lights up the terrain. The range of values is -90 to 90, where 0 is north and any positive value indicates clockwise rotation. This option is active when **Shading** is set.

Elevation

Sets the height of the "sun" as it lights up the terrain. The range of values is 0 to 90, where any positive value indicates that the sun is above the horizon.

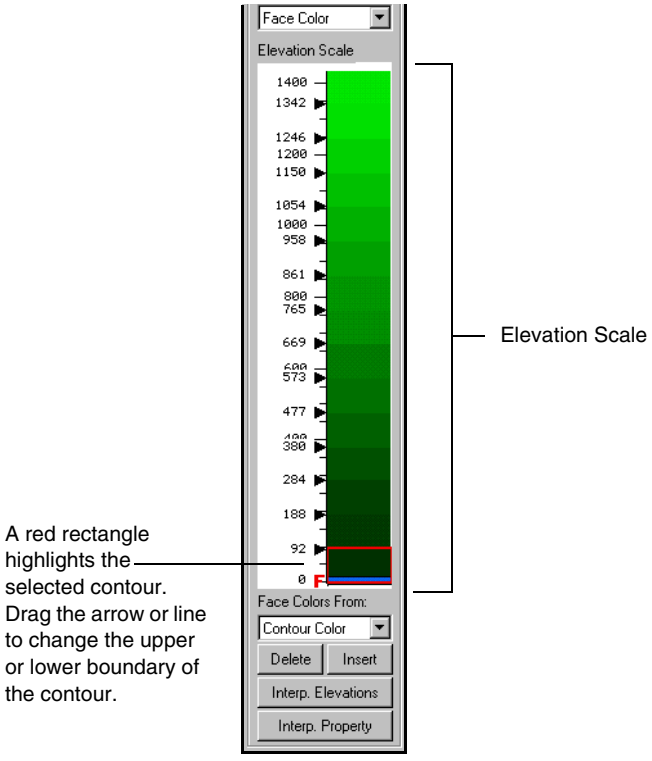
Changing Contours and Their Properties

A *contour line* is a line on a map that joins points of equal elevation. A *contour* is an area between two contour lines that represents a range of elevations. The entire range of contours is displayed in an **Elevation Scale**.

When you import elevation data, the map in the **Gaming Area** displays color-coded contours calculated from the data. You can change the range of elevations represented by any contour, add or delete contours, and change contour properties by making changes to the **Elevation Scale**.

To Select and Resize a Contour

To select a contour in the **Elevation Scale**, click it. A red outline appears around the contour. To change the size of a contour, drag the upper or lower boundary to the desired elevation or click on it to edit its value directly. The other contours resize to accommodate the change. In addition, the **Gaming Area** updates to reflect the change.



To Add a Contour

To add a contour, select a contour and click **Insert**. The selected band splits in half. If no contour is selected, the lowest one splits in half.

To Delete a Contour

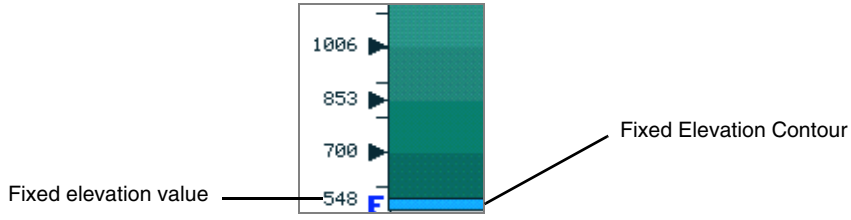
To delete a contour, click it, then click **Delete**. If no contour is selected, the lowest one is deleted. The elevations represented by the deleted contour are added to the next highest contour.

To Create Uniform Contours

To create contours of equal size, click **Interp. Elevations**. Existing contours are resized until each spans an equal range of elevations. If you find that you have too many or too few bands, use **Insert** or **Delete** to add or eliminate selected contours, and then click **Interp. Elevations** to redistribute the elevations assigned to each contour.

To Change the Fixed Elevation Contour

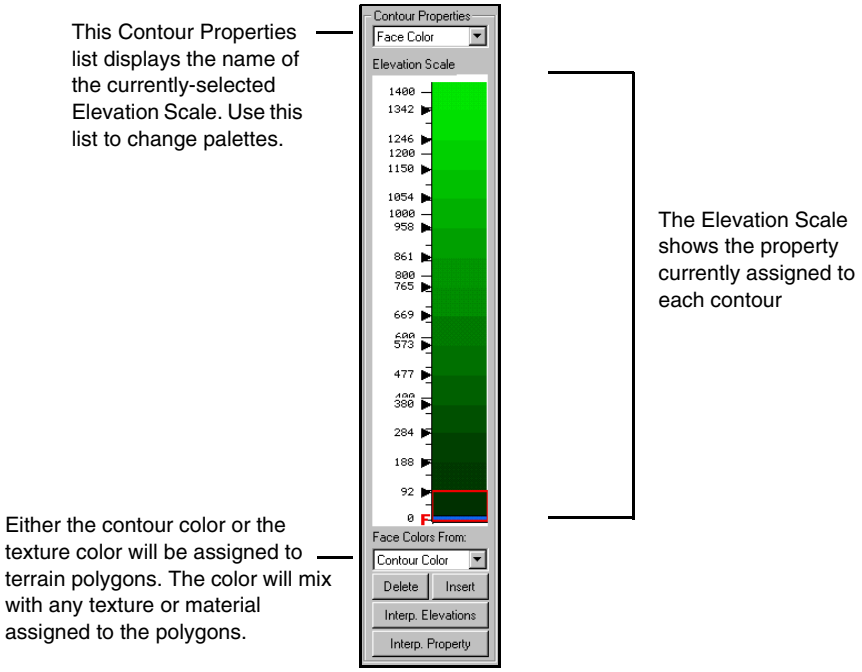
The **Fixed Elevation Contour** (labeled **F** in the display) assigns a color, texture, and material to one specific elevation. This is typically used to make sea-level polygons blue, but you can change the fixed elevation by dragging the **F** up or down, or by editing the Fixed Elevation value. Assignments made to the **Fixed Elevation Contour** are not affected by **Interp. Elevations** or **Interp. Property**.



Assigning Properties to Contours

Contour properties determine the contour or texture color, texture, and material assigned to polygons in the new terrain database. As each polygon is generated, its color, texture, and material properties are taken from the contour that corresponds to the polygon's median elevation.

You can assign a contour color, texture color, texture pattern, and material to any or all contours using the **Elevation Scales** in the *Terrain* window.



The properties you assign have the following priorities:

- The **Face Colors From** setting chooses whether the contour color or texture color shown in the **Elevation Scale** is applied to the polygons when the terrain database is generated. Choosing **Contour Color** causes **Texture Color** to be ignored; if you choose **Texture Color**, the **Contour Color** setting is ignored.
- If you assign a texture or material to a contour, the face color or texture color mixes with the material or texture pattern. One way to avoid this is to assign a **Texture Color** of white to every contour and select **Face Colors From: Texture Color**. (The default assignment for **Texture Color** is white.)
- If you set **Indirect Texture**, the terrain is textured according to the pattern of a color-keyed image and its corresponding text file. In this case, all other contour properties are ignored (see “Using Indirect Texture” on page 1185 for more information).

To Assign Properties to a Contour

Step 1. From the pull-down menu, select the **Elevation Scale** (**Face Color**, **Texture Color**, **Material**, or **Texture**) you want to change.

Step 2. Double-click a contour to open the corresponding Color, Texture, Material, or Texture Color palette.

Step 3. Select a color, texture color, texture, or material. The item you select in the palette is assigned to the contours that you click.

Step 4. To change another contour, select it, and then click another item in the palette. Repeat this step for every contour you want to change, or use interpolation to fill in the other entries in the palette.

Step 5. You do not have to make assignments to every contour. If a texture or material entry is blank, a value of -1 (no texture/material) is assigned to the faces in the corresponding range of elevations. Default colors and texture colors appear in the **Elevation Scale** when you have selected **Face Color** or **Texture Color**.

To Interpolate a Palette

Interpolation fills in the current **Elevation Scale** using the contents of the lowest and highest contours.

To interpolate a palette with face or texture colors:

Step 1. From the **Contour Properties** list, select **Face Color** or **Texture Color**. Use the **Elevation Scale** to assign colors to the lowest and highest contours, and then click **Interp. Property** to fill all remaining contours with a smooth gradient of these two colors.

To interpolate a palette with textures or materials:

Step 1. Select **Texture** or **Material** from the **Contour Properties** list. Use the **Elevation Scale** to assign a texture pattern or material to the lowest contour, and then click **Interp. Property** to copy this texture or material to all other contours. Texture patterns or materials assigned to higher contours are removed from the **Elevation Scale** (but remain in the Texture or Material palette).

Setting Terrain Levels of Detail

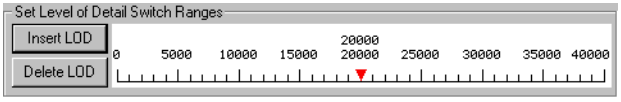
Levels of detail (LODs) are sets of faces that provide varying amounts of resolution for each terrain area. LOD switching allows the realtime software to display areas of terrain that are close to the eyepoint at high resolution, and areas that are farther away from the eyepoint at low resolution. As the eyepoint moves through the database, the realtime software switches each LOD in or out dynamically.

The point at which a lower resolution LOD is replaced with the next higher level of resolution is called the *switching distance* and is specified by the **Switch Out** and **Switch In** LOD attributes. Creator's **Switch Out** attribute for each lower resolution LOD matches the **Switch In** distance of the next higher resolution LOD when generated with the Basic Terrain or Terrain-Pro option. For more information about levels of detail, see "Levels of Detail" on page 707.

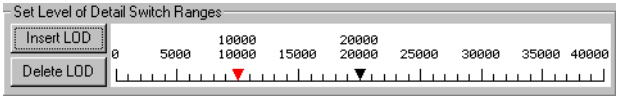
To Create Terrain LODs

LODs provide a way to reduce the polygon count by displaying simpler representations of terrain that is far from the eyepoint and more complex representations of terrain that is close to the eyepoint.

Step 1. The LOD scale is located at the lower-left corner of the *Import Terrain* window. Click **Insert LOD**. A red marker appears at 20,000 database units on the scale. You now have two LODs calculated for your terrain: a high LOD that switches in at the eyepoint and switches out at 20,000 units, and a low LOD that switches in at 20,000 database units and switches out at 40,000 units (the current end of the scale).

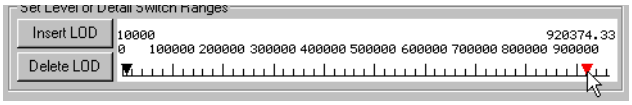


Step 2. Click **Insert LOD** again. A new red marker appears at 10,000 units.

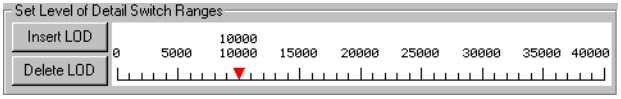


Step 3. Click the marker at 20,000 units. It turns red, indicating that it is the current LOD. The marker at 10,000 turns black.

Step 4. Drag the red marker to the right, beyond the 40,000 mark. Notice that the scale expands to accommodate higher values. The units on the scale also become larger, so that all LODs can be displayed on the scale.

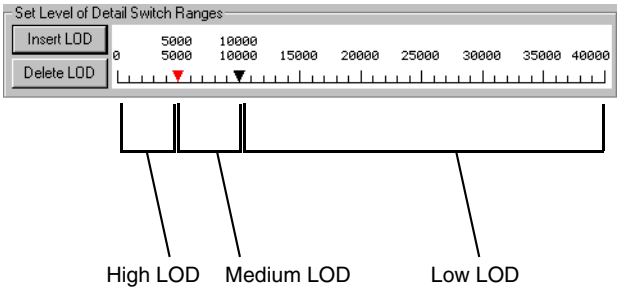


Step 5. Click **Delete LOD**. The LOD that switches in at 10,000 units is now the current LOD.



Step 6. Click **Insert LOD**. The switch-in point for the new LOD is inserted half-way between the eyepoint and the switch-in point for the current LOD. You should now have two markers that define three LODs: a high LOD that switches in at the eyepoint and switches out at 5,000 units, a medium LOD that switches in at 5,000 units and switches out at 10,000 units, and a low LOD that switches in at 10,000 and switches out at 40,000.

Note: You can change an existing LOD by dragging it to a new location on the scale or by clicking on the number and editing its values directly.



Setting Panel Preferences

The *Terrain* window has six panels where you can set preferences for terrain generation and feature projection. Click each of the tabs to view the **Project**, **Map**, **Triangle**, **Texture**, **Batch**, and **Feature** panels.

The Project Panel

The **Project** panel lets you control the location of all the files associated with a terrain project. You can change the elevation data file being processed, and set the preference files, property palettes, light point palettes and output directory to be used in the process. You can also specify color, material, and texture palettes. These are all optional selections. Creator uses defaults if you do not name specific files. Selecting your own files allows you to save your preferences as project files and reload them whenever the same settings are needed.

The Project panel has the following settings:

Terrain File and Preferences Settings

DED



Click the button to open the *Choose New Terrain File* dialog box, where you can select and load a new DED file.

Preferences



Specifies a file to store contour palette, LOD, and Triangulation settings. By default this file is located in the same directory as the .ded file with the file name <.ded file name>.prefs, You can highlight the field and type a different path and/or file name, or click the browse button to open the *Choose Terrain Pref File* dialog box, and select a different file of previously saved settings. If you do not choose a specific file, a new file of terrain window settings will be created in the same directory as the current DED file named <DEDfilename>.prefs.

Batch



Click the button to open the *Choose Terrain Batch Pref File* dialog box, where you can select a saved batch preferences file. If you do not choose a specific file, Creator creates a new batch preferences file with the name of the current DED file and a .bpf extension.

GeoFeature Preferences

Feature Preferences



Displays the path and file name of the Feature Preferences file to be used when processing feature data with the .ded file. Click the button to open the *Choose DFAD Preferences file* dialog box, where you can select the dfad.prefs file to use. Creator uses the default dfad.prefs file if you do not choose a specific file.

Rules and Actions



Displays the dfadbat1.prefs file to be used when processing feature data with the .ded file. Click the button to open the *Choose DFAD Batch Preferences file* dialog box, where you can select the dfadbat1.prefs file to use for batch conversions. Creator uses the default dfadbat1.prefs file if you do not choose a specific file.

Palettes

Color



Displays the path and file name of the Color Palette file to be used when processing the elevation data.Click the button to open the *Choose Color Palette File* dialog box, where you can select the.color file to use.

Material



Click the button to open the *Choose Material Palette File* dialog box, where you can select the .materials file to use.

Texture



Click the button to open the *Choose Texture Palette File* dialog box, where you can select the .txt file to use.

Note: The .txt file extension denotes a texture palette file, which is in standard ASCII text file format.

Light Points



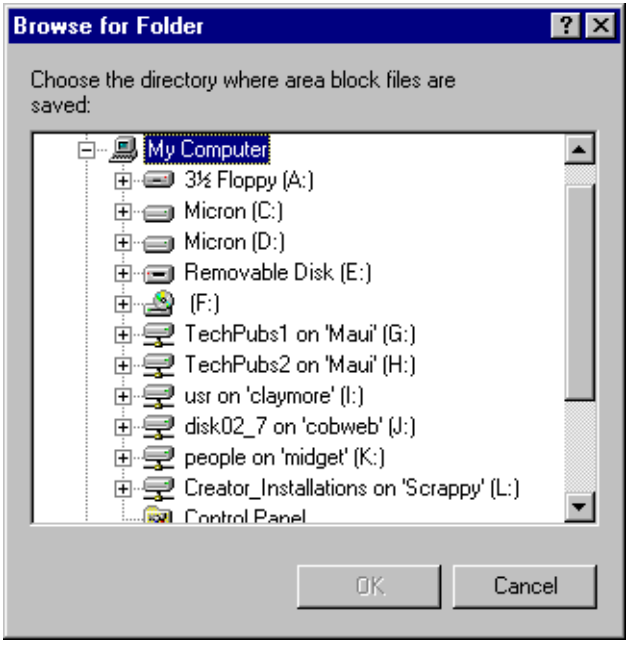
Click the button to open the *Choose Light Point Preferences File* dialog box, where you can select the .dat file to use.

Note: The .dat file extension denotes a light point palette file, which is in standard ASCII text file format.

Output Directory



Click the button to open the *Browse for Folder* dialog box, where you can select the directory in which to save your generated terrain files. If you do not set this path, Creator prompts you for it the first time you run a batch file.

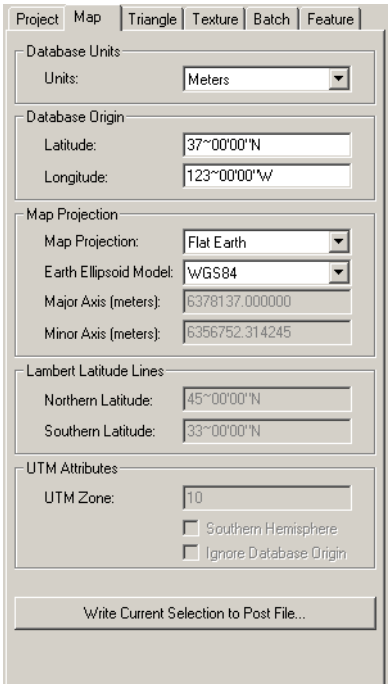


The Map Panel

The Map panel preferences control how the terrain will be projected, based on specific units and coordinates for each vertex. The type of projection used is determined by the shape of the Earth, and whether the terrain has a flat or curved surface; it should produce the least amount of distortion in the terrain. The **Map** panel also sets the unit of measure for the databases that are created.

Map Panel Preferences

The Map panel contains information about mapping gridded terrain posts into database coordinates.



Database Units

Defines database units for the database header attributes. Units can be **Meters**, **Kilometers**, **Inches**, **Feet**, or **Nautical Miles**.

Database Origin

Specifies the latitude (**Latitude**) and longitude (**Longitude**) to place at the database origin. **Database Origin** defaults to the southwest corner (lower-left corner) of the DED file. The database origin corresponds to 0,0,0, in database units.

Map Projection

Selects a map projection to use when converting post values to database coordinates. This information is saved in the new database header attributes. You can change the projection of a database by using **Convert Database Projection** from the **Terrain** menu. (See “Convert Database Projection” on page 1059.) The map projection types:

Flat Earth

Uses latitude at the origin to derive a single convergence factor for adjusting every *x* value. This results in a rectangular database.

Trapezoidal

Is an azimuthal projection that is most accurate at its central point, with distortion increasing as the distance increases from this point.

Lambert Conic Conformal

Uses two standard parallels and is most accurate for areas located in the middle latitudes between 84 degrees North and 80 degrees South. Distortion increases closer to the poles.This projection method requires you to supply the two lines of latitude in the **Lambert Upper** and **Lower** fields.

UTM

Generates a database using a cylindrical projection, rotated 90 degrees. UTM divides the earth into six degree longitudinal zones with a central meridian. It is most accurate along the meridian. Distortion increases as you move farther east or west of the meridian. Enter the UTM Attributes.

Geocentric

Creates a database using a round earth projection where Z radiates from the center of the Earth through the North Pole.

Earth Ellipsoid Model

A geometric model having fixed dimensions for major and minor axes which is used to reconcile the Earth's curvature and measured values at different latitudes. Options are **WGS84**, **WGS72**, **Bessel**, **Clark 1866**, and **User Defined Ellipsoid**. If you choose **User Defined Ellipsoid**, enter the Major Axis and Minor Axis for the ellipsoid.

Lambert Latitude Lines

Specifies two lines of latitude for a Lambert database projection, **Northern Latitude** and **Southern Latitude**.

UTM Attributes

If you selected a **UTM** Map Projection, enter a new UTM projection number (1-60), or leave the current UTM projection number; set **Southern Hemisphere** if the database is in

the southern hemisphere; and set **Ignore Database Origin** to override all of the database origins and use the origin defined by UTM.

Write Current Selection to Post File

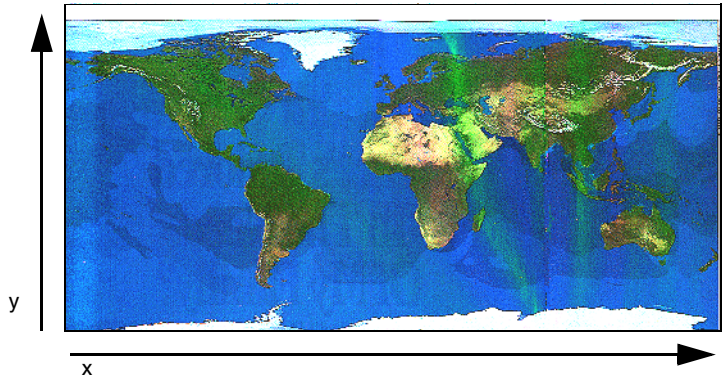
Saves the selected **Gaming Area** in a new DED (gridded terrain post) file.

Map Projection Types

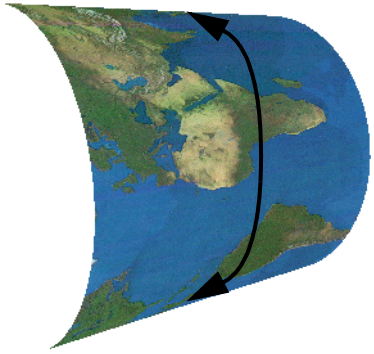
The **Map** panel in the *Terrain* window has a number of choices. What do these mean, and how do you know which is the right one for your purpose?

The goal of choosing projection and ellipsoid settings is to introduce the least amount of distortion into your generated terrain. Because the Earth is not a perfectly round ball, projections and ellipsoids compensate for Earth's irregularities at certain places. Terrain near the equator needs a different projection than terrain near either pole. The most commonly used projections are Flat Earth and Universal Transverse Mercator (UTM).

Flat Earth projection treats latitude and longitude coordinates as x and y coordinates. This results in a rectangular database. Flat Earth projections are generally used for databases which run more east-to-west and are not too close to either pole.

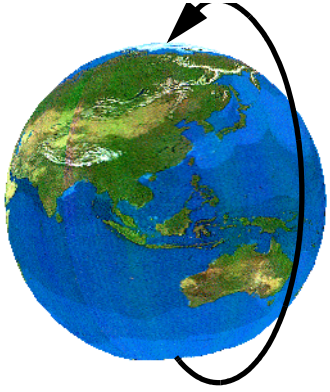


Universal Transverse Mercator (UTM) works well for databases that run more north-to-south, or along polar routes. UTM is divided into six degree zones.

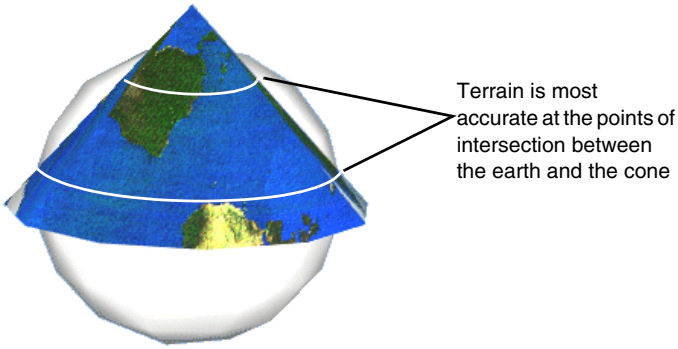


Most accurate along the center vertical, becoming less accurate as you move east or west

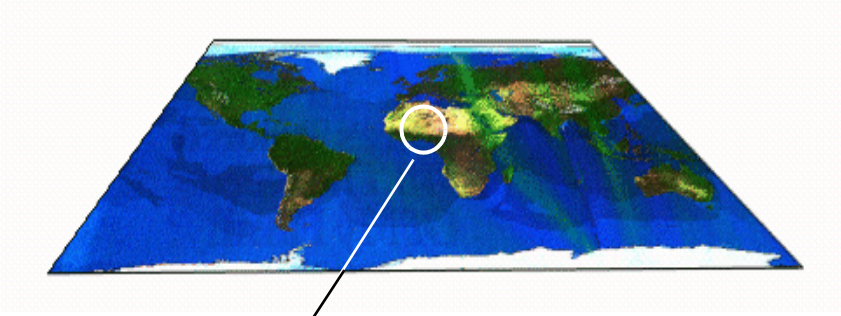
A Geocentric projection is a round earth projection where z radiates from the center of the Earth through the North Pole. This projection is used for large area databases or for simulating the earth's curvature.



Lambert Conic Conformal works best for areas located in the latitudes between 60 to 80 degrees north or south. Within these middle latitudes, this projection compensates for the longitudinal convergence near the poles, and the longitudinal expansion near the equator.



Trapezoidal is best for small (one degree x one degree or less) databases. This projection compensates for latitudinal convergence near the poles.



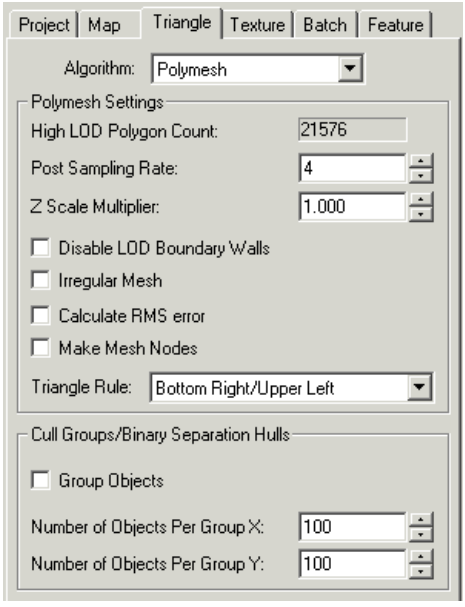
Ellipsoids are mathematical formulas that define the shape of the Earth, which bulges in the middle and is flattened at the poles. Cartographic data uses several different ellipsoids. For example, the ellipsoids used primarily for mapping data in United States are WGS 1972 and WGS 1984. Ellipsoids are used in conjunction with projections to convert real world locations to places on a map. All maps specify the earth ellipsoid best suited for the area.

When you apply a geospecific texture to your terrain, you must make sure it has the same projection and ellipsoid as your terrain or it may not map correctly.

The Triangle Panel - Choosing an Algorithm

The Triangle panel is unique, in that it has different preferences, depending upon which conversion algorithm you select. Creator gives you several algorithms to choose from when converting elevation data to terrain. When you select an algorithm, the panel displays the preferences available for that algorithm.

Triangle panel with Polymesh algorithm selected.



These preferences are described in “Setting Polymesh Conversion Preferences” on page 1012, “Setting Delaunay Conversion Preferences” on page 1019, “Setting TCT Preferences” on page 1031, and “Setting CAT Conversion Preferences” on page 1039.

The conversion algorithm you choose will depend upon the type of hardware your database will be running on. Factors to be considered include processing speed, type of edge matching supported, tri-stripping ability, polygon limitations, and BSP compatibility. The following table describes each algorithm’s advantages and limitations and the type of edge matching it uses. Use this table as a general guide for choosing the best conversion algorithm for your needs.

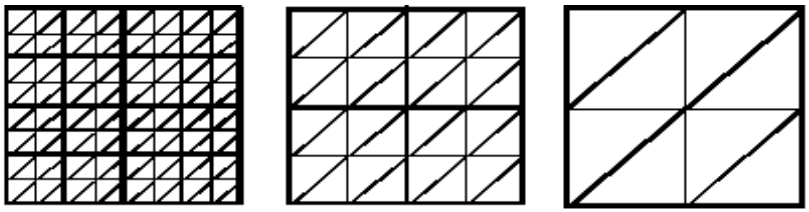
Algorithm/Edge Matching Type	Advantages	Limitations
Polymesh/No walls	• Triangle-strips well • No inherent limitation on the number of LODs created • Exactly predictable - no variation in terrain skin, resulting in efficient paging • Creates terrain quickly • Processing is not vertex limited • BSP-compatible	• May overtessellate flat areas • Can produce edge gaps between LODs • No error tolerance testing
Polymesh/Walls	• Exactly predictable - no variation in terrain skin resulting in efficient paging • Creates terrain quickly • No inherent limitation on the number of LODs • Triangle-strips well • Processing is not vertex limited • BSP-compatible	• May overtessellate flat areas • Can produce artifacts (discontinuity) along edges between LODs • No error tolerance testing • Walls do not tri-strip • Terrain with extreme variations in elevation are not represented as well as with other algorithms
Polymesh/Irregular Mesh, no walls	• No inherent limitation on the number of LODs created • Processing is not vertex limited • Error tolerance testing • Triangle-strips well • More accurate terrain description for the same number of polygons • Exactly predictable	• Slow processing speed • Not BSP-compatible

Algorithm/Edge Matching Type	Advantages	Limitations
Delaunay/Exact Edge Delaunay/No Matching	• Error tolerance testing • Best terrain fit for the least amount of polygons • Edge matching between LODs	• Polygon limitation • Unpredictable number of polygons • Creates short triangle-strips • LOD edge matching can result in large polygons and slivers
Terrain Culture Triangulation (TCT)	• Error tolerance testing • Feature pre-projection • Pathfinder roads are drivable • Best terrain fit for the least amount of polygons	• Polygon limitation • Unpredictable number of polygons • Creates short triangle-strips • Single LOD
CAT/Morphing	• Error tolerance testing • Best terrain fit for the least amount of polygons • Edge matching between LODs	• Only runs on systems running SGI Performer • Very CPU-intensive • Features must be cut into terrain with adaptive attributes added for morphing between LODs • Geospecific textures must be at least as large as the terrain database because cut-in is not possible

Polymesh Terrain Conversion

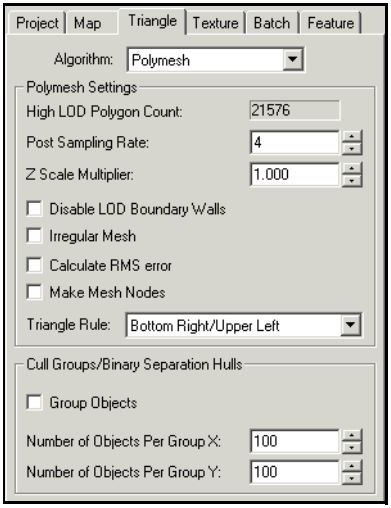
Polymesh is a rectangular or trapezoidal grid created by subsampling the elevation grid. It is generally used to create terrain on systems that use binary separating planes. The Polymesh algorithm converts terrain in the following manner:

- Creator samples every *n*th post (*n* = **Post Sampling Rate**) in the original DED file (gridded terrain post data). These posts become vertices in the new database.
- The *z* component of each new vertex is multiplied by **Z scale**.
- In the highest detail LOD, each block of four vertices becomes one terrain object made up of two triangular polygons. Each block of objects of size **Number of Objects per Group X** by **Number of Objects per Group Y** becomes one terrain group.
- For each LOD, Creator forms one terrain group to correspond to each 2 x 2 block of terrain groups in the next higher detail LOD. Because the number of objects per group is constant across all levels of detail, terrain objects are larger, as shown in the figure below.



Setting Polymesh Conversion Preferences

To generate a Polymesh terrain, click the Triangle tab in the *Terrain* window, and then choose **Polymesh** from the **Algorithm** list. The Triangle tab displays **Polymesh Settings** and **Cull Groups/Binary Separation Hulls** parameters, which control the amount of resolution in the terrain database as well as its hierarchy and other characteristics.



Polymesh Settings

Polymesh settings determine how many triangles are formed, including which posts are used as vertices of these triangles, when the terrain is generated.

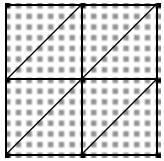
High LOD Polygon Count

Displays the number of triangular polygons that will be generated given the current **Post Sampling Rate**.

The value of **High LOD Polygon Count** provides a good estimate of the computational power required to generate and display the current terrain database. As you raise and lower **Post Sampling Rate**, the number of polygons goes down or up proportionately. When you choose a **Post Sampling Rate**, take into account your system's disk space and available memory as well as the number of polygons your image generator can process efficiently.

Post Sampling Rate

Defines the sampling rate of the gridded terrain post data for the highest detail LOD. The corners of each **Post Sampling Rate** block of posts become vertices in the terrain object. All other posts are discarded.



At a post sampling rate of 8, the corners of each 8 x 8 block of posts become vertices in the new object. Each square is one object. All other posts are discarded.

Z Scale Multiplier

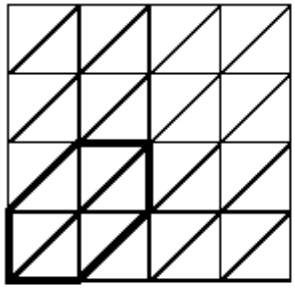
Scales each post value as it is converted to a database coordinate. For example, a **Z Scale Multiplier** of 3.000 changes the *z* component of each vertex to three times its original elevation, exaggerating the elevation differences in the final terrain.

Disable LOD Boundary Walls

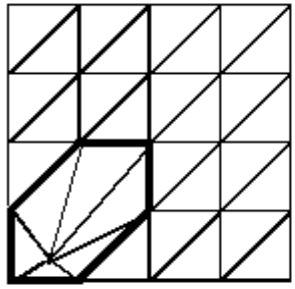
Inhibits the placement of a wall (or skirt) around the perimeter of each database and LOD within a database. Walls are generally inserted to prevent holes from appearing between databases or levels of detail.

Irregular Mesh

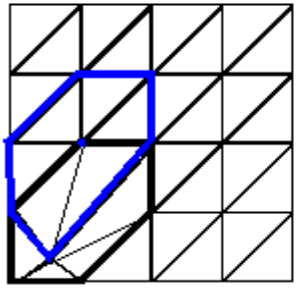
Uses the Post Sampling Rate a little differently. It creates a non-rectangular polygon, samples each post lying within the polygon for the maximum deviation, and then moves the polygon's midpoint to the new location. The result is irregular-shaped triangles that describe the terrain more accurately than regular polymesh for the same number of polygons.



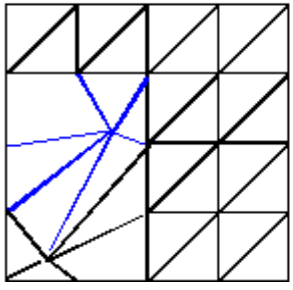
A polygon is created in the tri-strip.



Creator tests each post that falls within the polygon for maximum error and moves the midpoint of the polygon to the new location.



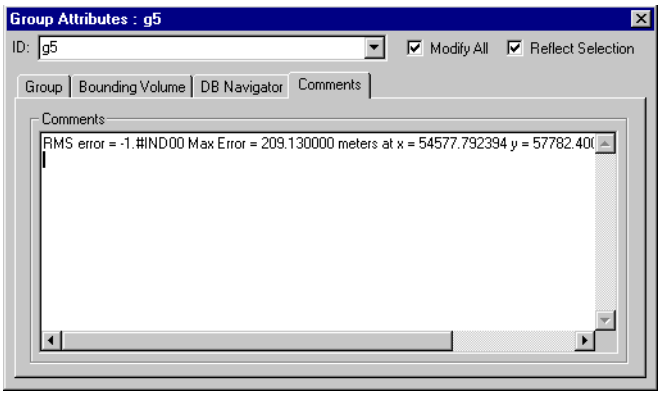
The next polygon in the tri-strip is created and the process begins again.



The resulting triangles are irregular but describe the terrain more accurately and preserve the polygon count.

Calculate RMS Error

Measures the amount of deviation between sampled posts and calculates the root mean square. The resulting information is written to the Comments panel of the *Group Attributes* window.



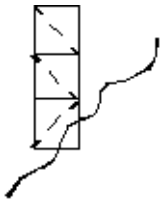
Make Mesh Nodes

Combines the terrain face nodes into mesh nodes to facilitate faster performance. You cannot project features with terrain, or post-project features onto the resulting terrain if you select this option. (See “Using Meshes With Terrain” on page 383 for more information.)

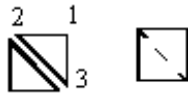
Triangle Rule

The **Triangle Rule** list has the following options:

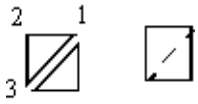
- **Best Fit** produces terrain that better adapts to the coastline or other flat areas. If three vertices are at the same elevation the vertices will form a triangle.



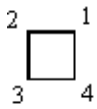
- **Best Fit** and **Quads in Flat Area** default to finding **Bottom Right/Upper Left** whenever possible.
- **Bottom Right/Upper Left** produces terrain that allows optimal triangular meshes to be built in SGI Performer.



- **Bottom Left/Upper Right** produces terrain that is best suited for machines (game consoles, for example) or rasterizing algorithms that optimize on quadrilaterals.



- **Quads in Flat Area** produces terrain for machines or rasterizing algorithms that optimize on quadrilaterals. If four vertices are at the same elevation, the vertices will form a quadrilateral instead of two triangles.



Cull Groups/Binary Separation Hulls

These parameters control how the Polymesh terrain triangles are divided into objects and groups for efficient culling and/or to make BSP separation of the terrain possible.

Group Objects

Modifies the hierarchy to produce one face per object and one object per group to ensure that the database is suitable for binary separation.

Number of Objects Per Group X

Defines the number of objects in each group along the *x* axis. This number is the same in every LOD. The number of objects per group is varied in each LOD by using fewer terrain objects in lower resolution levels of detail.

Number of Objects Per Group Y

Defines the number of objects in each group along the *y* axis. This number is the same in every LOD. The number of objects per group is varied in each LOD by using fewer terrain objects in lower resolution levels of detail.

Delaunay Conversion

In general, Delaunay triangulation proceeds as follows for each LOD:

1. Each terrain post in the selected area of the DED file is sampled.
2. If **Ridge and Valley Detection** is set, a set of posts that defines the ridges and valleys is added to the triangulation. If **Preserve Coastline** is set, Creator tests the edge of each new polygon for intersection with the coastline and adds coastal posts to the triangulation. The longest edges are considered first, and edges on the group boundary have higher priority than interior edges.

   This step repeats until the difference between the elevation of each unused post and the average elevation of its neighbors is less than **Ridge and Valley Tolerance**, or Creator reaches the limit of **Maximum Faces/Group**.
3. If **Use Post Weighting Parameters** is set, Creator computes the distance from each post that is not included in the triangulation to the surface of the corresponding polygon and adds the post with the greatest distance to the triangulation. If **Preserve Coastline** is set, Creator tests the new polygon edges for intersection with the coastline and adds coastal posts to the triangulation.

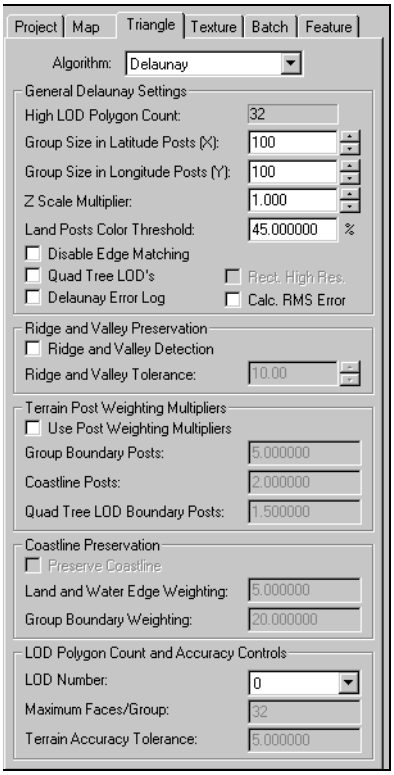
   This step repeats until the average distance between each unused post and the corresponding polygon is less than **Terrain Accuracy Tolerance** or Creator reaches the limit of **Maximum Faces/Group**.

Delaunay triangulation starts with the lowest resolution LOD. After a LOD is triangulated, its vertices are incorporated into the next higher LOD (if any) and form the basis of its triangulation. This ensures that vertices of each LOD match vertices in all higher resolution levels of detail, which results in smooth LOD switching.

Delaunay triangulation is generally used for systems that use z-buffering because it generates fewer polygons than with Polymesh for the same degree of accuracy.

Setting Delaunay Conversion Preferences

Delaunay conversion preferences control the amount of resolution in a terrain database and concentrate the resolution in specific portions of terrain, for example, along a coastline. Delaunay preferences select the options used to process terrain. If all Delaunay checkboxes are cleared, the triangulation incorporates only the corners of each group and is, therefore, extremely coarse.



When you select the Delaunay algorithm, the **Triangle** panel contains areas called **General Delaunay Settings**, **Ridge and Valley Preservation**, **Terrain Post Weighting Multipliers**, **Coastline Preservation**, and **LOD Polygon Count and Accuracy Controls**.

General Delaunay Settings

These settings affect the size and height of the database.

High LOD Polygon Count

Displays the number of triangular polygons that will be generated given the current **Group Size in Latitude Posts (X)** and **Group Size in Longitude Posts (Y)**).

The value of **High LOD Polygon Count** provides a good estimate of the computational power required to generate and display the current terrain database. As you raise and lower the group sizes, the number of polygons goes down or up proportionately. When you choose a **Group Size in Latitude Posts (X)** and **Group Size in Longitude Posts (Y)**, take into account the disk space and memory available on your computer, and the number of polygons your image generator can process efficiently.

Group Size in Latitude Posts (X and Y)

Group Size in Latitude Posts (X) and **Group Size in Longitude Posts (Y)** define the length and width of each rectangular terrain group in the highest LOD produced by the Delaunay algorithm. For example, if **Group Size in Latitude Posts (X)** = 8 and **Group Size in Longitude Posts (Y)**= 8, each group in the highest resolution LOD corresponds to an 8 x 8 area of posts. Each lower level resolution LOD contains twice as many posts in x and y as the next higher resolution LOD.

Small values of posts/group result in many more polygons than large values because each group's corner posts are included in the triangulation, and because the face limit on triangulation is defined per group.

If only one LOD is being created, the area is not chopped into rectangular groups (to reduce the number of polygons). In this case polygons are sorted into groups based on the center of the polygon.

Z Scale Multiplier

Scales each post value as it is converted to a database coordinate. For example, a Z scale of 3.000 changes the z component of each vertex to three times its original elevation, exaggerating the elevation differences in the original terrain.

Land Posts Color Threshold

Specifies a minimum percentage of non-zero posts that must be present for the terrain color to be assigned to a polygon.

If the number of non-zero posts is less than this value, the Fixed Elevation color is assigned to the polygon. If the number of non-zero posts is greater than this value, the normal terrain color is assigned.

The intent of this control is to let you substitute the Fixed Elevation color (usually blue) for polygons that are mostly water.

Disable Edge Matching

When edge matching is disabled, the group edges are not constrained by the lower levels of detail. This option should only be used with post processor edge matching software.

Create Quad Tree LODs

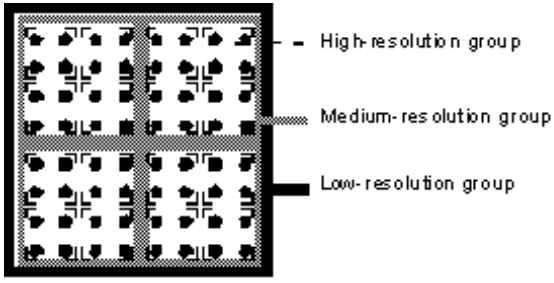
Controls the hierarchy of each square of terrain in different levels of detail.

If **Create Quad Tree LODs** is cleared, each square area of terrain is represented by a single group in each LOD. In this case, the amount of resolution is determined by **Maximum Faces/Group**.

If **Create Quad Tree LODs** is set, every higher resolution LOD has four groups that represent the area covered by a single group in the next lower resolution LOD.

Consider the following example in which three levels of detail are being created, **Group Size in Latitude Posts (X)** = 2, **Group Size in Longitude Posts (Y)**= 2, and **Create Quad Tree LODs** is set. The highest resolution LOD contains one group for every 2 x 2 square of posts, the next lower resolution LOD contains one group for every 4 x 4 square of posts, and the lowest resolution LOD contains one group for every 8 x 8 square of posts.

> **Note:** The value 2 was chosen for simplicity; generally, groups are larger.



For more information, see “Quad Tree LODs in Delaunay Terrain Conversion” on page 1026.

Delaunay Error Log

Creates a text file listing the name of each database created in one batch with the Delaunay algorithm, followed by the maximum and average Delaunay error for each group in each LOD. The error log has the same name and directory path as the original DED file, plus the extension .log.

```
AREA BLOCK: testN37_46_3W122_31_20.flt
   g3:   Delaunay Error:   Level: 2 Maximum: 509.200897  Average: 121.778015
   g4:   Delaunay Error:   Level: 1 Maximum: 503.773529  Average: 187.857330
   g5:   Delaunay Error:   Level: 0 Maximum: 34.135006   Average: 43.011360
   g10:  Delaunay Error:   Level: 1 Maximum: 0.000000 Average: 0.666667
   g11:  Delaunay Error:   Level: 0 Maximum: 0.000000 Average: 0.666667
```

Rect. High Res

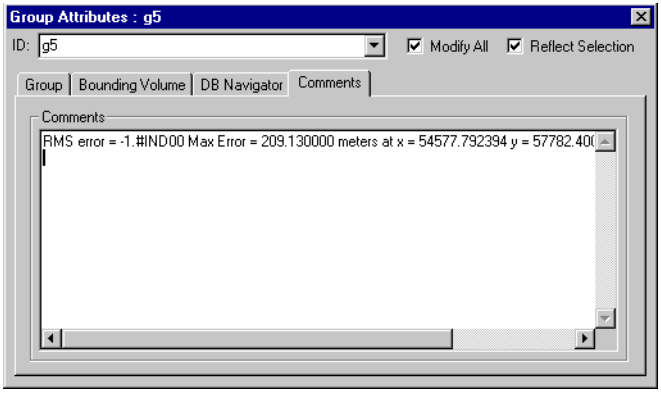
When **Create Quad Tree LODs** is set, **Rect. High Res** forces the creation of groups in the highest LOD that have straight boundaries and are exactly one-quarter of the area of the next lowest resolution group.

Normally when calculating the highest LOD, **Create Quad Tree LODs** divides the area of each lower-resolution group into quarters and then places the triangulated polygons in the appropriate groups, by area. This results in high-resolution quadrants that are approximately, but not exactly, one-quarter of the area of the next lowest-resolution groups, and each quadrant has irregular boundaries that produce fewer artifacts than straight, “hard” boundaries.

When **Rect. High Res** is set, the polygons are triangulated to form straight boundaries for each quadrant in the high resolution. This increases the polygon count and can result in visible boundaries. Set this option only if your realtime system requires straight quadrant boundaries in the highest-resolution LODs.

Calculate RMS Error

Measures the amount of deviation between posts and calculates the root mean square by adding the sum of the squares of the differences in each coordinate of two vectors. The information is written to the Comments panel of the *Group Attributes* window.



Ridge and Valley Preservation

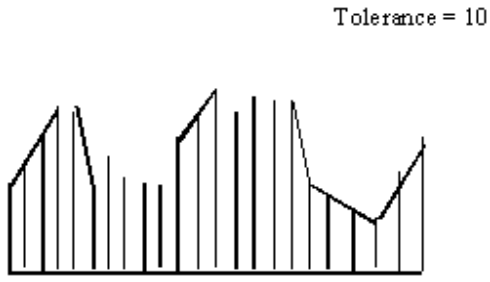
These settings specify the amount of resolution for ridge and valley detection. During **Ridge and Valley Detection**, triangulation continues until either the difference between the elevation of each unused post and the average elevation of its neighbors is less than this value or the LOD reaches the limit of **Maximum Faces/Group**.

Ridge and Valley Detection

Enables ridge and valley detection. A set of posts that defines the ridges and valleys is added to the triangulation. The longest edges are considered first, and edges on the group boundary have higher priority than interior edges.

Ridge and Valley Tolerance

Controls the amount of resolution in meters for ridge and valley detection. When the distance between post height meets or exceeds the value set by **Ridge and Valley Tolerance**, the post becomes a ridge or valley vertex.



Terrain Post Weighting Multipliers Preferences

These options fine tune the process of coastline detection during terrain generation, whether or not they are visible in the terrain window.

Use Post Weighting Multipliers

Computes the distance from each post that is not included in the triangulation to the surface of the corresponding polygon, and adds the post with the greatest distance to the triangulation. Repeats until the average distance between each unused post and the corresponding polygon is less than **Terrain Accuracy Tolerance**, or until Creator reaches the limit of **Maximum Faces/Group**.

Group Boundary Posts

Scales the posts along group boundaries. Increasing this field makes it more likely that posts on the edges of each group will be preserved in the triangulation. This helps prevent the introduction of artificial spikes along the coast.

Coastline Posts

Scales the posts along the coastline. Increasing this field makes it more likely that coastline posts will be preserved.

Quad Tree LOD Boundary Posts

If **Create Quad Tree LODs** is set, this setting scales the error for posts along the edges of the next higher LOD. Increasing this field increases the likelihood that such posts will be preserved.

Coastline Preservation

These settings help preserve detail along coastlines.

Preserve Coastline

Enables coastline preservation by testing the edge of each new polygon for intersection with the coastline and adds coastal posts to the triangulation. The longest edges are considered first, and edges on the group boundary have higher priority than interior edges. This process is interwoven with **Ridge and Valley Preservation** and **Terrain Post Weighting Multipliers**. If neither **Ridge and Valley Preservation** nor **Terrain Post Weighting Multipliers** are set, **Preserve Coastline** has few edges to follow and the coast will not be well-defined.

Land and Water Edge Weighting

Scales the edges that cross from land to water. Increasing this field makes it more likely such edges will be preserved by **Preserve Coastline**.

Group Boundary Weighting

Scales the error for posts on group boundaries. Increasing this field makes it more likely that posts along group boundaries will be preserved by **Preserve Coastline**.

LOD Polygon Count and Accuracy Controls

These settings affect the number of faces per group and accuracy tolerance for each LOD.

LOD Number

Selects each LOD for setup. The displayed value of **Maximum Faces/Group** and **Terrain Accuracy Tolerance** apply only to the selected LOD.

Maximum Faces/Group

Defines the maximum number of faces per group in each LOD. This value sets an upper limit on the number of triangles generated by the **Ridge and Valley Detection** and **Preserve Coastline** settings. If the terrain has only one LOD **Maximum Faces/Group** sets the total number of faces for the entire database.

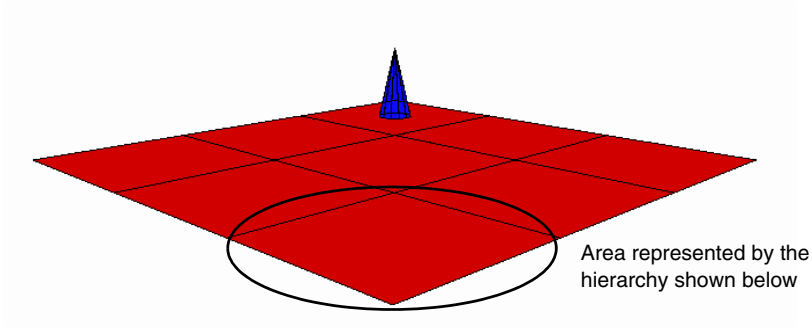
Terrain Accuracy Tolerance

Defines the accuracy tolerance for the selected LOD. Triangulation continues until the distance from each post to the corresponding triangular polygon in the terrain database is less than or equal to **Terrain Accuracy Tolerance**.

Quad Tree LODs in Delaunay Terrain Conversion

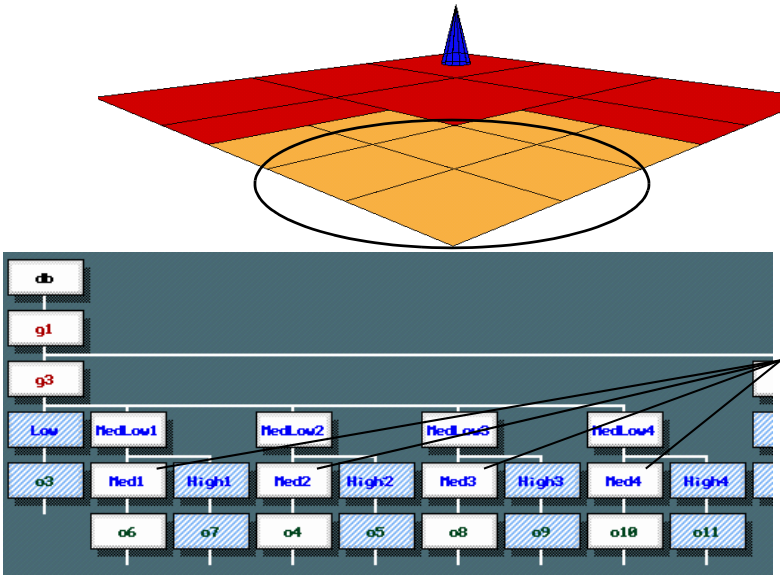
The quad tree method of calculating LODs for terrain uses the group size defined by **Group Size in Latitude Posts (X)** and **Group Size in Longitude Posts (Y)** to calculate the number of groups in the highest LOD. The number of polygons in each group of the highest LOD is controlled by **Maximum Polygons per Group**. In the second-highest LOD, four of the high LOD groups are represented by a single group, so the resolution (and the number of polygons in the new group) is one-fourth that of the highest LOD. In the third-highest LOD, four groups from the second-highest LOD are represented by a single group, and the resolution is one-fourth of the next highest LOD, and one-sixteenth of the highest LOD (so the number of polygons per group is one-sixteenth that of the highest LOD). Additional LODs are calculated in the same way.

In this example, the terrain area is represented by nine low-resolution groups.

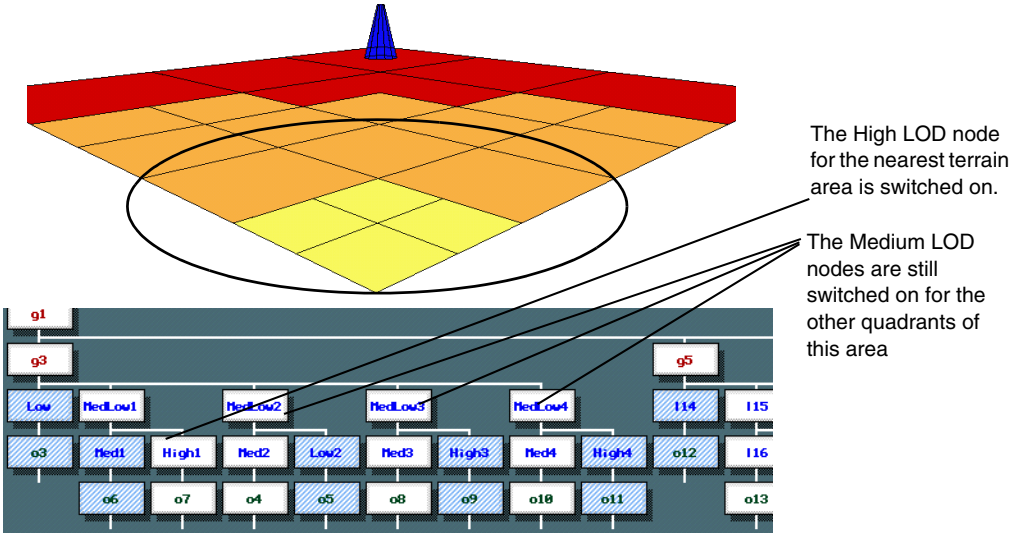


The Low node is switched on for this area. The four nodes to the right of this node represent the switched-off Medium LODs for the same area.

Moving closer to the object, each low-resolution group in the foreground switches to four medium-resolution groups.



Moving even closer, the nearest medium-resolution group is replaced by four high-resolution groups.



Terrain Culture Triangulation

Terrain Culture Triangulation (TCT) is a constrained Delaunay algorithm that extracts feature data from DFD files as it processes elevation data to create terrain polygons with features.

Features such as roads, rivers, lakes, and streams become a part of the terrain skin, rather than lying on top of it. This is accomplished by pre-projecting features – the feature data is processed first and then the terrain skin is created, so that both features and terrain are positioned at the same elevation. Post projected features will not project onto pre-projected features. For example, pre-projected roads will cut through a post-projected town.

Pre-projection is an advantage in ground-based simulations; lakes and roads lie flat with the terrain and rivers flow downhill. However, by cutting the features into the terrain, more polygons are created, so if you do not need this ground-level realism (for example, if you are creating an air simulation) other algorithms, such as Delaunay or CAT, may be more appropriate.

Generating TCT Databases

TCT databases can only be generated as a Batch process. In Batch, Creator makes a *master file* which contains external references to all the other databases created together in batch mode. The master file also contains external references to any cutout files you used while generating the terrain.

The master file gets its name from the terrain data you are processing. For example, if you are processing data from a file called wyoming.ded, the resulting master file will be called wyoming.flt. The next time you generate databases from that terrain (while in the same directory) the file name will increment to Wyoming1.flt, then Wyoming2.flt, etc.

Since the master file contains only external references to other files, modifications such as applying texture to the terrain should be made to the referenced file. Of course, external references can be converted to geometry using the **File/Convert Externals** command.

You can generate TCT databases with pre-projected features cut into the terrain, post-projected features, or project only the features without the terrain.

TCT Workflow Guidelines

When you generate terrain with TCT, you can create a terrain skin that seamlessly integrates feature data, such as roads, lakes, or buildings. In addition, TCT has an optional "Pathfinder" feature that automatically builds drivable roads, road intersections, and bridges.

TCT is available for a single LOD only in this version of Creator. Single LOD TCT functionality is intended for simulations such as ground warfare applications. The following assumptions apply:

- Only one LOD is required.
- You will test and edit source feature data after any simplification.
- Feature data at overlaps will be manually reduced if the features are overly complex.
- You will test and manually fix area block boundary and vector vertex conflicts.

Feature data is typically very dense. That is, it contains far more information than is required for your simulation. This data can be "thinned" using Rules and Actions. In a Rule, you define which data to extract from the feature data. For example, for your dirt

roads you may want to select only feature data defined as Feature Type 254 (Cart Track, Trail). This Rule is then used during terrain generation to select data that matches that description.

However, the data may still be too dense for your simulation, or it may conflict with other feature data. For this reason, you should first project the feature data without terrain, examine and correct it, and then generate your final terrain. You can detect these conflicts easily, because Creator "flags" the conflicts by highlighting overlapping features in red. Note, however, that bridges that are automatically created with Pathfinder are flagged red because, by definition, they are a linear crossing another linear. Ignore these bridges when correcting the feature data.

When you project the feature data without terrain, see "Guidelines for Correcting Feature Data" on page 1030 for tips on how to correct the feature data.

Guidelines for Correcting Feature Data

1. Any suitable source data can be imported and simplified. Save simplified source data as the real source for the database.

   Source data is converted into polygons using TCT **Project Feature Only (No Terrain Polygons)** with area block boundaries that are the same as the final test database. Identify all data errors by this function.

2. Simplify source data manually so that it lies within the realistic limitations of the TCT polygonization engine and your realtime system. Overlapping features will be highlighted in red and the data must be corrected manually. The following conditions cannot be detected by Creator, however. Data that does not meet these conditions also will need to be corrected manually.

   - No more than four vectors should meet at any one vertex.
   - No vertices can lie on an area block boundary.
   - No other intersections are allowed within the influence area of any intersection.

3. The area of influence of an intersection is defined as the expanded size of the junction. This can be seen after **Project Feature Only (No Terrain Polygons)** is performed.

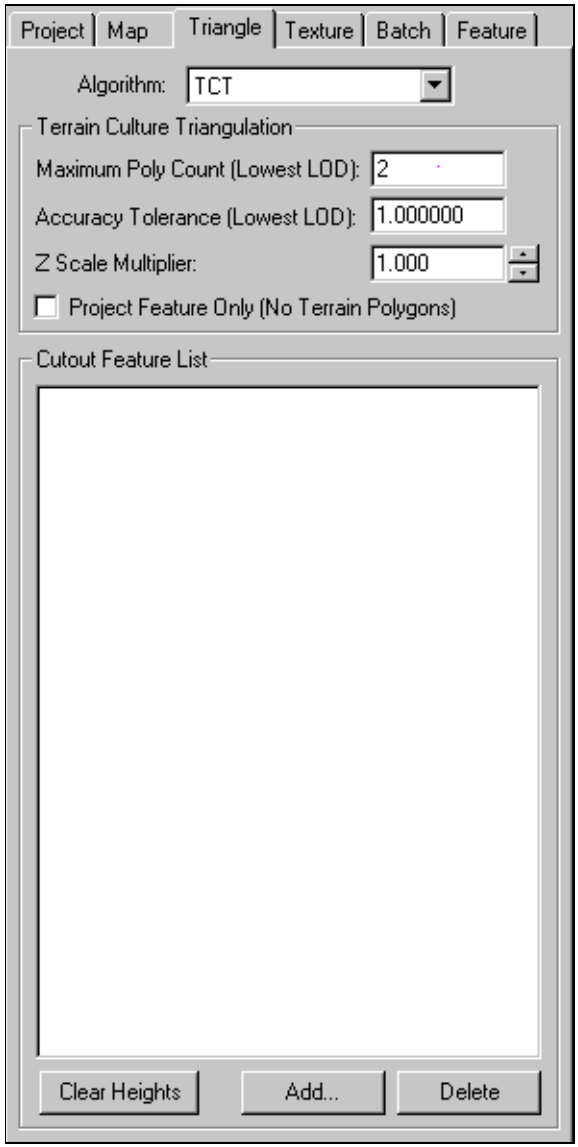
   - Linear intersections are allowed within areal features only when the influence area of any intersection does not intersect with the areal boundary.

- Linear intersections are not allowed when the influence area of any intersection intersects with any area block boundary.
- No portion of an expanded linear can intersect with an area block boundary, or with an areal.
- Two linears that do not actually intersect cannot have areas of overlap when the linears are expanded.
- The footprints of point features cannot cross area block boundaries.
- Areals within areals are not allowed unless their boundaries intersect. Limitations for number of vectors intersecting and area of influences still apply.
- Point feature footprints are treated as areals, so the same rules apply to them.
- Point feature projection should be tested without library substitution in TCT. Models should only be applied post-project, and the correlation between point feature footprints and library substitution post-project should be tested.

4. After you correct the source feature data, save it using **GeoFeature/Export/Export DFD**. The new file can then be used with full TCT terrain and feature projection.

Setting TCT Preferences

The *Terrain* window has six panels where you set preferences for terrain generation. To choose the TCT conversion method, click the Triangle panel in the *Terrain* window, and select TCT from the **Algorithm** list. The Triangle panel then displays the TCT preferences, which are described in the following section. For information about the *Terrain* window and setting other panel preferences, see “The Terrain Window” on page 988.


Project
Map
Triangle
Texture
Batch
Feature
Algorithm:
TCT
Terrain Culture Triangulation
Maximum Poly Count (Lowest LOD):
2
Accuracy Tolerance (Lowest LOD):
1.000000
Z Scale Multiplier:
1.000
Project Feature Only (No Terrain Polygons)
Cutout Feature List
Clear Heights
Add...
Delete

Triangle Panel TCT Preferences

The **Triangle** panel contain options for choosing your algorithm for generating terrain. When you select the algorithm, the panel displays the preferences available for that algorithm.

Algorithm

Specifies the method of terrain conversion. This list will include TCT only if you have installed the Terrain Pro option.

Maximum Poly Count (Lowest LOD)

Sets the total number of polygons (triangles) across the terrain surface. Cultural features are processed first, then more polygons are added until **Maximum Poly Count** is reached.

Maximum Poly Count works in conjunction with **Accuracy Tolerance** to determine the final number of polygons in the terrain. If the polygon budget is reached, Creator stops triangulating, even if **Accuracy Tolerance** is not met. The lower of the two values determines when Creator stops triangulating.

For example, when **Maximum Poly Count** is 2000, 2000 triangles form the terrain skin *and the polygons of all the cultural features*. If 1000 polygons are needed for the features, then the remaining 1000 are used to form the terrain skin. However, if 2000 polygons are needed for the cultural features, then they are processed first and the *minimum number* of triangles necessary to complete the terrain skin are created. In this case, the polygon budget would exceed the value you specified in the **Polygon Budget** field.

Accuracy Tolerance (Lowest LOD)

Calculates the Z distance to all posts above and below the triangle. If the distance is greater than the **Accuracy Tolerance** value, then it breaks up the polygon. If the distance is less than the **Accuracy Tolerance** value, Creator stops triangulating.

Accuracy Tolerance works in conjunction with **Maximum Poly Count** to determine the final number of polygons in the terrain. All triangles are tested and broken into additional polygons until either **Accuracy Tolerance** or **Maximum Poly Count** is reached, then Creator stops triangulating.

Project Feature Only (No Terrain Polygons)

This option processes cultural features without terrain. You can preview cultural features before processing area blocks, or extract features for other purposes, such as modifying them by hand and then creating a cutout file.

The **Project Feature Only** option also enables conflict checking by coloring red the polygons that conflict with each other.

> **Note:** Bridges always appear in red and do not necessarily indicate a conflict.

Cutout Feature List

The **Cutout Feature List** specifies the OpenFlight files to be used as cutout files during processing. For more information, see "Cutout Files" on page 1163.

Clear Heights

Clear Heights clears a temp file that stores the height information for features that are projected to a highest, median, or lowest z value. Features are projected to a z value when a TCT **Project to Terrain Data** action is created. You must clear the temp file when you reprocess terrain with features so that the features are not projected to the stored z values. You must also clear the temp file if you change database units in the **Header Attributes/Preferences** panel to ensure that the height values are correct.

The temp file is assigned the same name as the .ded file, for example filename.ded.temp, and is located in the same directory as the .ded file.

TCT and Levels of Detail

TCT feature data can be defined for only one LOD. However, you can use TCT in conjunction with CAT to produce terrains that have multiple LODs. These terrains are suitable for simulations that require a detailed section surrounded by less-detailed terrain.

For example, if you are creating a simulation where a helicopter lands on a pad, the pad part of the terrain needs more detail than the surrounding terrain which will be viewed only as the helicopter flies over. To get maximum detail in the pad area, you can process it using TCT and then use the resulting OpenFlight file as a CAT Cutout file when you generate the terrain that surrounds the helicopter pad. This method creates

levels of detail for the terrain which will change based on the distance from the eyepoint while the features will remain the same for every LOD.

Continuous Adaptive Terrain Conversion

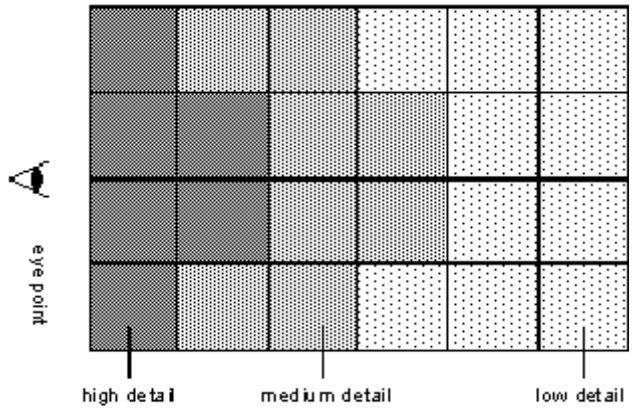
Continuous Adaptive Terrain (CAT) provides solutions to some of the traditional problems of terrain LOD management. By requiring the realtime software to do additional computation, CAT morphs terrain smoothly from one level-of-detail (LOD) to another. The CAT hierarchy is based on a triangular area which results in more efficient culling than with Delaunay or Polymesh. CAT runs only on realtime systems that use version 2.0 or later of the Performer loader or in applications based in Performer 2.0.

The approach to LOD management used by the Delaunay and Polymesh terrain conversion algorithms is to divide a large terrain area into tiles (rectangular blocks of terrain), and then switch the levels of detail belonging to each tile in or out in the realtime software to provide the best resolution for the visual scene. Because most of the computer processing is done by Creator, these methods generate databases that are displayed with low overhead in the realtime software. When the realtime software switches LOD tiles created with Delaunay or Polymesh, a visual discontinuity, known as "popping", occurs as each LOD abruptly switches with its higher or lower resolution counterpart.

CAT eliminates visible tile boundaries. LOD switching is performed on a face-by-face basis by replacing each triangular face with a corresponding set of faces from the next higher LOD. This allows terrain to morph smoothly from one LOD to another without popping when geometry moves from one LOD to another. The drawback is that the realtime software uses more overhead to process the databases than with Delaunay or Polymesh.

CAT is primarily intended for use with geospecific texture patterns.

The figure below shows a terrain area displayed in tiles of high, medium, and low resolution.



To produce the appearance of a continuous surface, the bordering edges of each tile must match those of its neighbors. Because the neighboring tiles may be at different levels of detail, every LOD must have exactly the same edges along its border. Thus, there can be no more vertices along any border than there are in the lowest resolution-LOD, which creates the appearance of walls running through the terrain.

Edge matching presents a trade-off in the design of the terrain database:

- Large tile dimensions minimize the appearance of walls, but limit the granularity of LOD switching.
- Small tile dimensions give greater control of LOD switching, but create more walls.

Setting CAT Levels of Detail

Levels of detail (LODs) are sets of faces that provide varying amounts of resolution for each terrain area. LOD switching allows the realtime software to display areas of terrain that are close to the eyepoint at high resolution and areas that are farther away from the eyepoint at low resolution. As the eyepoint moves through the database, the realtime software switches each LOD in or out dynamically.

CAT LOD Overview

The point at which a lower resolution LOD is replaced with the next higher level of resolution is called the switching distance, and is specified by the **Switch Out** and **Switch In** LOD attributes. The **Switch Out** attribute for each lower resolution LOD matches the **Switch In** distance of the next higher resolution LOD when generated with Basic Terrain Conversion or the Terrain-Pro option. The *Terrain* window's **Conversion Prefs** settings determine the **Switch In** and **Switch Out** points and the hierarchy of each terrain LOD in the final database.

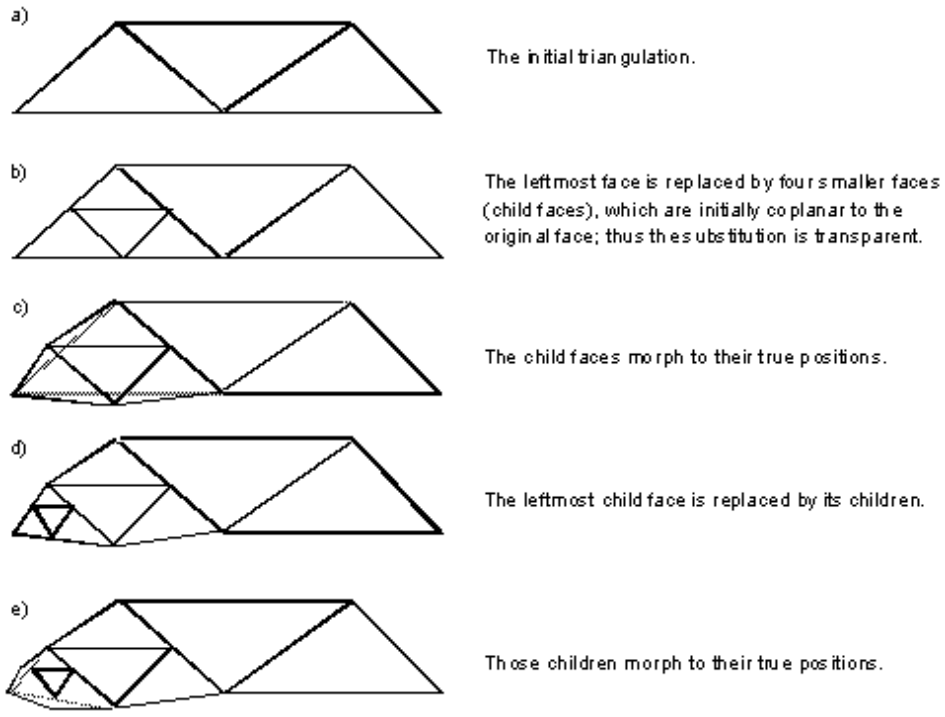
LOD Switching in CAT

CAT eliminates visible tile boundaries and delivers seamless LOD switching. The amount of detail is variable in realtime and can be adjusted to focus the detail at any location across the terrain.

CAT has the following advantages over terrain produced with Delaunay or Polymesh:

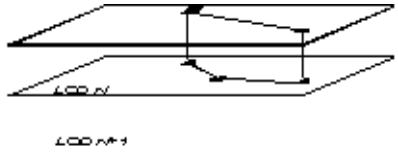
- Instead of dividing terrain into tiles, LOD switching is performed on a face-by-face basis by replacing each triangular face with a corresponding set of faces from the next higher LOD.
- Morphing between the faces of different levels of detail prevents LOD popping. For information on morphing, see "Levels of Detail" on page 707.

The following figure illustrates CAT LOD switching. The eyepoint starts high above the lower-left corner of the terrain and descends as time progresses. As the eyepoint approaches the lower-left corner of the terrain, more detail is added. This process repeats until the display contains the desired amount of detail.



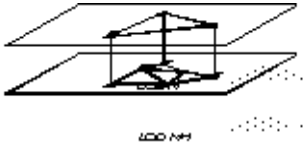
To display a terrain database using CAT, the realtime software weaves together a complete surface by selecting faces from different levels of detail. To make this possible, CAT creates levels of detail according to the following rules:

- The coarsest level of detail (LOD 0) can have any shape.
- Each vertex in LOD *n* is present in LOD *n+1*.
- Each edge in LOD *n* is either duplicated in LOD *n+1*, or split into two edges that connect to the end points of the corresponding edge in LOD *n*. These types of edges are called *child edges*. Child edges preserve the connectivity and general shape (if somewhat bent) of the parent edge.



Corresponding Parent and Child Edges

- Each face in LOD *n* is either duplicated in LOD *n+1*, or split into at most four new faces. These types of faces are called *child faces*.

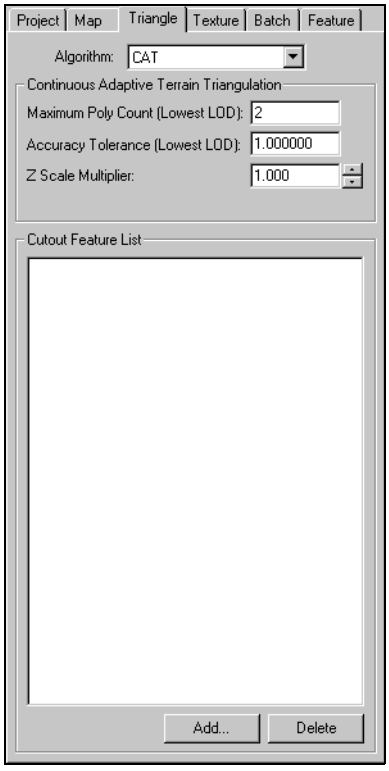


Parent and Child Faces

Combining a database that follows these rules with vertex morphing lets the surface evolve gradually during realtime, adding detail near the eyepoint without breaking the continuity of the surface.

Setting CAT Conversion Preferences

To generate a CAT database, select the **CAT** conversion algorithm, and use the preferences described below.



CAT Preferences

This section describes each of the preferences for generating a Continuous Adaptive Terrain.

Maximum Poly Count (Lowest LOD)

Displays the number of triangular polygons that will be generated for the lowest LOD.

Accuracy Tolerance (Lowest LOD)

Controls the amount of resolution in each LOD. As in the Delaunay algorithm, detail is added until the difference between the elevation of each unused post and the average elevation of its neighbors is less than **Accuracy Tolerance**.

Z Scale Multiplier

Scales each post value as it is converted to a database coordinate. For example, a **Z Scale Multiplier** of 3.000 changes the *z* component of each vertex to three times its original elevation, exaggerating the elevation differences in the final terrain.

Cutout Feature List

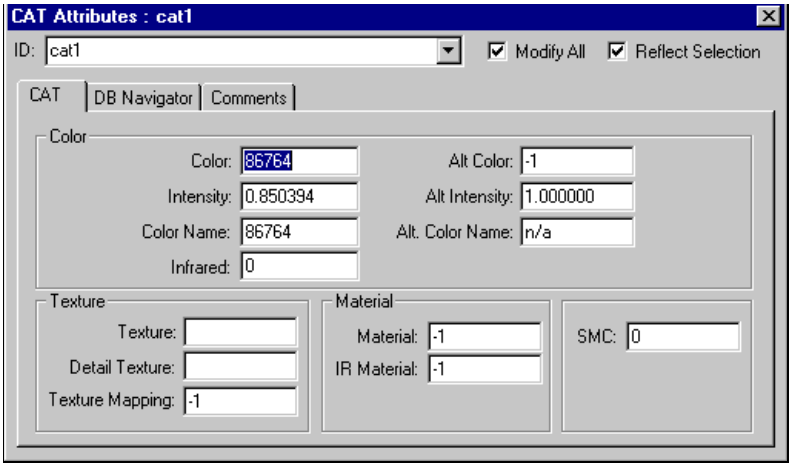
Specifies which OpenFlight files are used as cutout files when generating the terrain. Cutout files must have a terrain polygon (a polygon with the **Terrain Flag** checkbox set in the *Face Attributes* window). For more information about setting up and using cutout files, see "Cutout Files" on page 1163.

CAT Node Attributes

CAT attributes are a subset of face attributes. CAT faces are displayed in Creator using the normal face drawing attributes such as color, texture, and shading. However, during realtime simulation, the display parameters of terrain faces are determined by the attributes of the (single) CAT node that the terrain faces are attached to. Because of this, all terrain faces have the same color, texture, and other attributes.

> **Note:** Assigning color, texture and material attributes to faces in a CAT database are strictly for display purposes within Creator.

For best results, CAT should be used with a geospecific texture pattern that maps the appearance of the entire terrain.



CAT Attributes

Color

Contains the color palette index of the color assigned to the terrain faces.

Intensity

Contains the intensity multiplier (range 0.0 to 1.0) for the color of the terrain faces.

Color Name

Contains the symbolic color name of the terrain faces. If the **Color Name** does not match the color palette index in the **Color** field, **Color Name** takes precedence.

Infrared

Contains an intensity code for displaying the face in Infrared mode.

Alt Color

Determines the alternate color of the terrain faces. Alternate color is used to display intensity-format texture patterns.

Alt Intensity

Contains an intensity multiplier (range 0.0 to 1.0) for the **Alt Color** assigned to the terrain faces. If the value is less than 1.0, the full intensity value of the color is applied3.

Alt Color Name

Contains the symbolic color name of the terrain faces. If **Alt Color Index** and **Alt Color Name** do not reference the same color in the palette, **Alt Color Name** takes precedence.

Texture

Contains the Texture palette index of the texture applied to the terrain faces. If no texture has been applied, this field contains a value of -1.

Detail Texture

Contains the Texture palette index of the detail texture applied to the terrain faces. If no texture has been applied, this field contains a value of -1.

Texture Mapping

.Contains the index value of the selected texture mapping in the Texture Mapping palette If no texture has been applied, this field contains a value of -1.

Material

Contains the index value of the material assigned to the terrain faces. If no material has been applied, this field contains a value of -1.

IR Material

Defines the material assigned to this face in Infrared mode.

SMC

Contains the surface material code applied to the terrain faces.

Adaptive Mode



The **Adaptive** mode creates and selects an adaptive attribute that can be attached to any level of the database hierarchy. The Adaptive attribute enables the geometry of its parent node to adapt to Continuous Adaptive Terrain (CAT) by moving up or down as

the underlying terrain morphs between LODs. This attribute allows cultural features to adapt to a morphing terrain system.

The point of attachment in the database hierarchy determines the behavior of the Adaptive attribute. For example:

- If an Adaptive attribute is attached to a house at the group or object level, the house will move up or down along a vector, using the behavior specified by the Adaptive attribute parameters. The terrain will morph beneath the house in the realtime system.
- If Adaptive attributes are attached to vertices along the bottom of a house, the vertices will move up or down along a vector to close the gap between the lower edge of the house and the terrain as the terrain morphs between LODs.

Each Adaptive attribute has a switch range, allowing adaptive movement to be turned off when viewed from a distance. Note that this effect cannot be viewed from within Creator.

To Create an Adaptive Attribute Attached to the Center Base of a Geometry

Step 1. Set a database node as parent.



Step 2. Select **Adaptive** mode.

Step 3. Click **Create** in the **Geometry** toolbox. In the Graphics view, an "x" representing the origin of the adaptive attribute is attached at the center base of the geometry. In the hierarchy view, it is represented as a small square in the upper right corner of the node.



To Create Adaptive Attributes Attached to the Vertices of Geometry

Step 1. Set a database node to parent.

Step 2. Select the desired vertices.

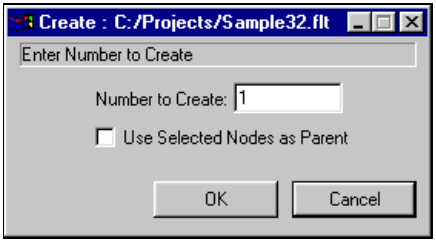


Step 3. Select **Adaptive** mode.



Step 4. Hold the Ctrl key while clicking **Create**. The *Create* dialog box appears.

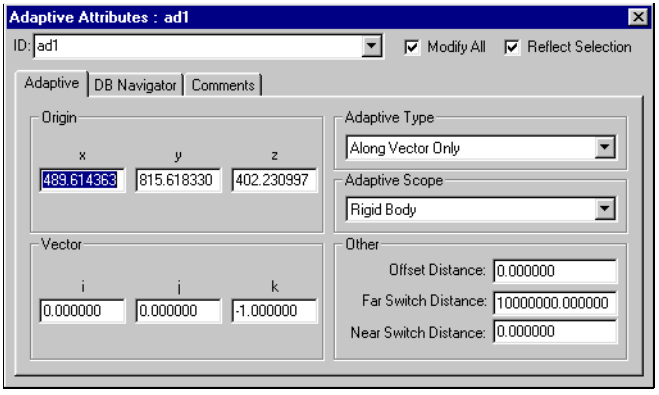
Step 5. Select the **Use Selected Nodes as Parent** checkbox.



Step 6. Click **Done**. An "x" representing the origin of the adaptive attribute is attached to each selected vertex.

To Edit an Adaptive Attribute

Step 1. Click **Adaptive** mode and double-click the adaptive attribute "x" in the Graphics view. The *Adaptive Attributes* window appears.



Step 2. Edit the attributes as required.

Adaptive Attributes

Origin (X, Y, Z)

Specifies the point on the geometry that is used by the realtime software as an endpoint that a vector intersecting the terrain is generated from.

Vector (I, J, K)

Specifies the vector used with the origin by the realtime software to generate a line that intersects the terrain.

Adaptive Type

- **Along Vector Only** translates the adaptive origin along the vector until it intersects the terrain.
- **Along Vector Normal to Terrain** translates the adaptive origin along the vector until it intersects the terrain and then rotates it normal to the terrain.

Adaptive Scope

- **Rigid Body** transforms the geometry of the parent node.
- **Draped** transforms each vertex individually.

Offset Distance

Specifies the distance of the adaptive origin above the terrain (opposite the vector direction).

Offset Distance is typically used for floating light points above the terrain.

Far Switch Distance

Specifies the distance of the eyepoint from the adaptive origin at which the adaptive movement becomes active.

Near Switch Distance

Specifies the distance of the eyepoint from the adaptive origin at which the adaptive movement becomes inactive.

The Texture Panel

The Texture panel contains options for automatically applying a geospecific or indirect texture pattern to the terrain during processing. You can also save the contour map as a georeferenced image file. This image file can then be applied as a geospecific or indirect texture when processing your elevation data, or applied to an existing terrain file using the **GeoPut Texture** tool.

To create a georeferenced image:

Step 1. In the *Terrain* window, click the **Texture** tab to display terrain texture preferences.

Step 2. Set the image size by entering values in the **Texels X** and **Texels Y** fields. Remember that to correctly map a georeferenced texture, its size must be a factor of 2 (16, 32, 64, 128, etc.).

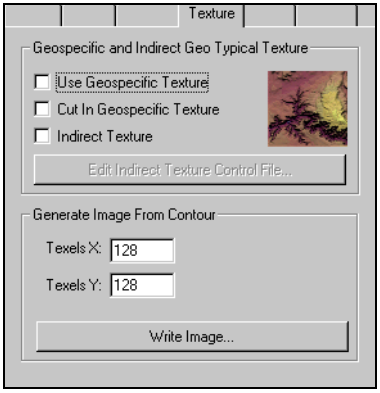
Step 3. Click **Write Image**. The *Write Contour Image to File* dialog box opens.

Step 4. Enter a file name in the **File name** field, select a directory if necessary, and then click **Save**. The file is saved in RGB format.

Step 5. Load the image into the Texture palette.

Texture Panel Preferences

Texture panel preferences specify the geospecific or indirect texture pattern to apply to a terrain, and creates a geospecific image of the contour map. For more information about applying geospecific and indirect textures, see "Applying a Geospecific Texture" on page 1191 and "Using Indirect Texture" on page 1185.



Use Geospecific Texture

Specifies that the chosen texture has longitude and latitude coordinates associated with it.

Cut in Geospecific Texture

Slices faces in the database along the boundaries of the geospecific texture. This preference prevents the texture from only covering part of a face.

Cut In GeoSpecific Texture is not available for CAT.

Indirect Texture

Maps a bitmapped image file to specific areas of the terrain.

Set the **Indirect Texture** box, then right-click the **Indirect Texture** sample box and select the indirect texture pattern from the Quick palette.

Edit Indirect Texture Control File

Specifies the texture, color, and material properties of the terrain using a bitmapped image and an ASCII control file that keys each color in the image to a specific group of properties. Save the ASCII file in the same directory as the indirect texture. Use the same root name that you assigned to the indirect texture, and assign a suffix of indirect. For example, if you named the indirect texture texture_code.rgba, name the ASCII file texture_code.indirect.

Generate Image From Contour

Saves the contour map as a georeferenced image file in RGB format.

- **Texels X** is the image size (in texels) along the x axis.
- **Texels Y** is the image size (in texels) along the y axis.
- **Write Image** saves the image as an RGB file in the size determined by Texels X and Texels Y.

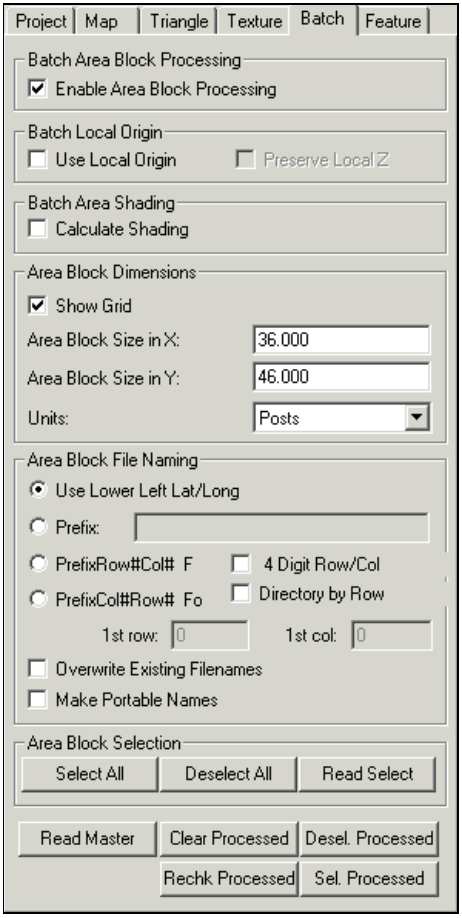
The Batch Panel

The Batch panel contains the settings for processing elevation data as area blocks. You use this panel to set the size of the area blocks, select the area blocks to process, toggle an area block grid on and off, and define how the area blocks are to be named when they are processed. You can choose to apply a local origin to each area block, which is useful for processing large area databases.

When you process terrain as a batch, your Batch panel settings are automatically saved in the same directory as the .ded file in a file named <ded file name>1.bpf. You can move or copy this file to another name in a different directory, if you wish, and use it as part of a project whenever you want to use the same settings.

Batch Panel Preferences

Batch preferences control the definition, selection, and processing of area blocks.



Enable Area Block Processing

Enables terrain area block batch processing.

Use Local Origin

Assigns a local origin to each area block tile that sets the center of the tile's bounding box is 0,0. For large area databases, this results in much smaller floating point values and facilitates smoother and faster switching.

Preserve Local Z

Forces the local origin to use Z=0. This option can be used for post processing operations that ignore origin offsets.

Calculate Shading

Calculates and saves vertex normals during the processing of selected area blocks, using the default light source. (Gouraud vertex colors are calculated whether or not **Calculate Shading** is enabled.) In the new database, the Light mode attribute of each face is set to **Dynamic**. Shading databases during batch processing ensures that shading matches across database boundaries.

Show Grid

Controls the display of a grid showing the position of each area block in the gridded terrain post file. The grid size is set by **Area Block Size in X** and **Area Block Size in Y**.

Area Block Size in X (and Y)

Defines the size of each area block in the gridded terrain post file. Size can be defined in **Posts** or **Minutes** (of latitude/longitude). The **Area Block Size** can be seen in the **Gaming Area** when **Show Grid** is selected.

Units

Defines the units used for each area block. Choose either **Posts** or **Minutes** (of latitude/longitude).

Area Block File Naming

Specifies a naming convention for the new terrain databases. Terrain databases are named either with the lower left latitude and longitude numbers or the **Prefix** name that you specify.

Selecting **Use Lower Left Lat/Long** names each database with the latitude and longitude of its southwest corner, for example N32_56_8W32_56_8.flt (this will be the origin of the new terrain database). If a generated name conflicts with an existing database, Creator appends version numbers (for example, N32_56_8W32_56_8v1.flt).

Selecting **Prefix** names each database with the specified prefix and adds a sequential number starting at 0 (zero), for example test0.flt, test1.flt, test2.flt. If a generated name conflicts with an existing database, Creator skips ahead to the next available number in the sequence (for example test1.flt, test2.flt, test5.flt). You can also specify a directory path where the terrain databases will be placed, for example, c:\temp\test1.flt(Windows), or /usr/temp/test1.flt (IRIX). If the path is not valid, the batch process is aborted and an error message is returned.

> **Note:** This path does not affect the master file that is created with batch jobs. The master file will be placed in the directory you choose later in the *Browse for Folder* dialog box. See "Understanding Master Files" on page 1173.

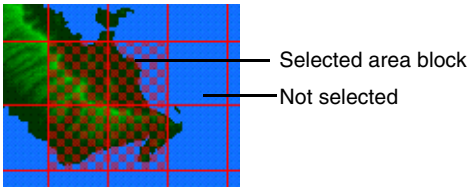
- **Prefix/Row Number/Column Number** names each file with a specified prefix, the row number, and the column number of the area block within the processed area. For example, MyFileRow001Col0030.
- **Prefix/Column Number/Row Number** names each file with a specified prefix, the column number, and the row number of the area block within the processed area. For example, MyFileCol0000Row004
- **4 Digit Row/Col** names each file with a specified prefix, a 4-digit row number, and 4-digit column number of the area block within the processed area. For example, Row1000Col0002. Set **4 Digit Row/Col** after setting **Prefix/Row Number/Column Number** or **Prefix/Column Number/Row Number.**
- **Directory by Row** creates a subdirectory for each row of area blocks, which is stored in the specified output directory. Each subdirectory is named with the row number of the area blocks. Set **4 Digit Row/Col** as well to assign a 4-digit row number.
- **1st Row** is the row number assigned to the first area block in the lower left corner; the default value is 0. This option can only be used when either **Prefix/Row Number/Column Number** or **Prefix/Column Number/Row Number** is selected.
- **1st Column** is the number assigned to the leftmost column; the default value is 0. The values entered can be either positive or negative. When 1st row and 1st column values are not specified, the area block in the lower left corner is

considered Row0, Column0. This option can only be used when either **Prefix/Row Number/Column Number** or **Prefix/Column Number/Row Number** is selected.

- **Overwrite Existing Filename** re-uses the existing file names and overwrites those files' information.
- **Make Portable Names** writes out the files in lowercase letters and inserts an underscore (_) where a space exists in the file name. The file can then be ported between Windows- and UNIX-based systems.

Select All

Selects all area blocks for processing. To select an area block, click it with the left mouse button. Selected area blocks are overlaid with a red diagonal mesh. To deselect an area block, click it again.



Deselect All

Clears the selection of all area blocks.

Read Select

Opens the terrain databases generated from the selected area blocks. After generating the terrain databases, select an area block or blocks and click **Read Select**.

Read Master

Reads the master file to open all generated terrain databases. After generating the terrain databases, click **Read Master**.

Clear Processed

Clears the shading of all processed area blocks.

Deselect Processed

Deselects all processed area blocks that are selected.

Recheck Processed

Checks for generated terrain database files in the specified output directory and shades the relevant area blocks.

Select Processed

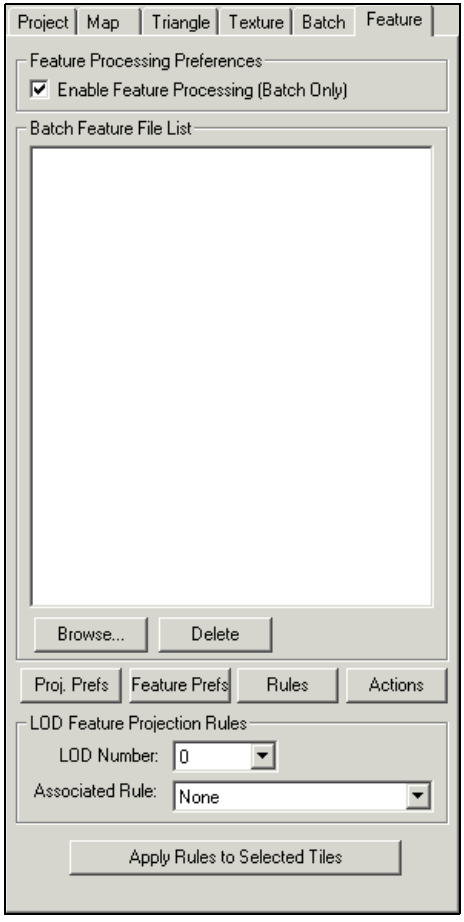
Selects all processed area blocks.

The Feature Panel

The Feature panel enables features to be projected when the terrain is generated as a batch process.

Feature Panel Preferences

The **Feature** panel specifies the DFD files from which to extract the feature data and sets the rules and actions to be applied to each LOD.



Enable Feature Processing (Batch Only)

Select this to enable batch feature processing. If this checkbox is dimmed, click **Batch** and set **Enable Area Block Processing**. Feature processing only works in batch mode.

Browse

Opens a file browser so that you can search for and select a DFD file for batch feature processing.

Delete

Deletes the currently selected DFD file.

Projection Preferences

Opens the *Projection Preferences* dialog box (see "Setting Global Projection Preferences" on page 1144 for more information on setting Library Substitution and Texture preferences for projected features).

Feature Preferences

Opens the *Feature Preferences* dialog box (see "Setting Feature Preferences" on page 1130 for more information on assigning preferences to control how features are projected into the database).

Rules

Opens the *LOD Rule List* dialog box (see "Creating Rules" on page 1153 for more information on setting rules for batch culture processing).

Actions

Opens the *Action List* dialog box (see "Creating Actions" on page 1146 for more information on setting actions for batch culture processing).

LOD Number

The number of the level of detail to which the feature projection rule will be applied.

Associated Rule

The rule to be applied to the LOD.

Apply Rules to Selected Tiles

Applies associated rules that are set in the *Rule List* window to selected area blocks.

Generating the Terrain

When you have set all your terrain conversion preferences you are ready to convert the terrain data into an OpenFlight database. You can process the terrain as a single database, or as a batch process that creates multiple databases. The following procedure shows non-batch processing. If you are generating your terrain as a batch process, see "Batch Processing" on page 1171 for information about creating terrain as a batch process.

Creating a Terrain

Step 1. Choose **Terrain/New Project**, and select a DED file to import.

Step 2. Set up the colors, textures and materials that will be used in the new database.

Step 3. Select the area to process in the **Gaming Area**.

Step 4. Select the **Project** tab and specify an **Output Directory** path, if you want. Click **Save Project** to create a project file. Path names to the DED file, attributes, and preferences are all saved in the project file.

Step 5. Set the **Map**, **Triangle**, and other conversion preferences.

For help on setting the preferences for each algorithm, see "Setting Polymesh Conversion Preferences" on page 1012, "Setting Delaunay Conversion Preferences" on page 1019, "Setting TCT Preferences" on page 1031, and "Setting CAT Conversion Preferences" on page 1039.

Step 6. Select **Batch** in the *Terrain* window.

Step 7. Clear the **Enable Area Block Processing**. This allows you to process a single terrain file.

Step 8. Click **OK** or **Apply**. **OK** closes the *Terrain* window. **Apply** leaves the *Terrain* window open, so you can easily open the processed terrain databases using **Read Select**.

Step 9. Creator displays the *Write Terrain Database File* dialog box. If you specified an **Output Directory** path (in the **Project** panel), the directory will show in this dialog box. You can, however, select another drive or directory.

Step 10. Click **Save** to save the terrain file with the default name (based on the DED file name). Or enter another file name and then click **Save**.

Step 11. If you create and save a project file, you can open the project file instead of re-importing a DED file and selecting all of the attributes, palettes, and preferences again. To open the project file on Windows, you can either select **Terrain/Open Project**, or double-click the project file icon. On IRIX, you can either select **Terrain/Open Project**, or enter creator filename.prj on the command line.

Terrain Tips

The following items are things to consider for creating efficient terrain databases.

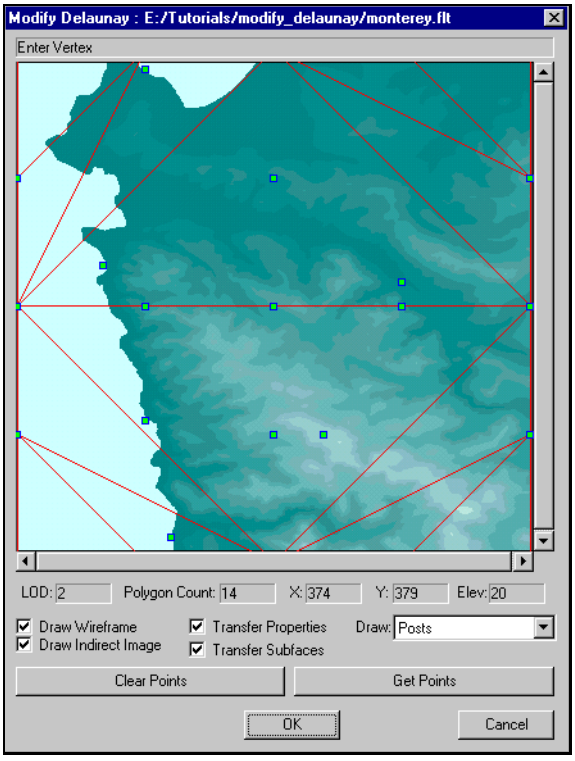
- Determine the number of polygons that the realtime system can display in a frame. This can only be done by experimenting with small areas and running the results on the realtime system.
- Choose a triangulation algorithm that is appropriate for your realtime system.
- Determine the proper group and area block size. Group size is determined by the time your realtime system spends culling and drawing. It should spend approximately the same amount of time culling the data to be viewed as it does to render the result to the screen. Area block size is determined by the amount of RAM on the computer you are using to generate the terrain. The less swapping of memory, the more efficient the processing time.
- Determine your scenario requirements.
- If you are using Delaunay, always use **Error Correction**. Use **Coast Detection** if you have coastline.

Modifying a Database

You can use the **Terrain** menu items **Modify Delaunay**, **Lat/Long to X/Y**, and **Convert Database Projection** to modify a terrain database. You can also the **Edit** menu item **Generate Mesh** to combine similar Face nodes in your database hierarchy into a single Mesh node for fast processing.

Modify Delaunay

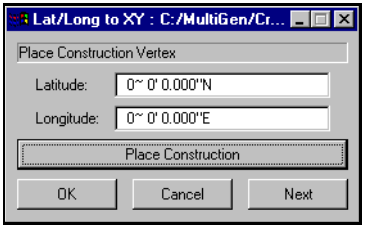
Modify Delaunay imports the post format file originally used to create a terrain database. You can manually retriangulate the terrain by adding new points that form the vertices of new terrain polygons. **Modify Delaunay** is especially useful for adding points to eliminate LOD "popping" or to enhance areas of high detail or interest.



For complete information about using **Modify Delaunay**, see the “Modify Delaunay” on page 960.

Lat/Long to X/Y

Lat/Long to X/Y opens a dialog box that allows you to specify a coordinate’s latitude and longitude and places a construction vertex in the database at the corresponding location.



Lat/Long to X/Y Preferences

In the *Lat/Long to X/Y* dialog box, enter the latitude and longitude of the construction vertex and click **Place Construction** to put it in the database.

Latitude/Longitude

The latitude and longitude of the construction coordinate.

Place Construction

Places a construction vertex at the specified coordinate.

Next

Allows you to place the next construction point.

Convert Database Projection

Convert Database Projection changes the map projection and ellipsoid type of a database. (For a description of database projections supported by Creator, see “Map Projection Types” on page 1006.)If you are using **Convert Database Projection** for a series of OpenFlight files created with the Batch option, you must also convert the master file after converting each of the files in the batch.

Convert Database Projection Preferences

The following projection types are supported.

Projection

- **Flat Earth**

  Uses latitude at the origin to derive a single convergence factor for adjusting every *x* value. This results in a rectangular database.

- **Trapezoidal**

  Calculates a new convergence factor for every latitude. Databases produced with this database projection are trapezoidal.

- **Round Earth**

  Generates a database which is approximately spherical.

- **Lambert Conic Conformal**

  Creates a database using the Lambert projection. This requires you to supply two lines of latitude using the **Northern Latitude** and **Southern Latitude** fields.

- **UTM**

  Generates a database using a Universal Transverse Mercator (UTM) projection.

- **Geocentric**

  Creates a database where the database's origin is placed at the center of the Earth.

- **Geodetic**

  Creates a database using the geodetic projection. **Latitude** (*y*) and **Longitude** (*x*) are offsets from the origin.

Use File Projection Attributes

Uses the projection attributes, such as the UTM zone, from the converted file. If you do not set **Use File Projection Attributes**, you must enter a UTM projection number (1-60); set **Southern Hemisphere** if the database is in the southern hemisphere; and set **Ignore Database Origin** to override all of the database origins and use the origin defined by UTM.

Use Origin From File

Uses the origin in the converted file as the database origin. If you do not set **Use Origin From File**, then you must enter the **Latitude** and **Longitude** values for the origin.

Ellipsoid

Ellipsoids are mathematical models of the shape of the Earth, which bulges in the middle and is flattened at the poles. An ellipsoid is the formula which defines that shape. There are many different ellipsoids used in Cartography, but the primary ellipsoids used in US data are WGS 1972 and WGS 1984. Ellipsoids are used in conjunction with projections to convert real world locations to places on a map.

The following ellipsoid types are supported:

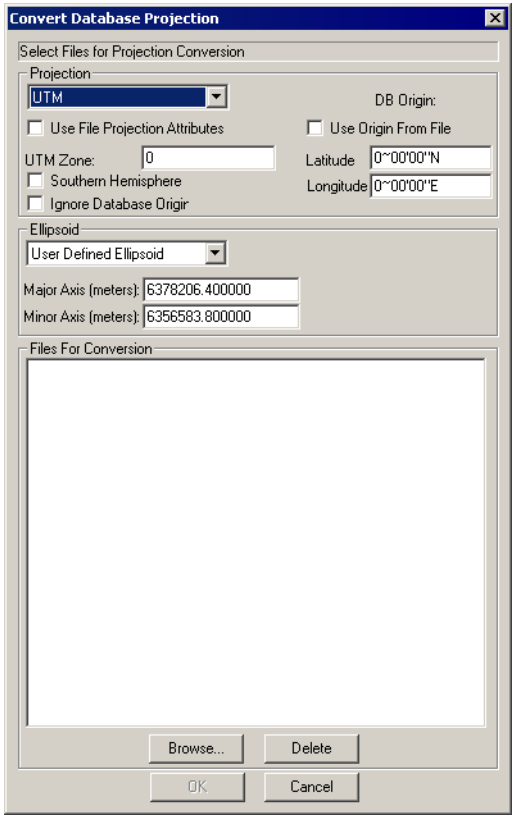
- WGS 1984
- WGS 1972
- Bessel
- Clarke 1866
- User Defined Ellipsoid, for which you enter the Major Axis and Minor Axis of the ellipsoid.

Files for Conversion

Use **Browse** to add files to this list. Use **Delete** to remove them from this list. When you click **OK**, all the selected files are converted to the new projection and ellipsoid type.

To Convert a Database Projection

Step 1. Open a terrain database file and choose **Terrain/Convert Database Projection**. The *Convert Database Projection* window appears.



Step 2. Click **Browse** and select one or more OpenFlight files in the *Open File* dialog box. To select consecutive files, select the first file in the list, then hold down Shift and select the last file. To select non-consecutive files, hold down Ctrl while clicking each file. You can also add OpenFlight files from other directories or drives.

Step 3. When you have selected all the files that you want to convert, click **Open**. The path and names of the selected files appear in the **Files for Conversion** list. To delete a file from the list, select it and click **Delete**.

Step 4. Select a new **Projection** and/or **Ellipsoid** type, as well as the projection preferences.

Step 5. Click **OK** to convert the selected databases.

29 Reading and Importing Elevation Data

Creator's Basic Terrain, CreatorPro, and TerrainPro options convert gridded elevation post data into a polygonal terrain database or a virtual world. The elevation data must be in Creator's proprietary Digital Elevation Data (DED) format. Creator includes utilities for converting source data in common formats such as NIMA (National Imagery and Mapping Agency, formerly the Defense Mapping Agency) DTED (Digital Terrain Elevation Data) and USGS (United States Geological Survey) DEM data to DED. Also included are utilities for converting for elevation data in float format to DED, and for extrapolating elevation data from the intensity values of pixels in bitmapped images. In addition, third-party convertors are available for converting data in other formats to DED.

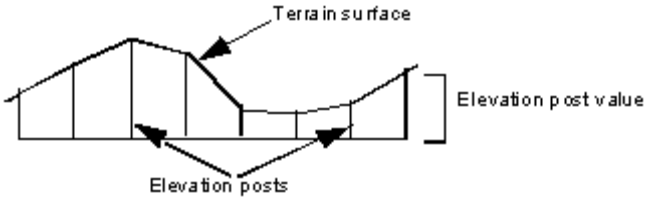
The process of creating a terrain database involves five basic steps:

- Raw elevation data is read from hard disk or compact disc (CD).
- The elevation data is converted to Creator's proprietary Digital Elevation Data (DED) format.
- The converted elevation data is imported into Creator for viewing in the *Terrain* window.
- All or part of the terrain area is selected for conversion.
- Parameters that control the conversion to polygonal data are chosen and conversion begins.

About Terrain Source Data

Creator creates an OpenFlight database from elevation data files that have been converted to DED format. *Elevation data* describes the topography of the Earth's surface. It can be obtained from USGS in DEM format, from NIMA in DTED format, or from third-party sources. It can also be created from a bitmapped image (from either a satellite photograph or created in a compatible paint program) that has been converted with a Creator utility.

Each 1 x 1 degree cell contains a grid of *elevation post values* that define the terrain's contours. For most latitudes, posts are evenly spaced every three arc seconds (92.43 meters) apart. These post values add a z value to the ground's x and y horizontal coordinates.



Generally, there are 1201 posts in a cell's latitude. The number of posts in longitude varies according to its latitude in order to keep post spacing consistent.

DED File Format Description

Creator imports gridded elevation data files in DED format only; all other elevation data files must be converted to DED before they can be imported. The file FORMAT.ded.txt, which documents the DED file format, is included in a folder within the Free Stuff directory on the Creator CD, and its contents are reproduced here:

```
/*****************************************************************\
    MultiGen Digital Elevation Data File Organization
\*****************************************************************/

DED file =

    standard header (ss_stdhdr/128) +
    statistics record (stats/32) +
    cell header (cell_hdr/40) * n +
    cell data * n

where n = no. of cells in this file.

/*****************************************************************/

#define SS_STDID "SSYS"
#define PARTNO "DMA1"
#define REVNO "V1.0"

#define M_TO_FT 3.28084
#define MIN_TO_FT6076.0/60.0/* 1 minute = 6076 ft based on nautical def.
*/
#define DEG_TO_RAD 0.017453292
```

```
/*****************************************************************/

typedef struct ss_stdhdr {/* standard file header block (128) */
    long len;                   /* size of header block in byte */
    char id [ 4 ];              /* file identification code "SSYS" */
    char part [ 8 ];        /* software part no. */
    char rev [ 8 ];         /* software revision no. */
    char create [ 26 ];     /* creation date/time */
    char update [ 26 ];     /* last update date/time */
    char spare [ 52 ];      /* spare */
} ss_stdhdr;

/*****************************************************************/

typedef struct stats {  /* statistics record (32) */
    long ncell;                 /* no. of cell in this file */
    short minz, maxz;       /* min/max z of the entire file */
    char spare [ 24 ];      /* spare */
} stats;

typedef struct cell_hdr {/* for each DMA cell on disk (40) */
    float latstart, latend; /* lat start end */
    float longstart, longend;/* long start end */
    float deltalat, deltalong;/* post spacing in seconds * 10.0 */
    float nptlat, nptlong;/* no. of points in lat or long */
    float deltax, deltay;/* actual distance between points in feet */
} cell_hdr;

/* cell_data structure:
    none; post elevation data stored in type "short". */

/*****************************************************************\
    partial dump of sample ded file with 1 cell...
\*****************************************************************/

 0000000  0000 0080 5353 5953 4445 4431 0000 0000  ~~~~SSYSDED1~~~~
 0000010  5631 2e30 0000 0000 5765 6420 4e6f 7620  V1.0~~~~Wed Nov
 0000020  3233 2031 323a 3533 3a32 3420 3139 3838  23 12:53:24 1988
 0000030  0a00 0000 0000 0000 0000 0000 0000 0000  ~~~~~~~~~~~~~~~~
      *  00  0000040-0000080
 0000080  0000 0001 0224 0b16 0000 0000 0000 0000  ~~~~~$~~~~~~~~~~
 0000090  0000 0000 0000 0000 0000 0000 0000 0000  ~~~~~~~~~~~~~~~~
 00000a0  4210 0000 4214 0000 c2e2 0000 c2e0 0000  B~~~B~~~~~~~~~~~
 00000b0  41f0 0000 41f0 0000 4496 2000 4496 2000  A~~~A~~~D~ ~D~ ~
 00000c0  4375 c785 4397 e666 079c 079d 079f 07a6  Cu~~C~~f~~~~~~~~
 00000d0  07ad 07b5 07b2 07af 07aa 07a4 079f 079c  ~~~~~~~~~~~~~~~~
```

```
00000e0  079a 0798 0796 0794 0792 0790 078e 078d  ~~~~~~~~~~~~~~~~
00000f0  078c 078b 0789 0788 0788 0787 0784 0781  ~~~~~~~~~~~~~~~~
0000100  077a 0774 076f 076d 076c 076b 076c 076d  ~z~t~o~m~l~k~l~m
0000110  076f 0770 0772 0773 0775 0777 077a 077e  ~o~p~r~s~u~w~z~~
0000120  0781 0783 0780 077f 077e 077e 0780 0783  ~~~~~~~~~~~~~~~~
0000130  0788 078c 078f 0791 0791 0792 0792 0790  ~~~~~~~~~~~~~~~~
0000140  078f 078d 078b 0788 0786 0786 0786 0786  ~~~~~~~~~~~~~~~~

/*****************************************************************\
   partial dump of sample ded file with 4 cells...
\*****************************************************************/

0000000  0000 0080 5353 5953 444d 4131 0000 0000  ~~~~SSYSDMA1~~~~
0000010  5631 2e30 0000 0000 4672 6920 4a75 6c20  V1.0~~~~Fri Jul
0000020  3239 2031 343a 3133 3a35 3120 3139 3838  29 14:13:51 1988
0000030  0a00 0000 0000 0000 0000 0000 0000 0000  ~~~~~~~~~~~~~~~~
     *  00  0000040-0000080
0000080  0000 0004 fffc 059f 0000 0000 0000 0000  ~~~~~~~~~~~~~~~~
0000090  0000 0000 0000 0000 0000 0000 0000 0000  ~~~~~~~~~~~~~~~~
00000a0  4214 0000 4218 0000 c2f6 0000 c2f4 0000  B~~~B~~~~~~~~~~~
00000b0  41f0 0000 41f0 0000 4496 2000 4496 2000  A~~~A~~~D~ ~D~ ~
00000c0  4372 a01f 4397 e666 4218 0000 421c 0000  Cr~~C~~fB~~~B~~~
00000d0  c2f6 0000 c2f4 0000 41f0 0000 41f0 0000  ~~~~~~~~A~~~A~~~
00000e0  4496 2000 4496 2000 436f 65cd 4397 e666  D~ ~D~ ~Coe~C~~f
00000f0  4218 0000 421c 0000 c2f4 0000 c2f2 0000  B~~~B~~~~~~~~~~~
0000100  41f0 0000 41f0 0000 4496 2000 4496 2000  A~~~A~~~D~ ~D~ ~
0000110  436f 65cd 4397 e666 4214 0000 4218 0000  Coe~C~~fB~~~B~~~
0000120  c2f4 0000 c2f2 0000 41f0 0000 41f0 0000  ~~~~~~~~A~~~A~~~
0000130  4496 2000 4496 2000 4372 a01f 4397 e666  D~ ~D~ ~Cr~~C~~f
0000140  0000 0000 0000 0000 0000 0000 0000 0000  ~~~~~~~~~~~~~~~~
     *  00  0000150-0000a70
0000a70  0000 0000 0000 0000 0000 0000 0000 0000  ~~~~~~~~~~~~~~~~
0000a80  0000 0000 0000 0000 0000 0000 0017 0032  ~~~~~~~~~~~~~~~2
0000a90  004d 0063 0073 007e 0082 0082 007f 007b  ~M~c~s~~~~~~~~~{
0000aa0  0078 0000 0000 0000 0000 0000 0000 0000  ~x~~~~~~~~~~~~~~
     *  00  0000ab0-00013d0
00013d0  0000 0000 0000 0000 0000 0000 0000 0000  ~~~~~~~~~~~~~~~~
00013e0  0000 0000 0000 0000 0000 0000 0003 000a  ~~~~~~~~~~~~~~~~
00013f0  002f 004a 0061 007b 0080 0083 0082 007f  ~/~J~a~{~~~~~~~~
0001400  007b 0078 0000 0000 0000 0000 0000 0000  ~{~x~~~~~~~~~~~~
```

Raster To DED

The **Raster To DED** feature within Creator allows you to build Digital Elevation Data (DED) files from several different data sources. These sources include raw data files, image files, as well as a built-in fractal pattern generator.

When building a DED file from a raw data file, the file is assumed to contain regularly gridded values representing elevations in a geographic area. The file can be ASCII or binary, and the elevation values it contains can be integer (in 1, 2 or 4 byte sizes) or floating point (in 4 or 8 byte sizes).

When building a DED file from an image file, the file can be any image format that Creator can recognize (RGB, INT, JPG, GIF, etc.). The elevation values can be derived from any of the color components (red, green, blue, alpha or intensity) represented in the image.

When building a DED file from the fractal pattern generator, the elevation values are randomly generated based on a fractal algorithm. An infinite number of patterns can be achieved and you can control certain aspects of the algorithm, including water level as well as the distribution of valleys and plateaus.

Regardless of the data source you use, you can specify the geographic extents where the data is located. You can also specify the minimum and maximum elevation values for the output DED file. In this way, you can “shift“ and/or “scale“ the range of elevation values contained in the data source before saving the DED file.

Preparing to Use Raster To DED

Depending on the data source you want to use to build your DED file, you may find it helpful to do a little preparation before using this tool. When converting a “raw“ data file, any information you can gather about the file is helpful(e.g., how may rows? how many columns? etc.). For image files, you might want to verify Creator recognizes the file. For fractal patterns, no file is required so you need only prepare your imagination to recognize a fractal pattern that suits your needs! Preparing to Convert Raw Data Files

If you want to use **Raster To DED** to convert a “raw“ data file, it helps to gather as much information about the file as you can. Although **Raster To DED** can help you

"guess" about how to interpret the values in your raw file, it works much better if you know this information in advance.

- Is the file in ASCII or binary format?
- How many rows and how many columns of data are contained in the file?
- Is the data arranged by rows or by columns?
- Where, geographically, is the data located?
- What units are the elevation values measured in?
- Does the data contain "missing" or "undefined" values?
- Do the elevation values begin at the start of the file or is there some header data before the first elevation value appears?

For binary data:

- Are the elevation values represented as integer or floating point values?
- What is the storage size of each elevation value? 1-byte, 2-byte, 4-byte or 8-byte?
- Are the elevation values signed or unsigned?
- Is the data stored in Big Endian or Little Endian format?

If you do not know all this information, do not worry. Even if you know nothing about the file, **Raster To DED** can help you look for clues!

Preparing to Convert Image Files

If you want to use Raster To DED to convert an image file, you do not need to know anything about the file (as long as you know Creator recognizes your particular image file format). If you are unsure whether Creator can recognize your image file, you can check by trying to load it into Creator's Texture Palette. If it loads there, **Raster To DED** will recognize it as well. If it does not load into the Palette, do not despair, you can still try to load it as a raw file! After all, most uncompressed image files are really "raw" files with embedded "header" information that helps an application extract the "raw" color values from the file.

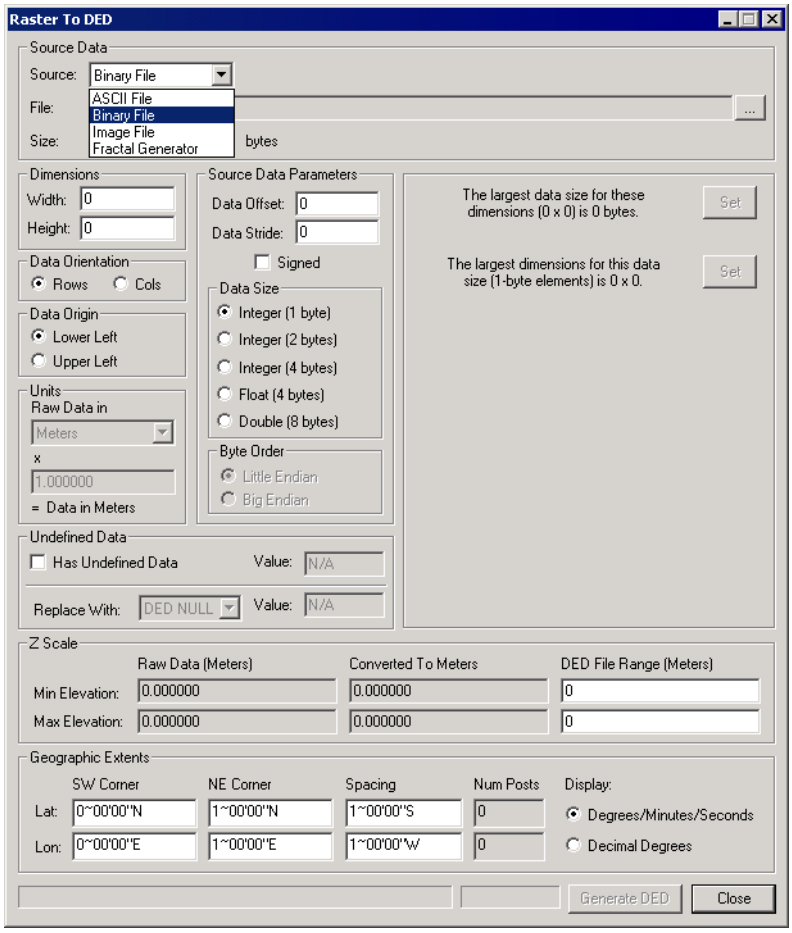
How to Use the Raster To DED Dialog

When you use **Raster To DED** to build a DED file, there is a general work flow to follow. The **Raster To DED** dialog is organized in a way that supports this work flow. The following procedure lists the steps in this work flow and, with each step, lists the

relevant areas in the **Raster To DED** dialog where you focus your attention while performing that step.

To use the Raster To DED:

Step 1. Choose **Terrain/Raster to DED** to open the dialog box.

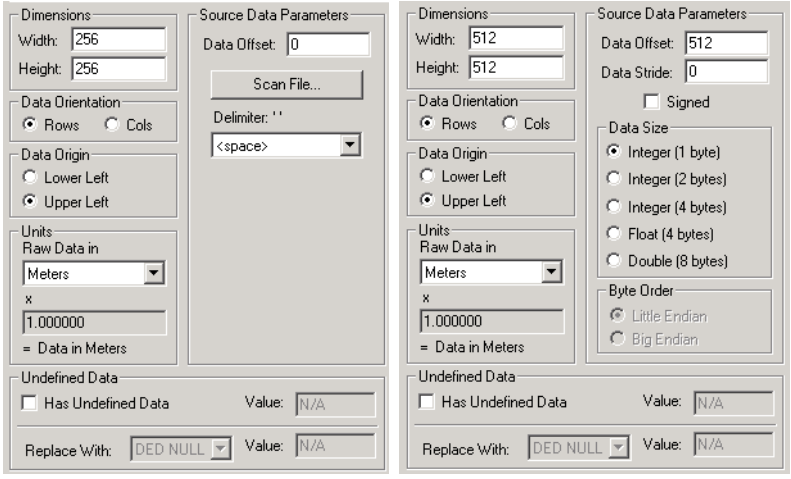


Step 2. Select a data source from the **Source** menu: **ASCII File**, **Binary File**, **Image File**, or **Fractal Generator**. (For additional details on these choices, see "Data Source" on page 1072.)

Step 3. Whether the data source is a raw (ASCII or Binary) or an image file, select the name of the file. Click the **Browse** button (**...**) to open a file browser window; use the file browser to locate your file. (For additional details, see "Data File" on page 1073.)

Step 4. If the data source is a raw (ASCII or binary) file, specify the physical layout of the data file so **Raster To DED** can know how to interpret the elevation values contained in it. To specify the physical layout, enter parameters for:

- **Dimensions** (for additional details, see "Dimensions" on page 1073).
- **Data Orientation** (for additional details, see "Data Orientation and Data Origin" on page 1074).
- **Data Origin** (for additional details, see "Data Orientation and Data Origin" on page 1074).
- **Units** (for additional details, see "Units" on page 1075).
- **File Parameters** (for additional details, see "ASCII and Binary File Parameters" on page 1076, and "ASCII File Parameters" on page 1076 and "Binary File Parameters" on page 1078).
- **Undefined Data** (for additional details, see "Undefined Data" on page 1079).



ASCII File Parameters Binary File Parameters

Step 5. If the data source is an image file, specify which color **Component** in the image is used as the elevation value in the output DED file. (For additional details, see "Image File Parameters" on page 1080.) Leave the default values for **Dimensions**, **Data Orientation**, **Data Origin**, **Units**, and **Undefined Values**, unless you have a special reason for manipulating the values in these fields.

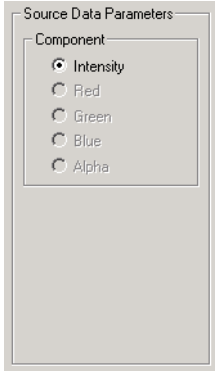
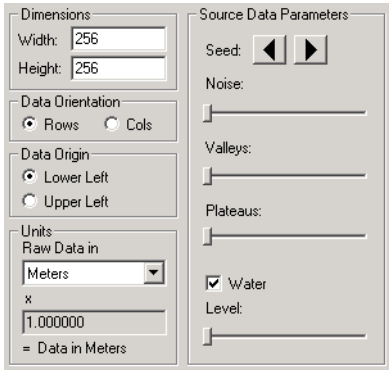


Image File Parameters

Step 6. If the data source is the **Fractal Generator**, specify:

- **Dimensions** (for additional details, see "Dimensions" on page 1073).
- The fractal parameters which govern how the fractal pattern is generated (for additional details, see "Fractal Parameters" on page 1081).



Fractal File Parameters

Leave the default values for **Dimensions**, **Data Orientation**, **Data Origin**, and **Units**, unless you have a special reason for manipulating the values in these fields.

Step 7. Preview/Validate the elevation values graphically. (For additional details, see "Preview Area" on page 1081.)

Step 8. Specify (if desired) how you want to convert the elevation values. (For additional details, see "Z Scale" on page 1082.)

Step 9. Specify the geographic extents of the output DED file. (For additional details, see "Geographic Extents" on page 1084.)

Step 10. After you have specified the necessary parameters, and are satisfied with the contour map displayed in the preview area, choose **Generate DED** to build the output DED file. (For additional details, see "Generate DED Button" on page 1084.)

Along the bottom of the dialog box there are status and progress bars that display messages and report progress during processing.

Data Source

Select the data source you want to use to build your DED file from the **Source** option menu. The choices include:

- **ASCII File** – your data source is an ASCII text file containing elevation values.
- **Binary File** – your data source is a binary file containing elevation values.
- **Image File** – your data source is an image file recognizable by Creator.
- **Fractal Generator** – you want to build a random fractal pattern.

After you select the data source you want to use, additional parameters for that data source type are displayed in the **Source Data Parameters** box. For additional information on the parameters required for the different data sources, see:

- "ASCII and Binary File Parameters" on page 1076.
- "Image File Parameters" on page 1080.
- "Fractal Parameters" on page 1081.

Note: When the data source is a raw file, you may not know whether it is an ASCII or binary file. If you are not sure whether your raw file is ASCII or binary, select **ASCII File**, then press the **Scan File** button in the **Source Data Parameters** box. When you do this, the first several lines of the file are displayed. By looking at these lines of the file, it is often very easy to tell if your file is ASCII or binary. For example, if you can read the lines in the display, chances are good that your raw data is ASCII, otherwise, you data is probably binary.

Data File

Whether the data source you select is **ASCII**, **Binary** or **Image File**, you need to select a file. To do this, press the **Browse** (**...**) button in this area. A file chooser dialog is displayed in which you can select the file to convert. Once you have selected a file, this area displays the name of that file as well as its size (in bytes). For ASCII and binary Files, this size information can be helpful in determining the number of rows and columns in your data, if you do not already know this information.

To select a different data file to convert, simply press the **Browse** button again, and choose another file.

Dimensions

If you select **ASCII** or **Binary File** as the data source, enter the number of rows (height) and columns (width) if known.

Note: If you do not know the dimensions of the raw data file, check out the Preview Area – it can help you determine these values.

If you select the **FractalGenerator** as the data source, enter the dimensions you want for your fractal pattern. For technical reasons, you are limited to square patterns whose dimensions are powers of two.

Data Orientation and Data Origin

The elevation values contained in the raw data file may be thought of as a continuous "stream" of numbers. Together, the **Data Orientation** and **Data Origin** help specify how to interpret that stream of numbers.

Possible values for Data Orientation include:

- **Rows** – The data for row 1 appears contiguously in the stream, followed by the data for row 2, row 3, etc.
- **Cols** – The data for column 1 appears contiguously in the stream, followed by the data for column 2, column 3, etc.

Possible values for **Data Origin** include:

- **Lower Left** – The first element in the stream represents the elevation value in the lower left corner of the output DED file.
- **Upper Left** – The first element in the stream represents the elevation value in the upper left corner of the output DED file.

Note: Most files are oriented by **Rows** and have their origin at the **Lower Left**. If you are not sure about these settings, select **Rows** for **Data Orientation** and **Lower Left** for **Data Origin**. You can always change these settings after you have previewed the data later in the Preview Area.

The following illustrations show how different settings for **Data Orientation** and **Data Origin** are used to interpret a simple raw data file containing a stream of 16 numbers. The illustrations show where each elevation value (represented by its numerical position) in the stream would be placed in the output DED file of dimension 4 x 4 given the different combinations of settings.

Order of elements in raw data file stream:

1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16

Position of elements in raw data file stream:

Data Orientation: Rows
Data Origin: Lower Left

13	14	15	16
9	10	11	12
5	6	7	8
1	2	3	4

Data Orientation: Rows
Data Origin: Upper Left

1	2	3	4
5	6	7	8
9	10	11	12
13	14	15	16

Data Orientation: Columns
Data Origin: Lower Left

4	8	12	16
3	7	11	15
2	6	10	14
1	5	9	13

Data Orientation: Columns
Data Origin: Upper Left

1	5	9	13
2	6	10	14
3	7	11	15
4	8	12	16

Units

By convention, elevation values stored in DED files are measured in meters. If the elevation values stored in your raw data file are measured in any other units, they need to be converted before they are output to the DED file. From the combo box in this area, select the units in which your data is measured. When you do this, the values read from the data file are automatically converted to meters. If the combo box does not include the units you need, select **Other Units** and enter the scale factor in the corresponding text field.

Note: The values in your data file are multiplied by the scale value you enter here in order to put them in "meters" before being output to the DED file.

Note: If you do not know the units in which your data is measured, select **Meters**. When you preview the data, you can check the minimum and

maximum elevations displayed in the Z Scale area of the dialog. If you know, roughly, the range of elevation values you expect in the file, the minimum and maximum elevations displayed in the Z Scale area should match what you expect. If they do not, select a different unit from the combo box until the range "looks right."

Note: You can use the scale value here for more than unit conversion. For example, assume you want to create fantasy terrain in which the elevations are exaggerated. You can do this by selecting **Other Units** and entering a desired scale factor. In this way, all elevation values are multiplied by the scale factor you enter. Try different scale factors to see the effect it has on the output DED file. There are other ways to alter the values output to the DED file, one of which is described in the Z Scale section below.

ASCII and Binary File Parameters

This is where you describe the format of the elevation values contained in the raw (ASCII or binary) data file. Depending on whether you select **ASCII File** or **Binary File** as the data source, different options become available for you to further describe the layout of your raw data. The following sections describe the additional options needed to describe ASCII File and Binary File parameters.

ASCII File Parameters

If your data is ASCII format, there are two additional pieces of information required about your file: **Data Offset** and **Delimiter**. The **Data Offset** value specifies where the first elevation value is located in your data file (relative to the beginning of the file) while the Delimiter value specifies which ASCII character is used to "separate" successive elevation values in the file.

If your file contains only elevation values (that is, the first elevation value is located at the beginning of the data file), enter 0 in the **Data Offset** field. If your data contains "header" information (that is text that appears at the start of the data file before the first elevation value), you must tell Raster To DED how many characters to "skip" over to get to the first elevation value. If you know this offset value, enter it into the **Data Offset** field directly. If you are not sure what the exact offset value is, press the **Scan File** button. When you do this, the first several lines of the data file are displayed in the Scan File window. In this window, select the line where the first elevation value is

located. When you do this, the Data Offset value is automatically calculated for you based on which line you selected. After you have selected the line where the first elevation value is located, press the **OK** button to accept this offset value or **Cancel** to ignore it and return to the Raster To DED dialog.

> **Note:** Tip: Because different computer platforms handle carriage control differently in text files (some use a single <CR>, others use the combination <CR><LF>), it is not always reliable to simply open the file in your favorite text editor and "count" the number of characters before the first elevation value to determine the **Data Offset** (you do not know whether to count "returns" in the file as one or two!). Using the **Scan File** technique described above is more reliable but is still not foolproof. For this reason, it is recommended that you check the minimum and maximum elevations displayed in the Z Scale area of the dialog after you preview ANY file. If you entered an incorrect value for **Data Offset**, this often "throws off" the minimum and maximum elevations observed in the file. If the elevation range displayed is suspect, you may have "missed" the **Data Offset** value.

In addition to the **Data Offset**, you need to specify which delimiter is used in the file. From the combo box in this area, select the delimiter that separates elevation values in your data file. If the combo box does not include the delimiter you need, select **Other** and type the ASCII character corresponding to the delimiter you want in the corresponding text field.

> **Note:** If you are not sure which delimiter has been used in your data file, press the **Scan File** button. When you do this, the first several lines of the data file are displayed. By looking at these lines of the file, it is often very easy to tell which ASCII character has been used to separate the elevation values.

If your ASCII file contains "header" information, chances are good that this header includes the dimensions of the data. In other words, the number of rows and columns represented by the elevation values in the file may appear somewhere in this header text. Since the file is ASCII text, it may be obvious when you look at the first few lines of the file. When you press the **Scan File** button, **Raster To DED** scans the file looking for these dimensions. It searches for words like "row," "height," "col," and "width" in the file. If it finds these words, it assumes the next number it finds in the file is one of the dimensions. If it finds the word "row" or "height," it guesses that the next number

it finds is the number of rows in the data. Similarly when it finds the word “col“ or “width,“ it guesses that the next number it finds is the number of columns in the data.

> **Note:** When trying to match these words, it does so regardless of “case“ and does not require full word matches. For example, “Row,“ “ROW“ or “Rows“ all match when looking for the word “row.“ Similarly, “Cols,“ “col“ and “Columns“ all match when looking for “col.“

If your ASCII file encodes the dimensions of the data with key words that are not recognized by **Raster To DED**, you can still tell **Raster To DED** how to recognize them. To do this while the **Scan File** window is displayed, select the line in the file that contains the dimension value you want. Then press the **Get** button adjacent to the dimension field you want to grab from that line. When you do this, **Raster To DED** uses the first number it finds on that line as the specified dimension. For example, consider an ASCII file whose header contains the following lines:

```
Size Y: 256
Size X: 512
```

In this file, the number of rows are encoded in the header and labeled by “Size Y.“ Similarly, the number of columns are labeled by “Size X.“ To grab the number of rows, select the line containing “Size Y“ in the **Scan File** window. Then press the **Get** button adjacent to the Rows field. When you do this, **Raster To DED** finds the number 256 within that line and uses it as the number of rows for your data. To grab the number of columns, select the line containing “Size X“ and press the **Get** button adjacent to the Columns field.

Binary File Parameters

If your data is binary format, there are four additional bits of information required about your file:

Data Offset – This value specifies where the first elevation value is located relative to the beginning of the file. This value is measured in bytes.

Data Stride – This value specifies the number of bytes to skip between successive elevation values in the raw data file. Normally, elevation values appear contiguously in a data file, in which case you would set this value to zero. If the values in your file are not contiguous, set this value to reflect the number of bytes between successive elevation values. This value is measured in bytes.

Signed – This checkbox specifies whether the elevation values are interpreted as signed or unsigned numbers.

Data Size – This specifies the storage type and size of each of the elevation values in the data file. There are five supported data sizes:

- 1-byte Integer – 8 bit integer number
- 2-byte Integer – 16 bit integer number
- 4-byte Integer – 32 bit integer number
- 4-byte Float – 32 bit single precision floating point number
- 8-byte Double– 64 bit double precision floating point number

Byte Order – This specifies how the individual bytes of the elevation values are "packed" (where the least and most significant bytes are). This is only applicable to 2, 4 and 8-byte data. The choices are:

- Little Endian – Specifies that the least significant byte is stored in the first byte of the data. The Intel 80X86 and Pentium and DEC Alpha RISC processors are Little Endian.
- Big Endian – Specifies that the most significant byte is stored in the first byte of the data. Most UNIX operating systems (for example, all System V) and the Internet are Big Endian. Motorola 680x0 microprocessors (and therefore Macintoshes), Hewlett-Packard PA-RISC, and Sun SuperSPARC processors are Big Endian.

Note: If you are not sure what to choose for any of these settings, do not be afraid to guess. The Preview Area often shows you whether you guessed correctly. And if you happen to guess wrong, it is easy to try another setting before actually converting the data to DED format.

Undefined Data

Your data file must contain some elevation value for each row and column (slot). That is, there cannot be "holes" in the regular grid represented by your data. There may be cases, however, when one or more elevation values within the grid are "unknown" or "undefined." If this is the case, there is a specific value that has been designated to mean "elevation value not known at this slot."

Note: The DED file represents "unknown" elevation values with -32768. If your data contains such "unknown" or "undefined" values, select the

Has Undefined Data checkbox and enter this special value in the corresponding **Value** field. When you specify a value in this way, **Raster To DED** ignores elevations it finds that have this value when it calculates the minimum and maximum elevations for the data file. Then when it builds the output DED file, it inserts the value -32768 at those corresponding locations in the output DED file.

Note: If you are not sure whether or not your file contains "undefined" data, deselect **Has Undefined Data**. If it turns out that your file does contain undefined data, when you preview the data, the minimum and maximum elevations displayed in the Z Scale area of the dialog will not match what you expect. This is because the "undefined data value" used is very likely to be some large (or small) value that is incongruous with the rest of the elevation values in the file. For example, if your data file contains elevations in the range 10–200 meters and you see a minimum elevation value of -9999, it is probable that your data does contain undefined data and that the undefined data value is -9999.

Image File Parameters

If you select Image File as the data source, you can specify which color component in the image is used as the elevation value in the output DED file. You can select from:

- **Intensity** – the elevation values are computed from the "lightness" of the pixels in the image
- **Red** – the elevation values are computed from the red color component of the pixels in the image
- **Green** – the elevation values are computed from the green color component of the pixels in the image
- **Blue** – the elevation values are computed from the blue color component of the pixels in the image
- **Alpha** – the elevation values are computed from the alpha color component of the pixels in the image

Note: Depending on the type of your image file, some of these choices may not be available. If your image file is RGB, for example, since it contains only red, green and blue color components, you are not able to select **Alpha**.

Fractal Parameters

If you select Fractal Generator as the data source, you can specify several parameters that control how the fractal pattern is generated.

- **Seed** – The fractal generator uses an integer "seed" value that can be varied to get different random effects. This results in literally millions of different patterns that can be generated. Use the arrow buttons here to vary the "seed" value to generate new and different patterns.
- **Noise** – Use this slider to increase or decrease the amount of noise introduced into the pattern.
- **Valleys** – Use this slider to increase or decrease the distribution of valley floors in the pattern.
- **Plateaus** – Use this slider to increase or decrease the distribution of plateaus in the pattern.
- **Water Level** – Use this slider to increase or decrease the area in the pattern covered by water.

Preview Area

This area, on the right hand side of the dialog is used for two purposes. While you are specifying the physical layout of a **Binary** data file, it displays "clues" to help you figure out the dimensions and data size of the data contained in your data file. While you are previewing your data, it displays a contour map of your data to help you decide if the settings you have specified are correct.

To help you figure out the physical layout of your **Binary** data file (this does not work for **ASCII** files), you can do either of the following:

- Enter the **Data Size** and let Raster To DED "guess" at the maximum dimensions.
- Enter the **Dimensions** and let Raster To DED "guess" at the likely Data Size.

Given the total size of the binary data file and the bits of information you do know about it, Raster To DED displays its "best guesses" for those bits you are unsure about in the **Preview Area**. If you want to try one of these guesses, press the **Set** button adjacent to that guess. When you do so, **Raster To DED** sets either the **Data Size** or **Dimension** values according to the guess.

After specifying all the information pertaining to the physical layout of your data file, you can preview its contents by pressing the **Preview** button. When you do this, if all the information you have entered about the file "makes sense," **Raster To DED** builds a contour map showing you what was read from the file.

The contour map uses a range of colors to represent different elevations. The colors used are displayed in the **Colors** swatch in the lower right corner of this area (this is only displayed when the contour map is displayed). By default, lighter shades of green indicate higher elevations, darker shades of green indicate lower elevations. You can change the colors used to draw the contours. You do this by specifying two colors. The first color (**Low** color) corresponds to the lowest elevation and the second color (**High** color) the highest. All elevations in between are calculated by interpolating between these two colors accordingly. To change the **Low** color, click in the left hand side of the **Colors** swatch. When you do so, a standard color chooser is displayed and you can pick a new color. Similarly, to change the **High** color, click in the right hand side of the **Colors** swatch. To revert to the default colors (green), click on the **Default** button adjacent to the **Colors** swatch.

> **Note:** "Undefined" or "unknown" elevations are always drawn in yellow – keep this in mind if you change the colors for the contour display.

You can change the contour preview to display a shaded relief map by selecting the **Shaded** checkbox.

By default, data values read from the data file are directly mapped to elevation values. If you are reading "image" data, this causes "brighter" colors in the image to be mapped to higher elevations. If this is not correct or you simply want to "flip" the data read from the data file, select the **Invert** checkbox. When working with "image" files, this causes "darker" colors to be mapped to higher elevations and "brighter" colors to lower elevations. When working with pure elevation values, you can use this to "flip" the elevation values and have a little fun with the data!

Z Scale

This area displays three sets of minimum/maximum elevation value pairs:

- **Raw Data** – The **Min** and **Max Elevation** values displayed in this column represent the smallest and largest "raw" elevation values found in the file – excluding elevation values that match the Undefined Data Value, if any. These values are for display only, you cannot edit them.

- **Converted To Meters** – The **Min** and **Max Elevation** values displayed in this column represent the smallest and largest elevation value converted to meters. That is: “Elevation in Meters“ equals “Raw Elevation“ multiplied by the scale factor specified in the Units section above. These values are for display only, you cannot edit them.
- **DED File Range** – The **Min** and **Max Elevation** values displayed in this column represent the smallest and largest elevation value (in meters) that are written to the output DED file. You can change these values in order to manipulate the elevation values of your data file before they are output to the DED file.

  Note: The **DED File Range** can be used to increase the height of each elevation value by a fixed amount. For example, if your data file contains elevations in the range 10–200 meters and you want to “raise“ them all, uniformly, by 100 meters, you would enter 110 and 300 in the **DED File Range** Min and Max fields. When you do this, elevation values of the data file are adjusted by a constant amount to fit within this new range you enter. Specifically, elevation values of 10 meters are adjusted to the new minimum value of 110, elevation values of 200 meters adjusted to the new maximum of 300, and all elevation values in between are increased by 100 meters.

  Note: The **DED File Range** can also be used to exaggerate elevation values in a non uniform way. Again, consider the example in which your data file contains elevations in the range 10–200 meters. In this example, however, you want to raise the peaks up to 1000 meters while keeping the low elevations relatively constant. To do this, you would enter 10 and 1000 in the **DED File Range** Min and Max fields. When you do this, elevation values of the data file are adjusted proportionally depending on where the original value falls within the original min/max range. Specifically, elevation values of 10 meters remain 10, elevation values of 200 meters are adjusted to the new maximum of 1000, and all elevation values in between are scaled proportionally. More simply, the higher the original elevation, the more it increases given the new **DED File Range**.

There are other ways to alter the values output to the DED file, one of which is described in the Units section above.

Geographic Extents

Before you can build the output DED file, you need to specify where on the earth your elevation data is located. This "geographic extents" information is encoded into the DED file so that the Terrain Generation tools in Creator can properly place the terrain database it builds in the correct geographic position. You can enter the extents for your data in one of the following ways:

- Enter the **South West (SW) Corner** and **North East (NE) Corner** and let **Raster To DED** calculate the **Post Spacing** based on the number of elevations in each dimension of your data file.
- Enter the **SW Corner** and **Post Spacing** and let **Raster To DED** calculate the **NE Corner**.

You can also specify how you want the latitude and longitude values to be displayed in this area by selecting either of the **Degrees/Minutes/Seconds** or **Decimal Degrees** radio buttons.

Generate DED Button

After you have specified the layout of your raw data file, are satisfied with the contour map displayed in the Preview Area, and have entered the desired Geographic Extents for your data, you are ready to actually build the output DED file. Do this by simply pressing the **Generate DED** button. When you do this, a file chooser dialog is displayed in which you can select a raw file to convert. Once you have selected a file to convert, this area displays the name of that file as well as its size (in bytes).

Status and Progress

Along the bottom of the dialog, status and progress bars that display messages and reports progress during processing.

Converting Raw Elevation Data

Raw elevation data can be obtained from USGS, NIMA, or a third party source. However, Creator terrain conversion does not process raw elevation data. Cells must be read and converted to Creator's gridded terrain post format, referred to as Digital

Elevation Data (DED) format. By convention, DED files are names with the extension `.ded`.

Creator provides offline utilities to convert geographic data from NIMA and USGS sources to DED format. Other included utilities can convert RGB, RGBA, or INT images, or elevation data in Float format to DED.

There are separate procedures for converting source data that is stored on compact disc (CD) and source data that is stored on a hard disk. The `readusgs` utility, which converts USGS DEM data to DED, can be used on data from either storage device. Float data must be stored on hard disk before conversion to DED.

To Convert DTED Files From CD-ROM

If your DTED source files are on a CD, use the following procedure to read, convert, and save the files to hard disk.

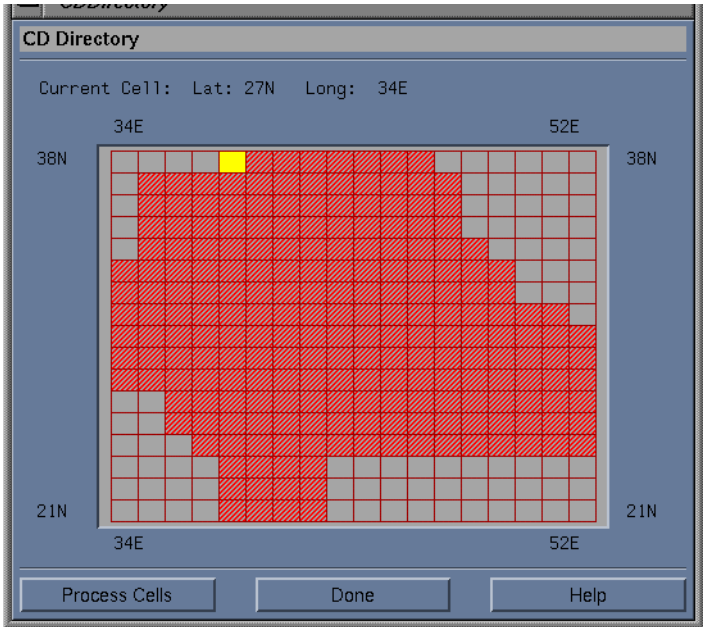
Step 1. Insert a DTED disk into the CD-ROM drive.

Step 2. Start Creator. You do not need to open a database window.

Step 3. Choose **Terrain/Read DTED CD** to open the *Read DTED CD (Open DMED File)* dialog box.

Step 4. Select a DMED file and click **OK**. The *CD Directory* window appears.

This window contains a grid showing the latitude, longitude, and extents of the terrain on the CD. Each red square represents one cell of data on the CD. Gray squares represent cells for which data is not provided.



Current Cell shows the latitude and longitude of the last cell that was selected.

Step 5. Select a set of terrain cells for processing. The selected cells turn yellow. If you do not select adjacent cells, gaps between cells are treated as areas of zero elevation.

Note: Each cell contains a large amount of data. One cell occupies 2.5 to 3 megabytes on disk, and approximately the same amount of extra workspace must be available for temporary files during processing. Be sure you have enough disk space to process the amount of terrain data you intend to select. It is best to process a few cells at a time.

Step 6. Click **Process Cells** to begin converting the gridded terrain post data to DED format. When conversion is complete, the selected terrain data appears in the *Terrain* window.

Step 7. In the *Terrain* window, click the **Map** tab and then click **Write Current Selection to Post File**.

Step 8. Save the selected gaming area in a new DED format file.

The *CD Directory* and *Terrain* windows stay open on the desktop so you can process additional terrain cells. Be sure to save your data each time you process new cells.

To Convert DTED Files From Hard Disk

Step 1. From a command prompt, change to the directory where Creator is installed (for example, `C:\MultiGen-Paradigm\creator` for Windows, or `/usr/local/MultiGen-Paradigm/creator` for IRIX). See Using the Conversion Utilities On Windows Systems or "Using the Conversion Utilities on IRIX Systems" on page 1207 for detailed instructions. The files you want to convert should also be located in this directory.

Step 2. Type `readdma`, and then press Enter.

readdma responds:

```
DMA DTED Raw Data Preprocessor Utility - Version <version num-
ber>.

From 0-Tape or 1_Disk?
```

Step 3. Type `1` and press Enter. (This utility is cross-platform, and the tape option is only available on IRIX systems.) The following prompt appears:

```
Input filename:
```

Step 4. Enter the name of the file you want to convert and press Enter. For example:

`n27.dt1` <Enter>.

readdma responds:

```
1-Level1 or 2-Level2 or 3-DTEDX?
```

Step 5. Enter the level number that corresponds to the NIMA level of your file and press Enter.

readdma responds:

```
WARNING: VOL, HDR records not found!
```

```
Output file name:
```

Step 6. Ignore the warning which appears because you are not reading the data from tape, and enter a name (including path) for the DED file you want to create.

readdma responds:

```
Default setting:

VOL=0 HDR=0 UHL=80 DSI=648 ACC2700 DATA 2414 EOF 4 UTL=5
Tape_Mark=0

0-Use default or 1-Specify manually?
```

Step 7. Type `0` and press Enter, unless your file does not follow standard NIMA DTED format.

`333` responds:

```
Number of cells on tape/file:
```

Step 8. Type `1` and press Enter. (Only files on tape have more than one cell per file.)

The utility issues file data messages during processing. When it is finished, you see the message `Done!` The new DED file can be imported into Creator.

To Convert USGS Data to DED Format

Step 1. From a command prompt, change to the directory where Creator is installed, for example, `C:\MultiGen-Paradigm\creator` for Windows, or `/usr/local/MultiGen-Paradigm/creator` for IRIX. The files you want to convert should also be located in this directory.

Step 2. Type `readusgs` and then press Enter.

`readusgs` responds:

```
USGS DEM Raw Data Preprocessor Utility - Version <version num-
ber>.

Enter Input DEM File Name?
```

Step 3. Type the name of the file you want to convert and press Enter. For example:

```
gcanyon.raw <Enter>
```

`readusgs` reads the file, issuing status messages in the process, and then prompts:

```
Enter Output .ded File Name?
```

Step 4. Type a name for the output file (`gcanyon.ded`, for example) and press Enter. Include a path (`/converted_files/gcanyon.ded`, for example) if you want to create the output file in a different directory than the one that holds the source file.

`readusgs` prompts:

```
Does <filename> specify a file name or directory name on the tar-
get (F=file, D=directory)?
```

Step 5. Type `F` to specify a file name and press Enter. `readusgs` creates the DED file, which can be imported into Creator.

Note: If you enter `D`, `readusgs` creates a directory with the name you specified, and then create a file named `ssys.ded.tmp` in that directory, you must rename the temporary file because subsequent conversions will overwrite it.

To Convert Float Data to DED Format

The `float2ded` utility converts elevation data in *float* format (a format that expresses elevation data as a series of floating point values) to DED format.

Step 1. From a command prompt, change to the directory where Creator is installed. The files you want to convert should also be located in this directory.

Step 2. Type `float2ded`, a space, the input file name, a space, the image width in pixels, a space, and the image height in pixels, and then press Enter. For example:

```
float2ded sfbayflt5.dat 3075 2890 <Enter>
```

`float2ded` displays messages about the DED file it is creating, and then saves the new file in the same directory, and with the same root name as

the source file. For example, `sfbayflt5.dat` is processed into the DED file `sfbayflt5.ded`.

30 Importing and Creating New Features

Feature data, also called “culture data” or “vector data”, consists of all parts of the database that are not terrain polygons. These features can be natural, such as lakes and rivers, or man-made, such as houses and roads.

With Creator, you can import existing feature data into your terrain database, create a new layer of feature data, add new features, modify existing features, and export (save) feature data as DFD for import into other databases.

The following sections describe how to create, modify, and export feature data.

Converting Feature Data

Before you can import feature data into Creator, you must convert the data to MultiGen’s proprietary DFD format. *DFD* (Digital Feature Data) is a form of DFAD that is processed more efficiently and supports a higher resolution than does standard DFAD.

Creator processes Levels I, I-C, II, and X DFAD files. These files are available on CD-ROM or 9-track tape, or they can be downloaded from the Internet. Data on 9-track tapes must be saved to a hard disk before it can be converted.

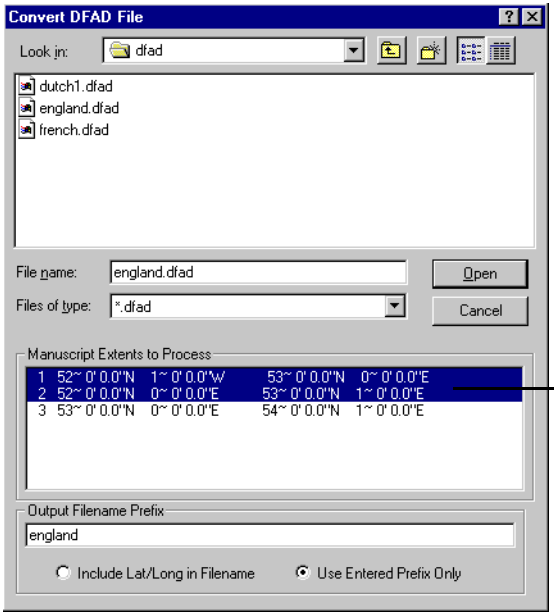
To convert DFAD data to DFD format:

Step 1. Open a terrain database and choose **GeoFeature/Import/Convert DFAD to DFD**.

Step 2. In the *Convert DFAD File* dialog box, select a raw DFAD file for processing.

Note: Make sure the file you select is a raw DFAD file. Because DFAD files have an unusual format, selecting another type of file may cause Creator to hang.

Step 3. Select a DFAD manuscript from the **Manuscript Extents to Process**.



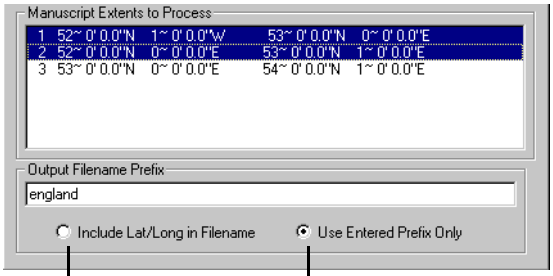
The manuscripts (geographical areas) contained in the selected files appear here.

To select additional manuscripts, hold down Ctrl while clicking each manuscript.

To select a sequence of manuscripts, select a manuscript and hold down Shift while clicking a manuscript above or below it in the list.

Step 4. Specify the naming convention for the DFD files that will be created. To do this:

- Enter an **Output Filename Prefix**. Each manuscript is saved in a separate DFD file using the directory path and prefix specified in this field.
- Choose **Include Lat/Long in Filename** to name the files with the prefix plus the manuscript number, or choose **Use Entered Prefix Only** to name the files with the prefix plus a unique number.



Selecting this button produces the files:

englandN52_0_0W1_0_0.dfd

englandN53_0_0E0_0_0.dfd

Selecting this button produces the files:

england0.dfd

england1.dfd

Step 5. Click **OK** to convert the selected manuscripts to DFD format.

Importing Feature Data Into Creator

After you convert feature data to DFD, you can import it into your terrain database.

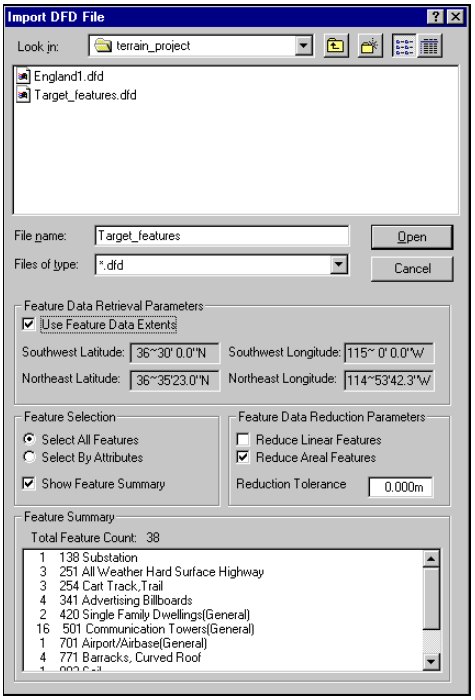
You should import feature data that falls within, or at least partially overlaps, the area covered by the terrain database. Imported feature data is shown as a construction tree in the Hierarchy view and as a construction layer in the Graphics view.

Note: If the geographical location of the feature data you import does not at least partially overlap the terrain area in the database, you will not see the construction layer in the Graphics view. You can select features in the Hierarchy view, modify their attributes, and export the modified data to a DFD file, but you cannot use modeling tools to edit the data in the Graphics view, or project the features onto the terrain.

To Import DFD Files

Step 1. Open the terrain database into which you want to import the feature data.

Step 2. Choose **GeoFeature/Import /Import DFD**. The *Import DFD File* dialog box appears.



Because DFD files usually contain far more feature data than you can use in a realtime system, the **Import DFD File** dialog box lets you preview and filter feature data before importing it.

Step 3. Select the DFD file that contains the feature data you want to import.

Step 4. To select more than one file, hold down Ctrl while clicking each file name. To select a block of files, click the first file name and hold down Shift while clicking a second file name above or below it in the list.

Step 5. Use the **Feature Data Retrieval Parameters** to specify the geographical area that contains the features you want to import.

To import features from a limited geographical area, set **Use Feature Data Extents** and enter the latitude and longitude values for the southwest cor-

ner (**Southwest Latitude** and **Southwest Longitude**) and northeast corner (**Northeast Latitude** and **Northeast Longitude**) of the area. Features outside this area will not be imported.

To import features from the entire geographical area of the DFD file, make sure **Use Feature Data Extents** is cleared.

Step 6. Set the **Feature Selection** parameters to either **Select All Features** or **Select By Attributes**.

Select All Features directs Creator to import every feature in the selected files.

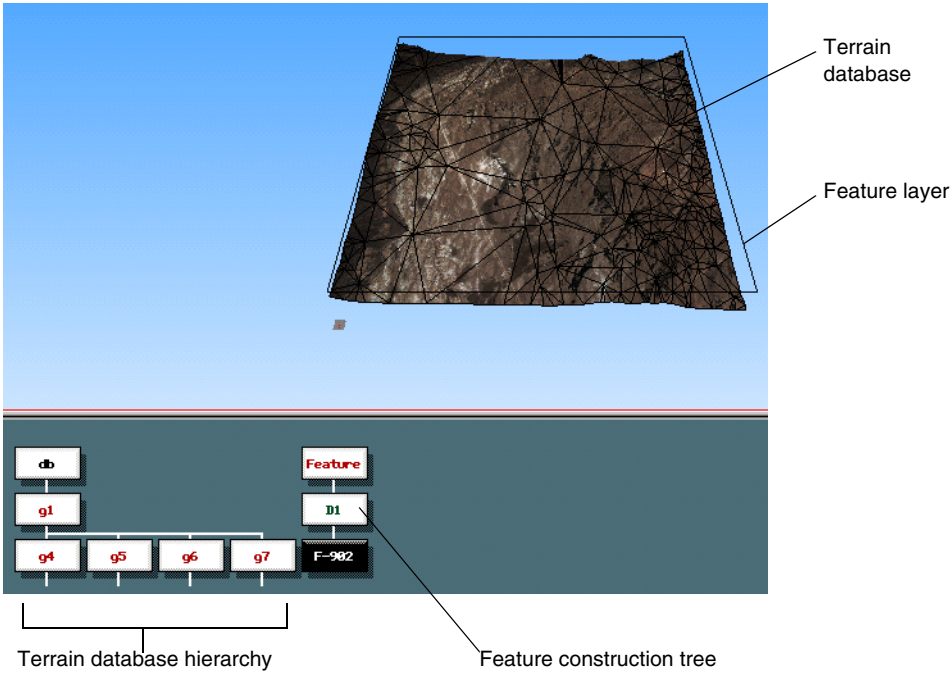
Select By Attributes opens the *Feature Search* dialog box, prompting you to choose search criteria. Only those features that meet the criteria you specify are imported. DFD files generally contain much more data than you can display in realtime. When you import only a subset of the feature data in a file, you can save a lot of time, because you do not have to select and delete unneeded features after they are imported. For instructions on how to use **Feature Search**, see "To Perform a Feature Attribute Search" on page 1123.

Step 7. Set the **Feature Data Reduction Parameters** to control the amount of detail imported for linear and areal features. When more than two vertices in a feature lie along a nearly straight line or edge, the **Feature Data Reduction Parameters** eliminate vertices in the middle. The **Reduction Tolerance** defines how much each vertex can deviate from a straight path between the two adjacent vertices before it is retained.

Note: You can also reduce the vertices of features *after* they are imported into a database (see "To Reduce Vertices" on page 1119).

Step 8. Click **OK** to import the selected featu-re data.

The imported feature data appears as a layer (the *Feature layer*) that floats above the terrain in the Graphics view, and as a separate single-layer construction tree in the Hierarchy view.



You can delete any part of this data, add features to it, and use Creator modeling tools to modify it. Once you are satisfied with the feature data, you *project* it to convert it to OpenFlight format and incorporate it into your terrain database (see "Projecting Features Onto Existing Terrain" on page 1157). You can also export the feature data to a DFD file to save your changes (see "Saving Feature Data" on page 1125).

Feature Data Retrieval Parameters

These parameters in the *Import DFD File* dialog box define the geographical area from which feature data will be imported.

Southwest Latitude, Southwest Longitude, Northeast Latitude, Northeast Longitude

Defines the rectangular area that contains the feature data you want imported. These four parameters give the geographical coordinates of the southwest and northeast corners of the area. Features located outside this area are ignored.

Use Feature Data Extents

Determines whether data will be imported from the entire area contained in the files you are importing, or whether feature data will be imported only from the area defined by the **Southwest Latitude**, **Southwest Longitude**, **Northeast Latitude**, and **Northeast Longitude**.

To import data from the entire geographical boundary of the files you have selected for import, make sure **Use Feature Data Extents** is cleared. In this mode, you can import features representing a geographical area that differs from the area of the terrain database. However, you cannot project a feature that does not correlate with terrain.

To specify a particular geographical area, enable **Use Feature Data Extents** and then enter the latitude and longitude values for the southwest and northeast corners of the area you want to import. These boundaries must fall within the area of the terrain database.

Feature Selection Parameters

Defines whether all features, or only features that meet specific criteria, are imported.

Select All Features

Imports all cultural features in the selected feature data files. Because these files contain large amounts of data, it is usually better to use **Select By Attributes**.

Select By Attributes

Prompts you to perform an attribute search, and then imports only those features that meet your search criteria (see"To Perform a Feature Attribute Search" on page 1123).

Show Feature Summary

Sets/clears the display of the **Feature Summary** display at the bottom of the dialog box.

Feature Summary

Displays the number and description of each feature type currently selected for import, shows how many features will be imported for each feature type, and displays the total number of features selected for import.

Feature Data Reduction Parameters

Eliminates vertices that fall along straight or nearly straight lines, reducing the number of polygons that will be created when each feature is projected.

Reduce Linear Features

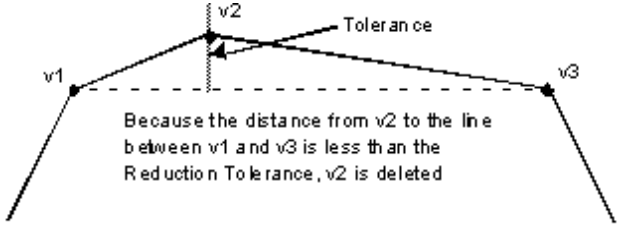
Enables vertex reduction for linear features.

Reduce Areal Features

Enables vertex reduction for areal features.

Reduction Tolerance

Controls vertex reduction. If the distance between a vertex and the line segment connecting its two neighbors is less than **Reduction Tolerance** (in database units), the vertex is deleted.



Imported features become a permanent part of the geometry when you project them or perform a library substitution (see “Projecting Features Onto Existing Terrain” on page 31-1157). However, if you close the database before projecting the features, they are lost, since the Feature construction tree is not saved with the database.

Before you project features, you can modify them in the construction layer using modeling tools, or you can select them and change their attributes (see "Modifying and Correcting Feature Data" on page 1110).

Creating New Features

If you are starting with a new file, you must have at least one terrain polygon in the database. A terrain polygon is a polygon whose **Terrain** parameter is set in the *Face Attributes* window.

You create new features by adding them to the feature construction layer. You can either import feature data to create the construction layer, or you can use **GeoFeature/Feature Construction/New Feature Layer** to create a new layer. Once you add a DFD construction layer, you set any feature in this layer as the parent, and then use commands in the **GeoFeature** menu to create new features.

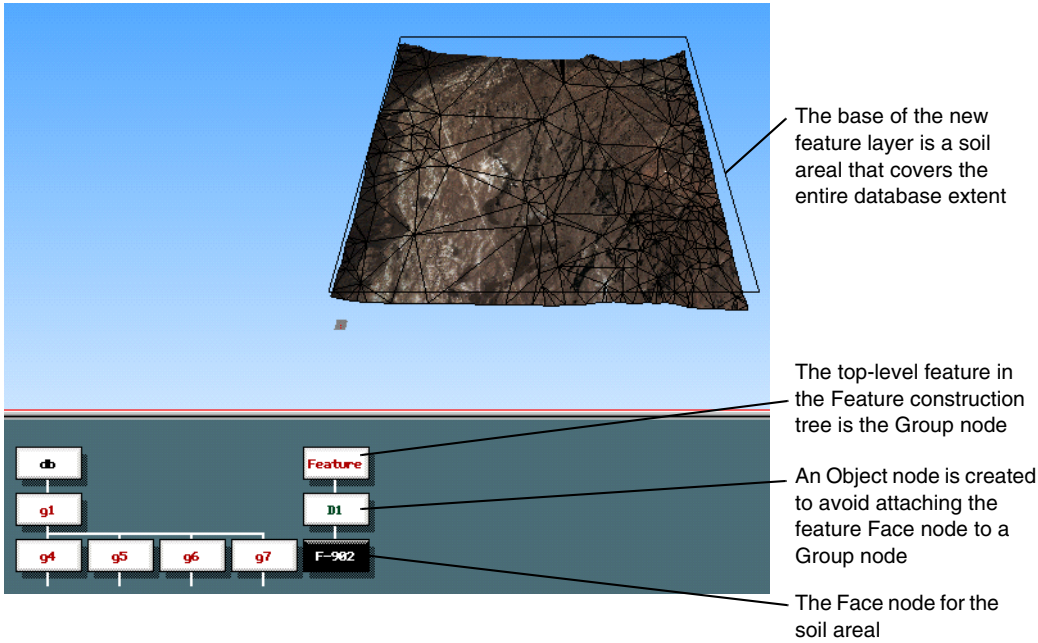
Feature data is displayed as a single construction layer. New features are added to the extreme right in the hierarchy no matter which node is set as the parent. Features do not become permanent parts of your terrain database until they are projected.

> **Note:** If you save the database and then close it, all feature data is lost unless you projected it or exported it to DFD format before you closed the database. See "Saving Feature Data" on page 1125 for more information.

Creating a New Feature Layer

To create a new feature construction layer, you must have a terrain polygon on which to place the new Feature layer. Then select **GeoFeature/Feature Construction/New Feature Layer**. A new Feature construction tree appears in the Hierarchy view. This tree has three nodes: the top node is the parent of the construction group, its child is an Object node that represents the extent of the database soil layer, and the third node is

the polygon that represents the soil layer. In DFD format, each feature is represented by an Object/Face node pair.



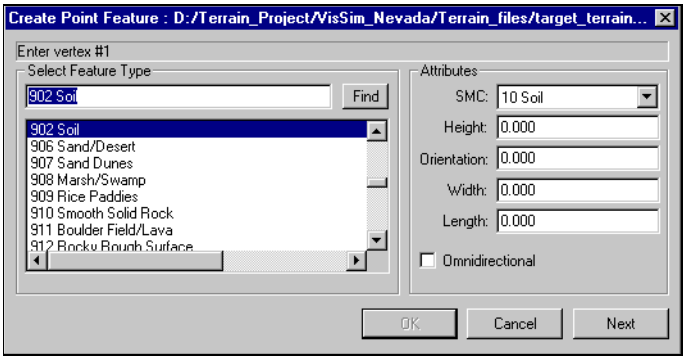
You are now ready to add features to this layer.

To Create a Point Feature

A *point feature* is a model that is centered around a single vertex. Buildings of special interest, for example, skyscrapers that are not part of an areal city, are typical point features. To create a new point feature:

Step 1. Select any feature in the construction layer in the Graphics view, or select any node in the feature construction tree in the Hierarchy view, and click **Set Parent**.

Step 2. Select **GeoFeature/Feature Construction/Create Point Feature**. The *Create Point Feature* dialog box appears:



Step 3. Select a **Feature Type** from the scrolling list or enter a number or description and click **Find** to search for the feature type you want. **Feature Type** selections in this dialog box are equivalent to DFAD FIDs, which assign categories and numbers to classify different types of features. For information about feature types, including instructions for creating a new feature type, see "About Feature Types" on page 1107.

Step 4. Click the arrow next to the **SMC** text field and select a Surface Material Code from the list that appears.

Note: Make sure the SMC you choose for this feature matches one of the SMCs selected for the feature type in the *Feature Preferences* window (see "Setting Feature Preferences" on page 1130).

Step 5. Enter the height of the feature in the **Height** text field. (The units for **Height**, **Width** and **Length** are expressed in the current database units.)

Step 6. If you want the feature to look the same from all sides, set **Omnidirectional** and enter a value in the **Length** field. The new point feature will be an eight-sided circular object with a diameter that equals the **Length**.

If you want the feature to be rectangular, make sure **Omnidirectional** is cleared, and enter values in the **Orientation**, **Width**, and **Length** fields.

Orientation describes how the feature is aligned when compared to the x and y axes of the origin. When this value is zero, the feature is aligned with

these axes. Enter a positive value to rotate the feature clockwise the specified number of degrees. Enter a negative value to rotate the feature counterclockwise.

Step 7. Click the point in the Graphics view where you want to place the feature.

Step 8. Click **Next** in the *Create Point Feature* dialog box to complete this feature and begin creating another feature, or click **OK** to complete this feature and close the dialog box.

The new feature appears as a polygon in the Feature layer of the Graphics view, and as a node at the far right end of the feature construction tree in the Hierarchy view, as soon as you click **Next** or **OK**. The polygon is oriented as specified, but you will not see its height until you project the feature into the database. See "Projecting Features Onto Existing Terrain" on page 1157.

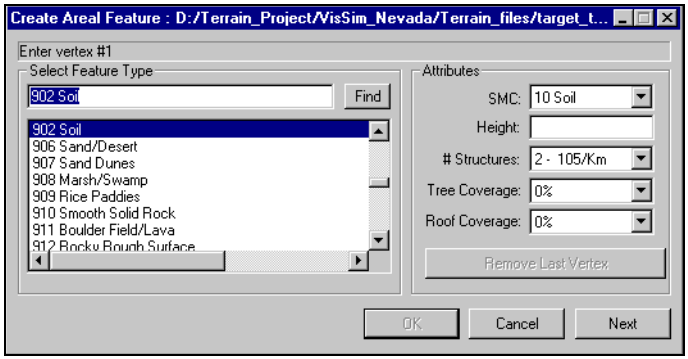
To Create an Areal Feature

An *areal* is a model that is created or positioned by a polygon. When you create an areal, you see a single polygon in the Feature layer. When you project the areal into the database, it acquires height, depth, color and texture. This is how, for example, a single irregular polygon in the Feature layer is transformed into an urban area in the database.

To create a new areal feature:

Step 1. Select any feature in the construction layer in the Graphics view, or select any node in the feature construction tree in the Hierarchy view, and click **Set Parent**.

Step 2. Select **GeoFeature/Feature Construction/Create Areal Feature**. The **Create Areal Feature** dialog box appears:



Step 3. Select a **Feature Type** from the scrolling list or enter a number or description and click **Find** to search for the feature type you want. **Feature Type** selections in this dialog box are equivalent to DFAD FIDs, which assign categories and numbers to classify different types of features. (For information about Feature Types, including instructions for creating a new feature type, see "About Feature Types" on page 1107.)

Step 4. Click the arrow next to the **SMC** text field and select a Surface Material Code from the list that appears.

Note: Make sure the SMC you choose for this feature matches one of the SMCs selected for the feature type in the *Feature Preferences* window (see "Setting Feature Preferences" on page 1130).

Step 5. Enter a **Height** for this feature. **Height** automatically is expressed in the current database units.

Step 6. Click the arrow next to the **# Structures** text field and, from the list that appears, select a range that approximates the number of structures per kilometer this areal will represent.

Step 7. Click the arrow next to the **Tree Coverage** field and, from the list that appears, select the percentage of the areal that would be covered by trees. For example, if the areal represents a tree canopy, select 100%; if the areal represents a lake, select 0%.

Step 8. Click the arrow next to the **Roof Coverage** field and, from the list that appears, select the percentage of the areal that would be covered by buildings. For example, if the areal represents a city, select 100%; if the areal represents a lake or a tree canopy, select 0%.

Step 9. Click points in the Graphics view to define the vertices of the polygon that defines the areal.

Step 10. Because areals are created counter-clockwise, the points should be defined in this order.

Step 11. Click **Next** in the *Create Areal Feature* dialog box to complete this feature and begin creating another, or click **OK** to complete this feature and close the dialog box.

Step 12. The new feature appears as a polygon in the Feature layer of the Graphics view, and as a node at the far right end of the feature construction tree in the Hierarchy view, as soon as you click **Next** or **OK**. You will not see the areal's height until you project it into the database.

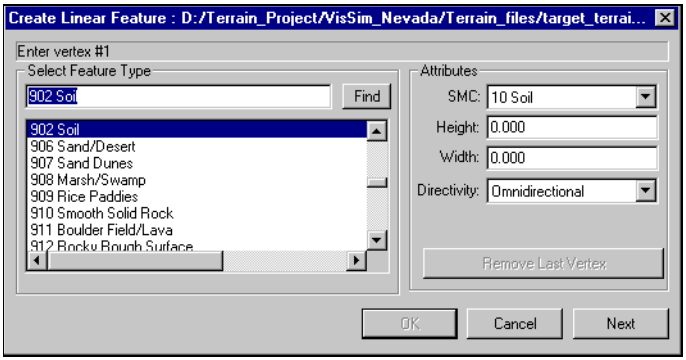
To Create a Linear Feature

A linear feature is one that can be represented by a line of vertices. Roads and rivers are typical linear features. A line of trees or a row of buildings might also be processed more efficiently in the realtime system if it is created as a linear feature, rather than as a series of point features.

To create a new linear feature:

Step 1. Select any feature in the construction layer in the Graphics view, or select any node in the feature construction tree in the Hierarchy view, and click **Set Parent**.

Step 2. Select **GeoFeature/Feature Construction/Create Linear Feature**. The *Create Linear Feature* dialog box appears.



Step 3. Select a **Feature Type** from the scrolling list or enter a number or description and click **Find** to search for the feature type you want. **Feature Type** selections in this dialog box are equivalent to DFAD FIDs, which assign categories and numbers to classify different types of features. (For information about Feature Types, including instructions for creating a new feature type, see "About Feature Types" on page 1107.)

Step 4. Click the arrow next to the **SMC** text field and select a Surface Material Code from the list that appears.

Step 5. Make sure the SMC you choose for this feature matches one of the SMCs selected for the feature type in the *Feature Preferences* window (see "Setting Feature Preferences" on page 1130).

Step 6. Enter a **Height** for this feature. **Height** is automatically expressed in the current database units.

Step 7. Enter a **Width** for this feature. **Width** automatically is expressed in the current database units.

Step 8. Click the arrow next to the **Directivity** text field and choose a value.

Step 9. **Unidirectional** permits movement along the linear in one direction only. A river, for example, flows in only one direction.

Bidirectional permits movement in two directions. A highway is generally bidirectional.

Omnidirectional permits movement in any direction.

Note: The **Directivity** value is used by the realtime system; it has no effect on the feature's appearance in Creator.

Step 10. Click points in the Graphics view to define the vertices of the linear.

Step 11. Click **Next** in the **Create Linear Feature** dialog box to complete this feature and begin creating another, or click **OK** to complete this feature and close the dialog box.

The new feature appears as a line in the Feature layer of the Graphics view, and as a node at the far right end of the feature construction tree in the Hierarchy view, as soon as you click **Next** or **OK**.

To see the width of the feature in the Graphics view, select the linear and then select **GeoFeature/Toggle Linear Features**.

Creating Features From an Image

The **Build Polys From Image** tool in the **GeoFeature** menu imports a large image as a series of tiles into the Graphics view. This allows you to digitize features using the image as a background pattern. Generally, geospecific textures are used for this operation, but you can use any texture.

Build Polys From Image is somewhat similar to the **Chop Texture** feature in the Texture palette, except the texture is not loaded into memory until after it has been tiled. The advantage to this is that very large textures, such as highly detailed, high resolution satellite photographs, are not downsampled to accommodate texture memory.

To use **Build Polys From Image**:

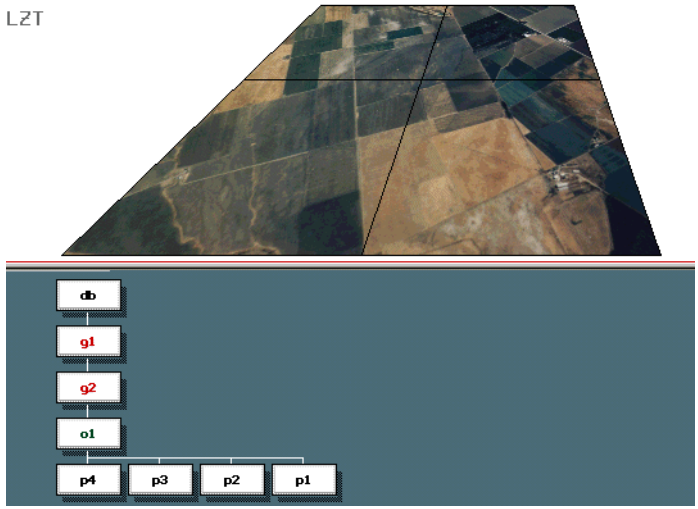
Step 1. Choose **GeoFeature/Feature Construction/Build Polys From Image**. The *Build Polys From Image* dialog box opens.

Step 2. In the **Texture** field, enter the directory path and file name of the texture file to import, or click Browse and select the directory and file in the *Browse Texture File Name* dialog box, and click **Open**.

Step 3. Enter the output directory where Creator will place the tiled texture files, or click **Browse** and select a directory in the *Browse for Folder* dialog box, and click **OK**.

Step 4. Set the **Tile Size**. This is the size, in texels, that Creator will size each of the texture tiles.

Step 5. Click **OK** to import the texture. The texture is divided into even size tiles, applied to polygons, and the texture tiles are loaded into the Texture palette.



Step 6. Turn on **Draw Texture** in the View panel.

Step 7. Select a polygon to be digitized in either the Graphics or Hierarchy view.

About Feature Types

MultiGen's DFD format uses Feature Types that are equivalent to the Feature Identification (FID) categories used by standard feature data formats such as DFAD and DLG. For example, you can choose a linear feature in the Feature construction layer and check its Feature Type to see whether it is a road, a river, a line of trees, and so forth.

The Feature Type is independent of the feature's classification as a point, linear, or areal feature. A Feature Type that identifies a particular category of tree, for example, can be applied to a point feature to identify a single tree, to a linear feature to identify a treeline composed of the same type of tree, or to an areal that represents a forest of trees.

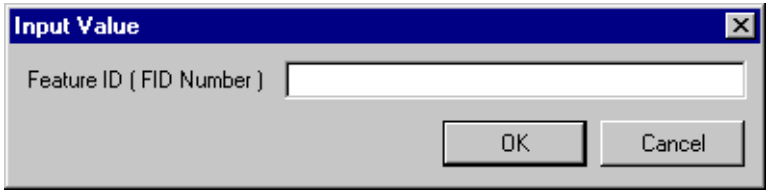
Creator uses Feature Types for more than simple identification of features, however:

- Each Feature Type is associated with a set of **Feature Preferences** that identifies valid SMCs for the feature. The combination of the Feature Type and SMC sets the colors, textures, and materials that will be applied to the feature when it is projected, and defines how the feature is placed on the terrain.
- If you choose the Batch method for projecting terrain with features, you create Rules and Actions that determine which features will be projected, and in what order. The primary attribute used in Batch processing for sorting features is the Feature Type.
- DFAD and DLG use numbers and brief definitions to define FIDs (102 Quarry, 103 Gas/Oil Derrick, and so forth). MultiGen's DFD format recognizes all DFAD and DLG FIDs as equivalent Feature Types, and also allows you to create your own Feature Types.

To Create a New Feature Type

Step 1. Select **GeoFeature/Feature Preferences**. The *Feature Preferences* window appears.

Step 2. Click the **Add FID** button. The *Input Value* dialog box prompts you to add a **Feature ID (FID) number**.



Step 3. Click the **OK** button. The *Input Value* dialog box prompts you to add a **Feature Type Description**.



Step 4. Enter a name for the new Feature Type in the **Feature Type Description** field. You can enter any unique label you want. Your custom Feature Types do not require numbers (in fact, adding numbers could cause errors if they conflict with established DFAD FID assignments).

Step 5. Click **OK**. The new Feature Type will appear in all Creator Feature Type lists.

Thinning and Reducing Feature Data

DFAD files usually contain far more data than can be processed in a realtime system. For this reason, Creator offers opportunities to thin the data (eliminate features you do not need) and reduce it (eliminate vertices to decrease the number of polygons in each feature you decide to keep).

- When you import data, the *Import DFD File* dialog box allows you to define attribute search conditions to import only the features that meet specific criteria. You can also eliminate data that does not fall within specified latitude and longitude locations, and set a reduction tolerance for linears and areals to eliminate vertices that are not necessary for retaining the general shape of the feature. (See “To Import DFD Files” on page 1093.)
- Once the data is imported, you can use the modeling tools to eliminate unnecessary vertices and edges, which reduces the overall polygon count when the features are projected. (See “Using Modeling Tools to Edit Features” on page 1115 and “To Reduce Vertices” on page 1119.)
- If you project a feature manually, you can perform attribute searches to project specific subsets of the feature data. (See “To Perform a Feature Attribute Search” on page 1123 and “Projecting Features Onto Existing Terrain” on page 1157.)

- If you project features in a batch operation, you create Rules and Actions that control exactly which features are projected, and in what order (see "Batch Terrain With Feature Projection" on page 1177).

Modifying and Correcting Feature Data

Once you import feature data, you can modify the wireframe construction objects in the Feature layer before you project them. These modifications can also be saved as feature data for future import by exporting the data to a DFD file.

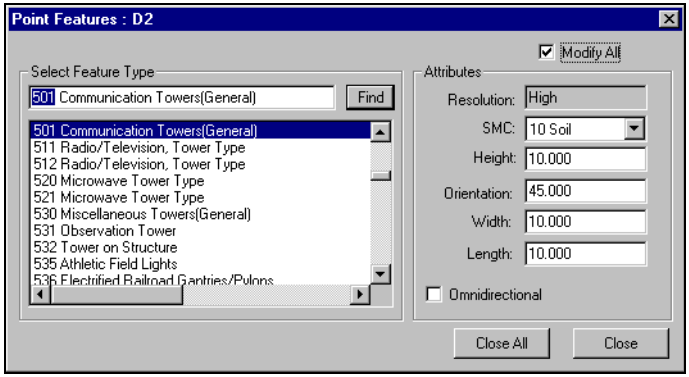
You can use some of Creator's modeling tools to change a feature's size, shape, or position. You can also change settings in the feature's attribute window, or use the **Reduce Vertices** tool to eliminate some of the vertices and thus reduce the polygon count when the feature is projected. The following sections describe these procedures.

To Change Feature Attributes

You can modify a feature by making changes in its attribute window. To do this:

Step 1. Select the feature or features.

Step 2. Choose **Attributes/Modify Attributes** (Ctrl+=). The attribute windows for the selected features appear. For example, a point feature has the following attributes:



Step 3. Change the value of the attribute you want to modify.

Step 4. When you are finished, click **Close**.

Point Feature Attributes

Point features have the following attributes.

Select Feature Type

Assigns a category and number to classify the feature. (For information about Feature Types, including instructions for creating a new feature type, see "About Feature Types" on page 1107.)

Modify All

Applies the changes that you make to every areal feature that currently is selected.

Resolution

Describes the type of library substitution that will occur (high or low) if you use library substitution while projecting this feature.

SMC

The valid Surface Material Code for this feature, which describes the composition of the feature. A point feature that has Feature Type 620 School might be composed of Stone/Brick or Concrete, for example.

Height

The **Height** that will be assigned to this feature when you project the feature, expressed in the current database units.

Orientation

Describes how the feature is aligned when compared to the *x* and *y* axes of the origin. When this value is zero, the feature is aligned with these axes. When this value is positive, the feature is rotated clockwise the specified number of degrees. When this value is negative, the feature is rotated counterclockwise.

Width, Length, and Orientation

The dimensions that will be assigned to the feature when the feature is projected, expressed in the current database units.

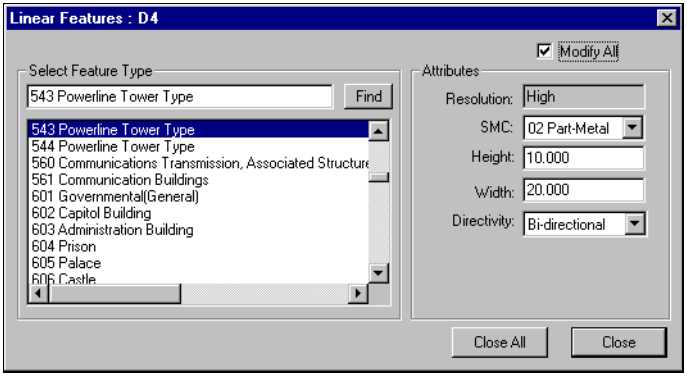
If **Omnidirectional** is set, the point feature will be an eight-sided circular object with a diameter that equals the **Length**.

If **Omnidirectional** is cleared, the feature will be rectangular, and its size with be determined by the **Width**, and **Length** attributes.

Close and Close All

Close closes the current attribute window. **Close All** closes all attribute windows that currently are open.

Linear Feature Attributes



Linear features have the following attributes.

Select Feature Type

Assigns a category and number to classify the feature. (For information about Feature Types, including instructions for creating a new feature type, see "About Feature Types" on page 1107.)

Modify All

Applies the changes that you make to *every areal feature that currently is selected*.

Resolution

Describes the type of library substitution that will occur (high or low) if you use library substitution while projecting this feature.

SMC

The valid Surface Material Code for this feature, which describes the composition of the feature. A road that has Feature Type 251 All Weather Hard Surface Highway might be composed of asphalt or concrete, for example.

Height and Width

The dimensions that will be assigned to this feature when you project the feature, expressed in the current database units.

Directivity

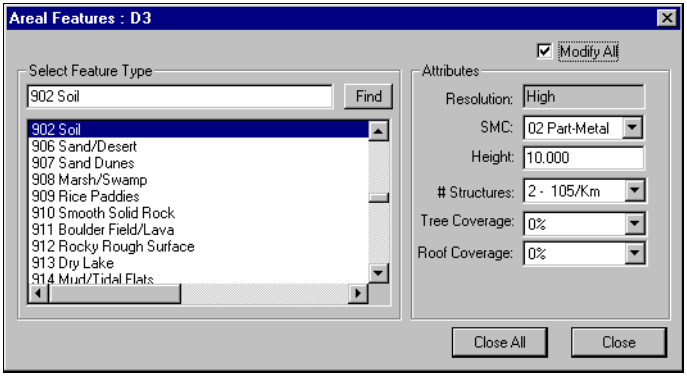
Controls movement along the linear. The **Directivity** value is used by the realtime system; it has no effect on the feature's appearance in Creator.

- **Unidirectional** permits movement along the linear in one direction only. A river, for example, flows in only one direction.
- **Bidirectional** permits movement in two directions. A highway is generally bidirectional.
- **Omnidirectional** permits movement in any direction.

Close and Close All

Close closes the current attribute window. **Close All** closes all attribute windows that currently are open.

Areal Feature Attributes



Areal features have the following attributes.

Feature Type

Assigns a category and number to classify the feature. (For information about Feature Types, including instructions for creating a new feature type, see "About Feature Types" on page 1107.)

Modify All

Applies the changes that you make to *every areal feature that currently is selected*.

Resolution

Describes the type of library substitution that will occur (high or low) if you use library substitution while projecting this feature.

SMC

The valid Surface Material Code for this feature, which describes the composition of the feature. An areal that has Feature Type 913 Dry Lake might be composed of sand or rocks, for example.

Height

The **Height** that will be assigned to this feature when you project the feature, expressed in the current database units.

Structures

The range that approximates the number of structures per kilometer this areal will represent.

Tree Coverage

The percentage of the areal that would be covered by trees. For example, if the areal represents a tree canopy, select 100%; if the areal represents a lake, select 0%.

Roof Coverage

The percentage of the areal that would be covered by buildings. For example, if the areal represents a city, select 100%; if the areal represents a lake or a tree canopy, select 0%.

Close and Close All

Close closes the current attribute window. **Close All** closes all attribute windows that currently are open.

To Copy and Paste GeoFeature Attributes

You can assign identical attributes to multiple features using the **Copy GeoFeature Attributes** and **Paste GeoFeature Attributes** tools in the **Properties** toolbox. All features involved in the operation must be in the same category of point, areal, or linear. You cannot copy GeoFeature attributes from a linear feature, for example, and paste them to an areal feature. To use these tools:

Step 1. Make sure you are in Object mode.

Step 2. In either the Graphics or Hierarchy view, select a single feature whose attributes you want to copy.



Step 3. Open the **Properties** toolbox.



Step 4. Click **Copy GeoFeature Attributes**.

Step 5. Select the features to which you want to copy the attributes.

Step 6. You can fence a wide area, if you want, because all selections that are not in the same category as the copied feature will be ignored.

Step 7. Click **Paste GeoFeature Attributes**. The attributes of all the selected features are changed to match the copied feature. See “Point Feature Attributes” on page 1111, “Areal Feature Attributes” on page 1113, and “Linear Feature Attributes” on page 1112 for descriptions of these attributes.

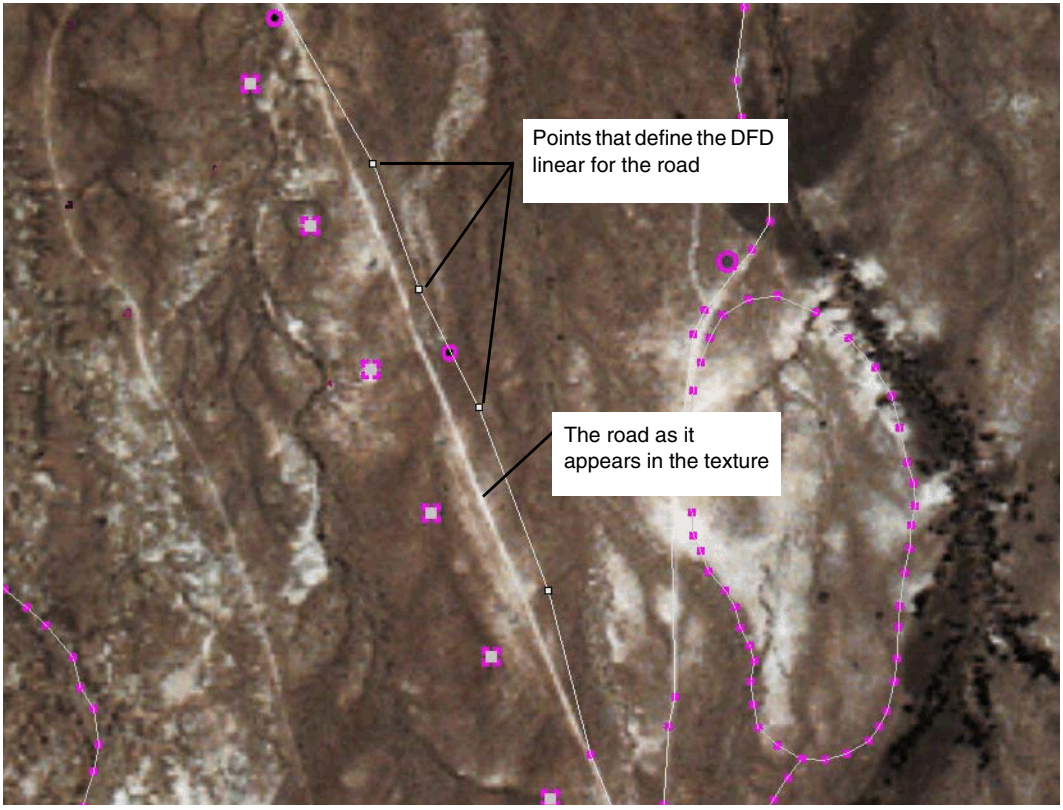


Using Modeling Tools to Edit Features

You can resize and reposition wireframe construction objects in the Feature layer with modeling tools that apply to faces and vertices. This lets you correct the feature data manually before projecting it into the database or exporting it to a DFD file.

- You can select an areal or linear feature in the Feature layer and change its size, shape, and orientation using the **Translate**, **Scale**, **Rotate About Point**, and **Modify Vertex** tools.
- You can move point features using the **Translate** tool. To change a point feature's dimensions and orientation, do not use the **Scale** and **Rotate About Point** tools; modify the feature's attributes instead (see "To Change Feature Attributes" on page 1110).

For example, in the following figure, the feature data for a road does not match its location in the satellite photo texture for the area.



To Edit a Feature With a Tool

This procedure describes how to set the grid and tracking plane and use the **Translate** tool to correct a feature in the Feature layer.

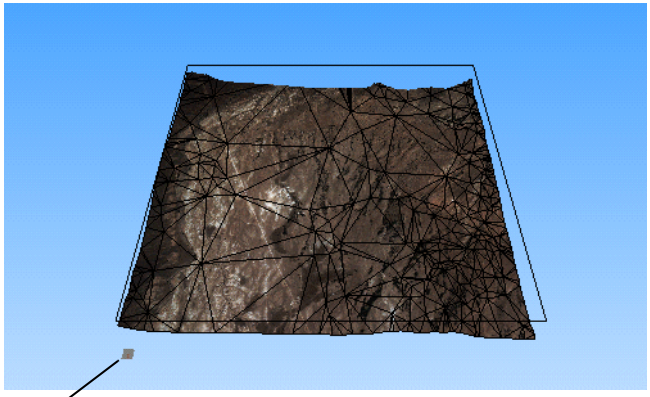
Note: For a instructions on how to use a specific modeling tool, see the *Online Help Summary; MultiGen Creator*.



Step 1. Click the **View Panel** button on the Toolbar to open the **View** panel.

Step 2. If the grid display is off, click the **Grid** button in the **View** panel to turn on the grid. The default grid size is very small compared to most terrain databases; if you cannot see the grid display, zoom out until you see the lower-left corner of the terrain area.





Step 3. Click the **Enlarge Grid** button in the **View** panel until the grid covers the feature you want to correct.



Step 4. Select the feature and choose **View/Fit on Select (V)**.

Step 5. From the **Eyepoint** menu on the Toolbar, set the eyepoint to **Top** to position it directly over the feature.



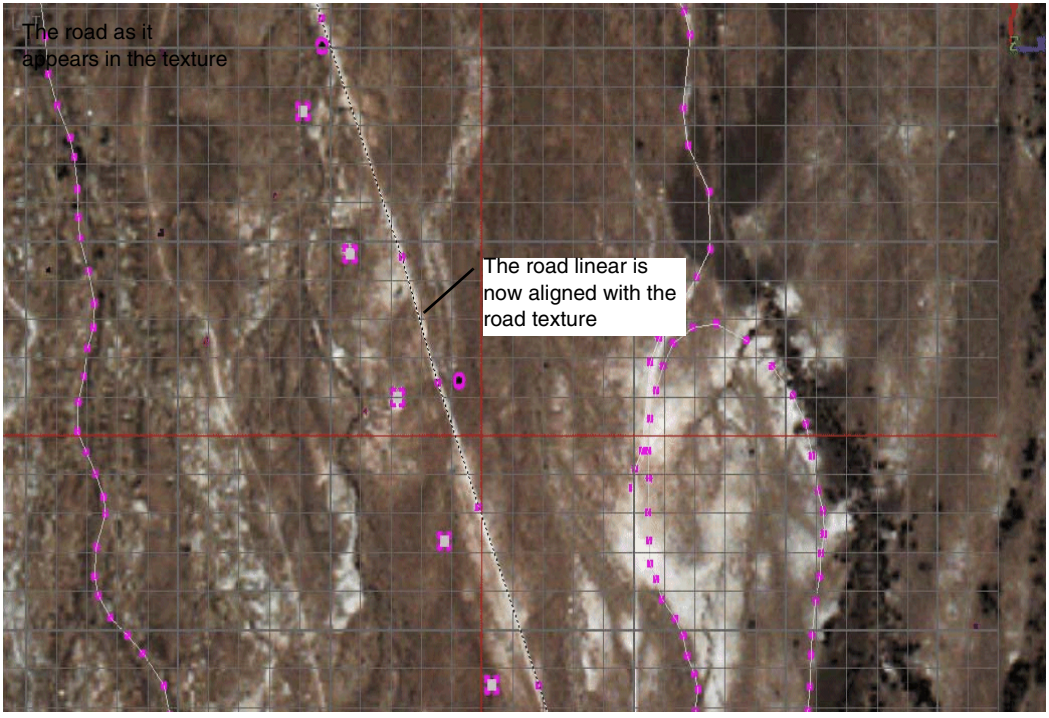
Step 6. Select **Vertex** mode (Shift+V) from the Toolbar.

Step 7. Select any vertex in the feature and click the **Trackplane From Vertex** button in the **View** panel to reposition the trackplane and grid by referencing the selected vertex.



This trackplane position ensures that any vertices you move during the modification will be coplanar with the vertex you selected in the feature.

Step 8. Select the vertices you want to move and use the **Translate** tool to reposition them over the road in the texture.



To Test a Modified Feature

To test the modified feature, change to Face mode, select the feature, and then select **GeoFeature/Feature Projection/Project Feature(s)**. You can select **Edit/Undo** to undo the projection if you want to perform further modifications or project the feature with other features.

Scatter Points in Areals

Scatter Points in Areals builds point features inside selected areal features that have the same FID, SMC type, and height as the areal. The number of features created is determined by the **# Structures** value in the areal feature's attributes.

To Scatter Point Features in Areals

Step 1. In the Feature layer, select the areal feature or features to scatter points in.

Step 2. Choose **GeoFeature/Feature Construction/Scatter Points in Areals**.

Step 3. In the *Audit DFAD* dialog box, set the parameters, and click **OK**.

Scatter Point Features In Areals Parameters

Omnidirectional

When set, the point feature orientation is omnidirectional.

Randomize Orientation

When set, the point feature orientation is random, based on the Orientation Value.

Orientation Value

The point features' percentage of rotation when **Randomize Orientation** is set.

Width Value

Width of the point features when **Randomize Orientation** is set.

Length Value

Length of the point features when **Randomize Orientation** is set.

Check Overlap

When set, checks for overlapping point features and moves them. If Creator cannot place point features without overlapping, it will create fewer features than the number specified by the **# Structures** value.

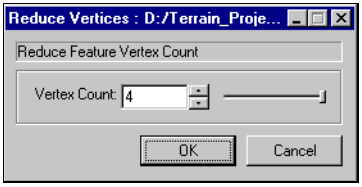
To Reduce Vertices

To make feature data more efficient to display, you may want to reduce the number of vertices in some of the wireframe features. **Reduce Feature Vertices** applies a dynamic vertex reduction algorithm to a selected wireframe object. Reducing vertices decreases the number of polygons when the feature is projected.

To reduce the number of vertices in wireframe construction data, do the following:

Step 1. Make sure you are in Object mode, and then select a feature, either in the Graphics view, or in the feature construction tree in the Hierarchy view.

Step 2. Choose **GeoFeature/Feature Construction/Reduce Feature Vertices**. The *Reduce Vertices* dialog box appears.



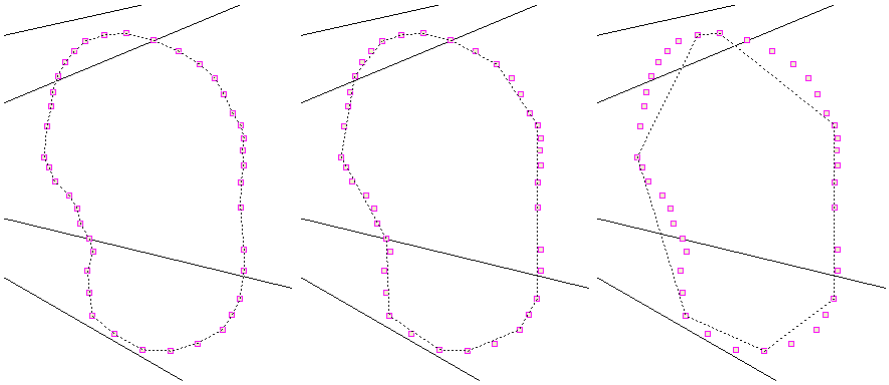
Step 3. Enter a number in the **Vertex Count** field, or use the slider to specify the number of vertices you want in the feature. The vertices that have the smallest impact on the shape of the feature are deleted.



When **Draw Vertex Large** is on (you set this in the **View** panel), all original vertices remain visible, while the wireframe display in the Graphics view updates dynamically. This makes it easy to judge how closely the new shape resembles the original feature.

For many features, it is possible to delete a significant number of vertices and still maintain a good representation of the feature.

Step 4. Click **OK** to save your changes.



An areal with 41 vertices

The same areal with 14 vertices

The same areal with 7 vertices

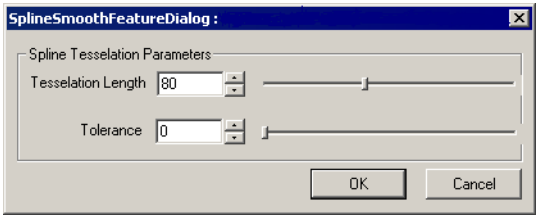
To Smooth Features

You can reduce vertices within a tolerance that you specify to smooth areal or linear features. **Spline Smooth Feature** allows you to set the distance between any two points, and then reduce the number of vertices between points to straighten splines, for example.

To smooth areal or linear features, do the following:

Step 1. Make sure you are in Object mode, and then select a feature, either in the Graphics view, or in the feature construction tree in the Hierarchy view.

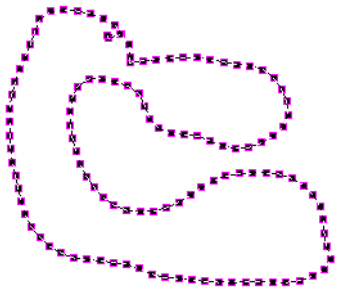
Step 2. Choose **GeoFeature/Feature Construction/Spline Smooth Feature**. The *SplineSmoothFeature* dialog box appears.



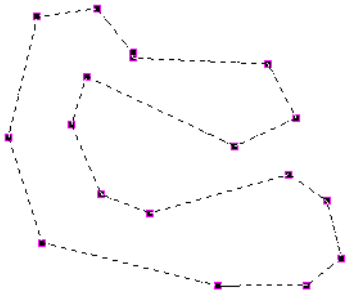
Step 3. Enter a number in the **Tesselation Length** field, or use the slider to specify the distance between any two points in meters.

Step 4. Enter a number in the **Tolerance** field, or use the slider to set the distance between any two points in which to delete vertices. As vertices are deleted, curves are smoothed to appear more linear.

Step 5. Click **OK** to save your changes.



An areal with 0 tolerance. No vertices were deleted.



When the tolerance was increased to 200, many vertices were deleted. The curves became more angular in shape.

Displaying and Selecting Features

Creator provides several tools that help you view feature data, making it easier to import, edit, and project features.

- **Select By Attributes** in the *Import DFD* dialog box can be used to select features before you import data and **GeoFeature/New Feature Search** or **GeoFeature/ Feature Search** can select features at any time after import.
- **GeoFeature/Show Feature Data** is a toggle that displays all features or only those features that are selected. This tool is especially useful for viewing only those features selected during an Attribute Search.
- **GeoFeature/Toggle Linears** switches the view of selected linear features from wireframe to solid, allowing you to see the actual width of linear features, making it easier to correct situations where linears intersect other features.

To Perform a Feature Attribute Search

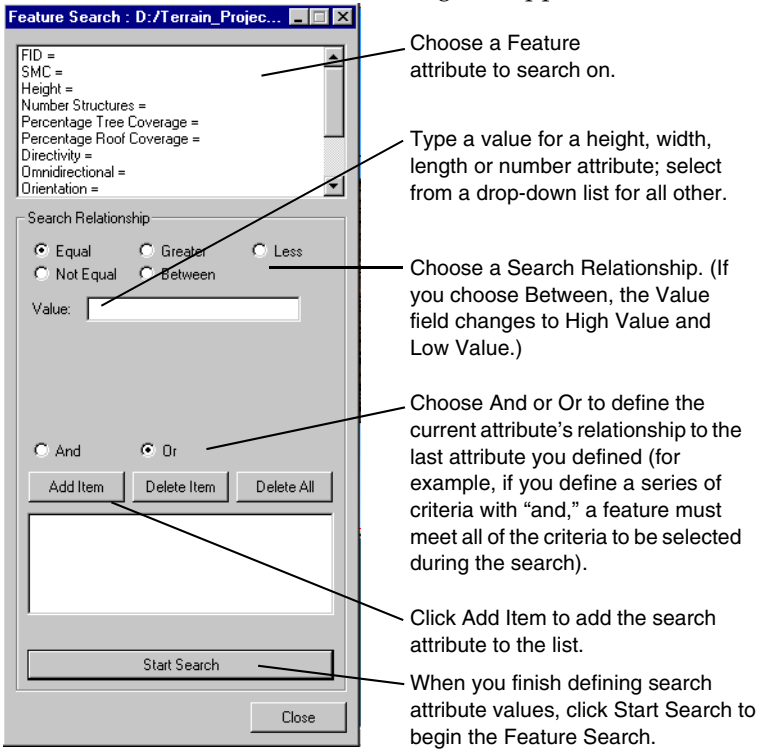
Attribute searching selects wireframe features by attribute value. This makes it easy to select construction objects to project into the database. Attribute searching is also useful, in conjunction with **Show Feature Data** (see "Showing and Hiding Features" on page 1124), in managing the number of wireframe features that appear in the Graphics view.

To perform an attribute search, do the following:

Step 1. Choose **GeoFeature/New Feature Search** to open the *Feature Search* dialog box, or choose **GeoFeature/Feature Search** to modify and rerun a previously-defined search.

If you are importing DFD feature data, choose **Select By Attributes** in the *Import DFD File* dialog box.

The *Feature Search* dialog box appears.



Step 2. Select an attribute from the list at the top of the dialog box.

Step 3. Select a **Search Relationship** (**Equal**, **Greater**, **Less**, **Not Equal**, or **Between**) for the attribute value. Relationships are displayed only when they have meaning for the selected attribute.

Step 4. Specify a value (or range of values) for the selected attribute.

Step 5. If the attribute value is chosen from a predefined set (for example, from the set of valid surface material codes), an arrow appears next to the **Value** text field. Click the arrow and select a value from the list that appears.

Step 6. Click **Add Item**. The new search condition appears in the list at the bottom of the dialog box.

Step 7. Add or delete search conditions until you are satisfied with the search.

To add a search condition, click **And** or **Or** to define the relationship between previous search conditions and the next search condition, and return to Step 2.

To delete a search condition, select it from the list at the bottom of the dialog box and click **Delete Item**. To delete all search conditions, click **Delete All**.

Step 8. Click **Start Search** to perform the attribute search. Features whose attributes match the search criteria are now selected in the database and are displayed in the Select list.

Showing and Hiding Features

GeoFeature/Show Feature Data is a toggle that controls the amount of feature data that is visible in a Graphics view.

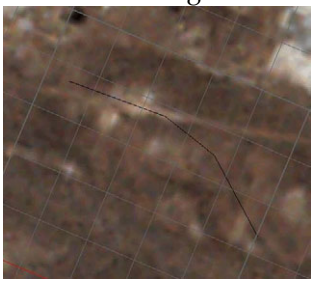
If you deselect **Show Feature Data**, you turn off the graphical display of all features in the Feature layer that are not selected. If you perform an attribute search, and then deselect **Show Feature Data**, you turn off the display of features that were not selected by the search.

When **Show Feature Data** is selected, all features in the Feature layer are displayed.

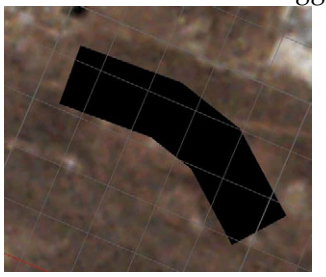
Using Toggle Linears

When you project features into the database, linear features are shown as wireframes (for example, a road is shown as a simple line). To switch between wireframe and solid views for linear features only, choose **GeoFeature/Toggle Linears**.

- If one or more linears are selected, only the view of the selected linears is changed.
- If nothing is selected, the view of all linears is toggled.



A linear in wireframe view



The same linear in solid view, showing the actual width

Toggle Linears is a useful tool for determining if linear features overlap other features.

Saving Feature Data

Projected features are saved with the database, but feature data in the feature construction tree is not saved. If you want to save feature data in DFD format, you must export it before you close the database. The following sections describe how to export feature data to DFD files.

To Export All Features to DFD

You can export all of the feature data in the current terrain database, or export only the selected part of the data to a DFD file.

To export all feature data to a DFD file:

Step 1. Make sure nothing is selected by choosing **Select/Deselect All**.

Step 2. In the Database window, select **GeoFeature/Export/Export DFD**. The *Export to DFD File* dialog box appears.

Step 3. Enter a name for the new DFD file (or select an existing file name to overwrite the file with new data) and then click **Save**. The feature data is saved to the file.

To Export Selected Features to DFD

To export a selected subset of the feature data to a DFD file:

Step 1. In either the Graphics or Hierarchy view, select the DFD features you want to export.

Step 2. In the Database window, select **GeoFeature/Export/Export Select to DFD**. The *Export to DFD File* dialog box appears.

Step 3. Enter a name for the new DFD file (or select an existing file name to overwrite the file with new data) and then click **Save**. The selected feature data is saved to the file.

31 Adding Features to Terrain

Features consist of all parts of a terrain database that are not terrain polygons. These features can be natural (lakes and rivers, for example) or man-made (houses and roads, for example). *Feature data* (also called "culture data" or "vector data") files can be obtained from many sources and are generally in DFAD, DLG, or Vector format. For each feature, the data usually consists of a code that indicate what kind of feature it is (house, road, or lake, for example), what category (linear, point, or areal) and information about its location, size, orientation, and composition (brick, gravel, water, and so forth). Before you import this data into a Creator database, you must first convert the data to Creator's DFD formats using one of Creator's offline converters, Gview (for SGI IRIX only), or a third party converter such as Okino PolyTrans™.

Once you have converted feature data to Creator's DFD format, you can import it into Creator and modify it. Modification includes thinning and reducing features, correcting errors, correlating the feature data with geospecific textures, and adding new features. The data can then be written out to a new feature data file.

The ultimate purpose for importing and modifying feature data is to *project* the features into the database. When you project feature data, you create 3D OpenFlight models from the data. As an alternative, you can perform *library substitution*, where instead of using the information in the feature data to create a simple structure, Creator substitutes a predefined OpenFlight model that is sized to match the size of the feature, and placed in the proper location, with the proper orientation.

Feature data does not have to originate in the "real world." With Creator, you can also create your own new features and add them to existing feature data files, or start your own new feature data files.

The general sequence of tasks that are performed when adding features to terrain are as follows. (These tasks are explained in detail in the rest of this chapter.)

1. Gather raw feature data and convert it to DFD format.
2. Decide whether you want to pre-project any features and prepare the necessary cutout or footprint files.
3. Import the feature data into your terrain database using the GeoFeature menu, or process your terrain as TCT and choose Project Culture Only (no terrain).
4. Correct overlapping and intersecting features using Creator editing tools.

5. Add or reduce features, if necessary.
6. Save the edited feature data to a new cutout or footprint file, or export the data to a new DFD file to be used for projecting the features.
7. Set up feature preferences, which define how each of the features will appear when then are projected.
8. Set up feature projection preferences, including library substitutions.
9. Set up rules and actions if you will be post-projecting features using the Batch terrain processing option, or set up Apply Rules to Files to batch-project features on existing terrain.
10. Project the features.

These steps are explained in detail in the following topics. For an overview of the process of importing, correcting, thinning, and projecting feature data, and how these steps fit into the overall goal of creating a terrain database that will run efficiently in realtime, refer to the book *Creating Terrain For Simulations*, which is included with Creator in printed form, and as an Acrobat (.pdf) file on the Creator CD.

Types of Feature Projection

"Importing Feature Data Into Creator" on page 1093 describes how to import existing feature data into your terrain database. Once you import the data, you can create a new layer of feature data, add new features, modify existing features, and export (save) feature data as DFD for import into other databases.

Once you have verified and corrected your feature data you project it onto your terrain. You can project features onto terrain using the following methods:

- "Manual Post-Projection" on page 1129
- "Automatic Post-Projection" on page 1129
- "Automatic Pre-Projection" on page 1129
- "Apply Rules to Files" on page 1130

Manual Post-Projection

When you post-project features onto existing terrain, you import the feature data in layers. You can reduce the number of features that are imported by performing attribute searches and only importing the feature you want.

Manual post-projection requires the following setup files: Feature Preferences, Global projection preferences, library substitution models, if needed, and texture, color and material palette files, if used.

Automatic Post-Projection

When you generate terrain as a batch process, you can also automatically project features. Automatic post-projection is controlled by rules and actions, which tell Creator what features to extract from the feature data and how to project them onto the terrain. Using the Feature panel of the *Terrain* window, you specify the feature files to use, and associate rules with appropriate LODs. When you click **OK** to generate the terrain, Creator processes the area blocks first and then automatically extracts the features from the DFD files and projects them and writes out the populated file.

Automatic post-projection requires the following files: Feature preferences, global projection preferences, rules and action files, texture, color and material palette files, if used, light string palette file, if light points or strings will be part of your database, and library substitution models, if needed.

Automatic Pre-Projection

Pre-projected features are projected before the terrain is generated. The terrain polygons are then stitched around the features. Because the features are cut in, and are part of the terrain rather than sitting on top of it, it requires fewer polygons. Pre-projected features are cutout files, footprint files, or Pathfinder features.

To pre-project features you must be generating terrain using TCT as a batch process, or CAT. TCT pre-projects both cutout files and footprint models. CAT processes only cutout files.

Pre-projection requires the following files: Feature preferences, global projection preferences, rules and action files, texture, color and material palette files, if used, light

point palette file, if light points or strings will be part of your database, cutout files, and library substitution footprint models.

Apply Rules to Files

If you have already generated your terrain as a batch process, you can automatically post-project features using the Apply Rules to Files tool in the GeoFeature menu. You can do this without opening any of the terrain files.

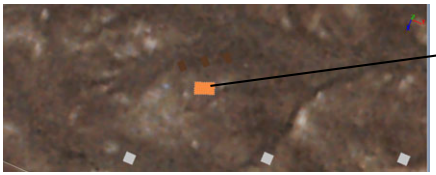
Apply Rules to Files requires the following files: Feature preferences, global projection preferences, rules and actions, texture, color and material palette files, if used, light string palette file, if light points or strings will be part of your database, cutout files, and library substitution/footprint models.

Setting Feature Preferences

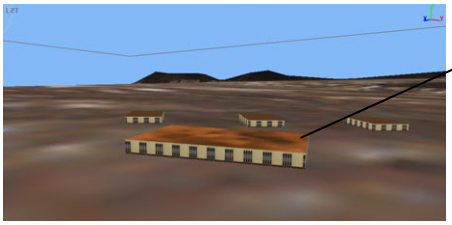
Feature Preferences control how features are projected into the database. When you project features, Creator creates 3D models based on each feature's attributes, and then applies colors, textures, materials, and so forth based on the settings in the *Feature Preferences* window.

You set Feature Preferences for each Feature Type in your feature data, and within a Feature Type, for each surface material code (SMC). What color or texture do you want to apply to roads with an asphalt SMC? What about concrete roads with the same Feature Type? Do you want an areal of a city to have a grid of glowing lights when it is projected? Should all buildings be planted on the terrain? All these features are controlled by the Feature Preferences.

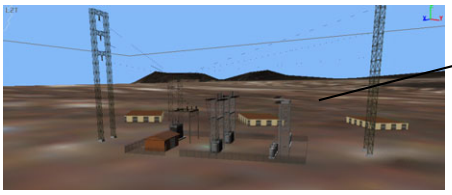
You can also use Feature Preferences to set parameters for *library substitution*. When you enable library substitution, Creator places a specified OpenFlight model on the terrain instead of creating a 3D representation from the feature data.



A power substation is created as a point feature in the Feature layer



When the feature is projected into the database, colors and textures are assigned based on the Feature Preference settings

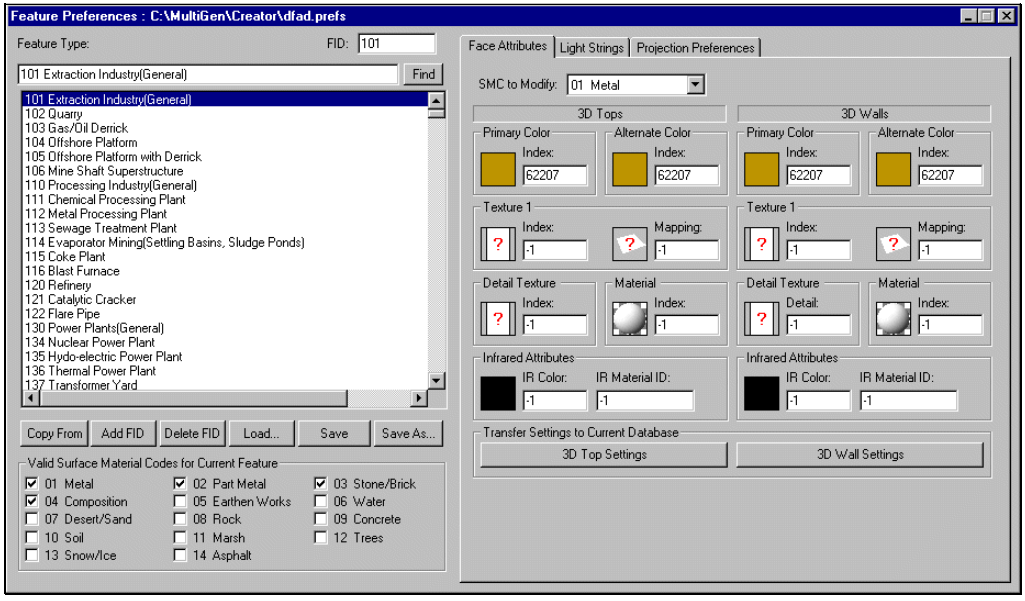


When the same feature is projected using library substitution, an OpenFlight model replaces the feature

A default Feature Preference file (dfad.prefs) is stored in the same directory as the Creator application. You can load other Feature Preference files and save feature preferences in the *Feature Preferences* window.

To Set Up Feature Preferences

Step 1. Select **GeoFeature/Feature Preferences** to open the *Feature Preferences* window.



Step 2. Select a **Feature Type** from the list. Each Feature Type has its own set of preferences. You can copy preferences from one Feature Type to another (see "Transferring Attribute Values" on page 1142) or go to the next step to create a new set of preferences for the Feature Type.

Step 3. Check all the **Valid Surface Material Codes** that apply to this Feature Type. (The default surface material codes are usually correct. See "Choosing Surface Material Codes" on page 1133.)

Each time you check a different SMC, it adds the SMC to the **SMC to Modify** list in the **Face Attributes** panel. This is because features get their texture, color, and material assignments based on the Feature Type/SMC pair assigned to them. For example a wood building and a stone building, both of which have the same Feature Type, receive different textures when they are projected.

Step 4. Click the **Face Attributes** tab and set preferences that control the color, infrared code, texture, and material of polygons that are projected for features that have the selected Feature Type (see "Choosing Face Attributes" on page 1134).

Step 5. You choose a different set of preferences for the tops and walls of the 3D models that are created from feature data. If you want the color, texture, material, and texture mapping settings that you choose for the 3D tops or 3D walls to become the current palette settings in the Database window (so you can create models that have matching attributes, for example) click **3D Top Settings** or **3D Wall Settings**.

Step 6. If you want to create light points at the each vertex of a projected feature, create light strings between the vertices of projected features, or cover the feature with a grid of light points, click the **Light Strings** tab and select Light Strings parameters (see "Light Point Preferences" on page 1137).

Step 7. Click the **Projection Preferences** tab and choose Feature Projection Preferences for the Feature Type. If you plan to use library substitution for the Feature Type, which places prefabricated models in the database rather than creating 3D models from the feature data, select **Library Substitution Preferences** and choose the model file(s). (See "Library Substitution Preferences" on page 1140.)

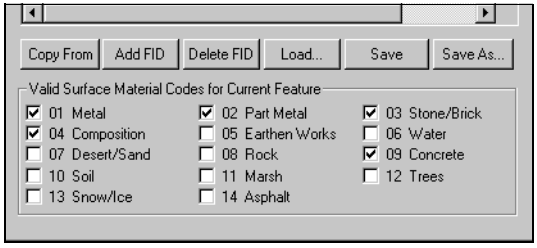
Valid SMCs and Face Attributes

Valid Surface Material Codes in the *Feature Preferences* window specify all the possible surface material codes (SMCs) that features in the selected Feature Type can have.

Each individual feature in the feature layer has a Feature Type and a single SMC associated with it. When you project a feature, Creator automatically applies the colors, textures, and materials specified by the **Face Attributes** for that Feature Type and SMC.

Choosing Surface Material Codes

To assign surface material codes to the selected Feature Type, check each SMC in the **Valid Surface Material Codes for Current Feature** that applies to the Feature Type.



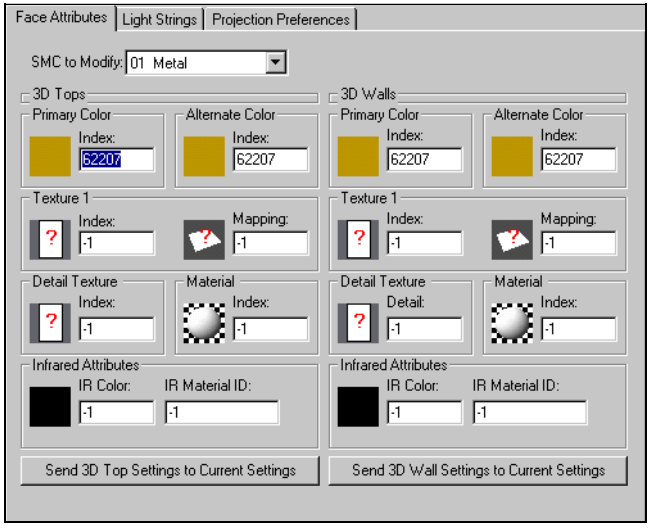
For simplicity, set only the surface material codes that are appropriate for the selected Feature Type. For example, Feature Type 114 Evaporator mining nearly always has a surface material code of Water, while the surface material code of Feature Type 110 (Processing, industry) could be Metal, Part metal, Stone/brick, or Composition. In most cases, the default set of surface material codes is correct.

You are now ready to assign **Face Attributes** to each surface material code.

Choosing Face Attributes

When you check one of the **Valid Surface Material Codes** in the *Feature Preferences* window, a new entry appears in the **SMC to Modify** list in the **Face Attributes** panel of the window. This is because texture, material, and color are assigned separately for each surface material. With this arrangement, a stone bridge can be textured with a different pattern than a metal bridge, for example, even though the two bridges have the same Feature Type. To choose Face Attributes:

Step 1. Click the **Face Attributes** tab to display the **Face Attributes** panel.



Step 2. Select a a surface material code from the SMC to Modify list. The **Textures**, **Detail Textures**, **Colors**, **Texture Mapping** index and **Materials** assigned to the selected surface material code appear in graphical displays in the panel.

If you click any button in the graphical display, its setting changes to whatever selection is current in the corresponding palette. For example, if you click the **Texture 1** display in the**3D Walls** area, the current texture in the Texture palette is assigned to **3D Walls**.

Step 3. Double-click the display for the attribute you want to change. This opens the corresponding Texture, Color, or Material palette

Step 4. Make a selection from the palette and then close the palette.

Step 5. Click the display for the attribute again to assign the new palette selection to it.

You can also edit each attribute directly by typing a new palette index or IR code in the corresponding text field. You do not need to set up a value for every type of attribute; if your application does not require **IR code** or **Detail Texture**, for example, leave a value of -1 (no assignment) in those fields.

Face Attribute Parameters

Face attributes associate a specific color, IR (infrared) code, texture, detail texture, and material with each surface material code that is enabled for the selected Feature Type. These attributes become face attributes when a feature with the selected Feature Type is projected into the database.

3D Tops

All attributes in the **3D Tops** area define attributes that are assigned to the tops of three-dimensional objects, and also to the faces of two-dimensional objects (such as lakes and roads) and to subfaces.

The **Primary Color** in the **3D Tops** area also determines the color of each wireframe feature in the Feature layer before it is projected into the database.

To make the current palette settings in the Database window the same as the settings you just chose for **3D Tops**, click the **3D Top Settings** button under **Transfer Settings to Current Database**.

3D Walls

All attributes in the **3D Walls** area define attributes that are assigned to the tops of three-dimensional objects, and also to the faces of two-dimensional objects (such as lakes and roads) and to subfaces.

To make the current palette settings in the Database window the same as the settings you just chose for **3D Walls**, click the **3D Wall Settings** button under **Transfer Settings to Current Database**.

Feature Preference Texture Considerations

The **Real World Size** (**Width** and **Height**) are very important when texture is mapped to each feature during the projection of feature. **Real World Width** and **Real World Height** determine the size of a single repetition of each texture pattern. Make sure you have good values in these attributes before you assign a texture pattern in the **Face Attributes** Feature Preferences.

Light Point Preferences

Light points are individual vertices that represent distantly-viewed points of light such as stars, runway lights, or city lights. Light Point feature preferences define the light points that are displayed when features that have the selected Feature Type are projected. You can also choose to disable light points for specific Feature Types, and enable them for others.

Choosing Light Point Preferences

Because each Feature Type can be assigned to areal, linear, and point features, there are separate preferences for linears and areals; point features use the same preferences as areal features.

To set Light Point feature preferences:

Step 1. Click the **Light Point** tab to display the **Light Point** panel.

Step 2. If you want to use the settings in an existing light point palette, click the arrow next to **Appearance Palette Entry** or **Animation Palette Entry,** and select a palette entry from the list that appears.

You can click **Palette** to display the *Light Point Palette* dialog box and create new palette entries to use. In this dialog box, you can also load a different palette by clicking the **Load Palette** button to import a palette file.

Step 3. Select either **Linears** to set preferences for linears with this Feature Type, or select **Areals** to set preferences for areals with this Feature Type.

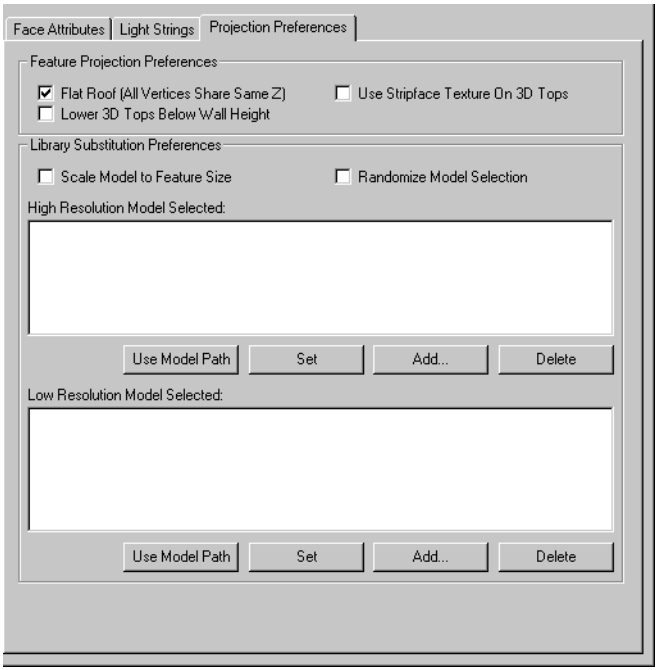
Step 4. Choose to project **No Lights**, **Lights Only**, or **Lights and Polygon**. For specific Feature Types, you can project polygons with no lights, for example. The default setting is **No Lights**.

Step 5. Set modeling attributes for light point color and light point placement for each Feature Type. See “Light Point/String Attributes” on page 634 for a description of each attribute.

Step 6. Click **Save** to save the settings.

Feature Projection Preferences

The **Projection Preferences** in the *Feature Preferences* window control the geometry and texturing of features as they are projected. Click the **Projection Preferences** tab to display these preferences.



Flat Roof (All Vertices Share Same Z)

Controls the way features with height are projected onto the terrain.

If **Flat Roof** is set:

- The tops of features with height are projected parallel to the *xy* plane, using the highest vertex in the feature.
- Texture applied to the sides of features with height repeat in both the ***u*** and ***v*** directions.

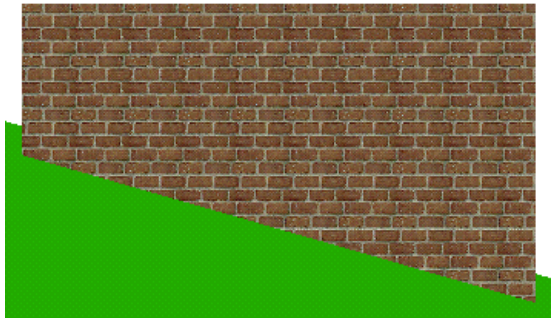
You should set the **Flat Roof** option to project small man-made objects, such as buildings. Because it raises the top of the entire feature to the level of the highest vertex, it is not appropriate for projecting areal features that cover large terrain areas.

If **Flat Roof** is cleared:

- The tops of features that have height are projected parallel to each terrain face.
- Texture applied to the sides of features with height repeats in the ***u*** direction, but clamps in the *v* direction.

Clear **Flat Roof** when you want features with height to follow changes in the terrain. Because patterns are clamped in *v*, it also lets you apply a texture pattern that does not repeat vertically, such as a tree.

> **Note:** Texture is only applied to polygons during projection if the **Apply GeoFeature Texture** preference is set in the *Projection Preferences* dialog box (see "Setting Global Projection Preferences" on page 1144).



Flat Roof (All Vertices Share Same Z) is set



Flat Roof (All Vertices Share Same Z) is cleared

Use Strip Face Texture On 3D Tops

Controls whether linear features that have height are textured using *global* texturing or *strip face* texturing. Global texturing is used to apply a noise texture pattern (a texture pattern with no specific orientation) to a feature. Strip face texturing is used to apply patterns that need to match along the center of consecutive faces, such as road stripes.

In global texturing:

- The horizontal faces of each feature are textured as a group.
- The texture origin is the southwest corner of the *bounding box* (surrounding rectangular area) of the feature.
- The texture pattern repeats across the faces, with the size of a single repetition determined by the texture pattern's **Real World Size** attributes.
- Because the faces of the feature are textured as a group, the ***u,v*** mapping values match across adjacent faces. This allows a realtime executive, such as Performer, to create a T-mesh (Triangle mesh), for efficient rendering of the faces.

In strip face texturing:

- Texture is mapped separately onto each horizontal face of the feature.
- The texture is oriented along the centerline of each face and matches at the center of consecutively textured faces. For proper alignment, the texture pattern should be oriented vertically with respect to the *v* axis.
- If one repetition of the texture pattern does not cover a texture face, the pattern repeats along the length of the face but is clamped (does not repeat) across the width of the face. The size of the texture pattern is determined by its **Real World Size** attributes.

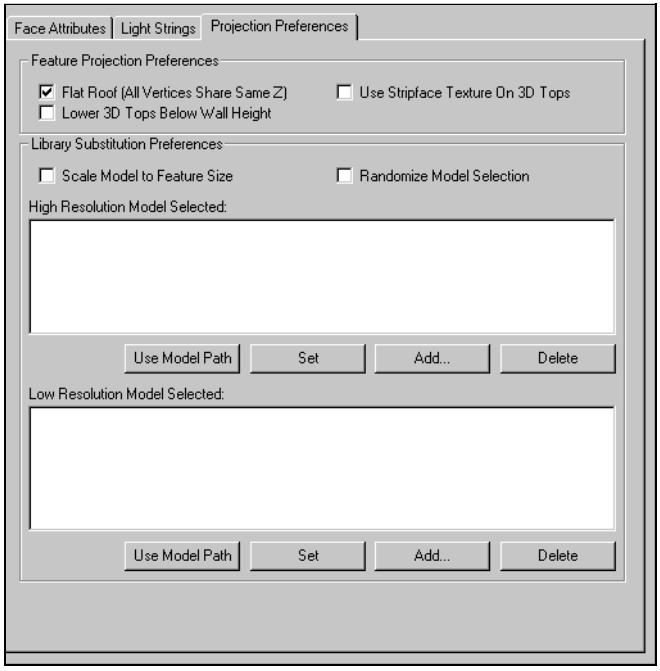
Texture is only applied to polygons during projection if the **Apply GeoFeature Texture** preference is set in the *Projection Preferences* dialog box. **Apply Linear Texture as if an Areal**, which also is available in the *Projection Preferences* dialog box, causes linears without height to be textured using strip face texture. If this preference is cleared, linear features without height (2D features) are textured with global texturing. See "Setting Global Projection Preferences" on page 1144 for more information about these Projection Preferences.

Lower 3D Tops Below Wall Height

Projects the roof of an object slightly lower than the tops of the walls. This preference can be used to soften the edge along the tops of tree blocks that are projected into the database.

Library Substitution Preferences

Library Substitution Preferences control how high resolution and low resolution library substitution is performed for the selected Feature Type.



Scale Model to Feature Size

Scales each library model to match the size of the feature it replaces in the database. Scaling is exact in the case of point and two vertex linear features. In all other cases, scaling is based on the feature's bounding box. Because the bounding boxes of rotated linears and areals can be much larger than the original features, scaling can produce models that are larger than the features they replace.

Randomize Model Selection

Makes a random selection of one model from the list to replace each occurrence of the selected Feature Type. If this option is cleared, the currently-selected model is inserted for every occurrence of the Feature Type.

High Resolution Model Selected

Defines a set of models that can be substituted for the selected Feature Type when you choose **GeoFeature/Feature Projection/High Res Library Sub**.

Low Resolution Model Selected

Defines a set of models that can be substituted for the selected Feature Type when you choose **GeoFeature/Feature Projection/Low Res Library Sub**.

Use Model Path

Strips the path from every model in the list. If the model is not found in the current folder, Creator checks the **Model Path** in the *GeoFeature/Projection Preferences* window (see "Setting Global Projection Preferences" on page 1144).

Library Model Format

The following rules apply to the format of library models:

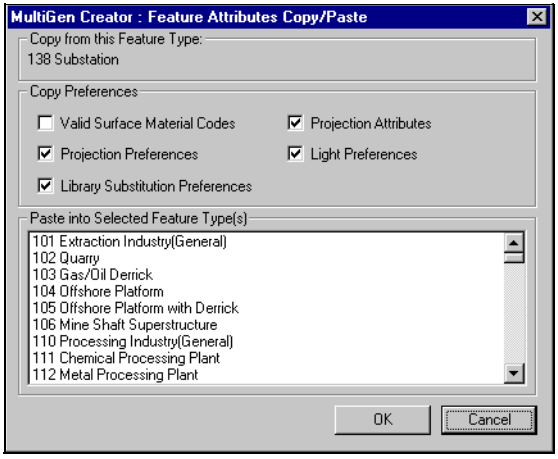
- Each model must reside in a separate file.
- If the model contains more than one top level node, only the first node is used. The top level node should be a group.
- The bottom of library models should be level. For example, in a model of a house, all vertices along the base should have equal *z* components.
- The rotation specified by a point feature's orientation attribute is applied to the model that is substituted for the point feature in the database.

Transferring Attribute Values

The same set of Feature Preference settings might be suitable for multiple Feature Types. To transfer attribute values from one Feature Type to another:

Step 1. In the **Feature Type** list, select the Feature Type from which you want to copy preferences.

Step 2. Click **Copy Values**. The *Feature Attributes Copy/Paste* dialog box appears.



Step 3. Set the checkbox next to each type of preference you want to copy.

Step 4. Select one or more Feature Types in the **Paste Into Selected Feature Types** list, and click **OK**.

Hold down the Ctrl key to select more than one Feature Type at a time or the Shift key to select a consecutive set of Feature Types.

Step 5. Click **OK** to transfer the preferences to the selected Feature Types.

Saving and Loading Feature Preferences

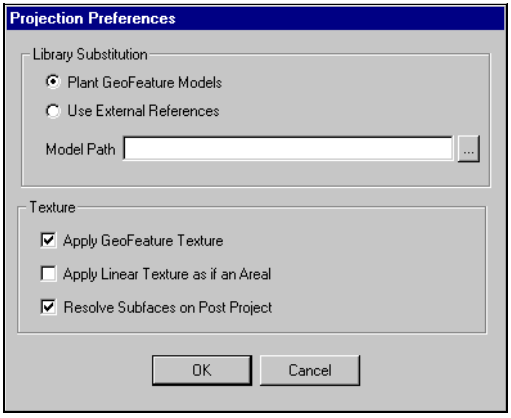
To save Feature Preferences in the default dfad.prefs file, click the **Save** button in the *Feature Preferences* window. To save the Feature Preferences in a different file, click the **Save As** button.

When you launch Creator, the Feature Preferences are always loaded from the directory that contains the Creator application. If you save a set of preferences to a location other than this directory, or if you save it to a file with a name other than dfad.prefs, you must reload the Feature Preference file each time you restart Creator. To do this, click the **Load** button in the *Feature Preferences* window and select the file.

Setting Global Projection Preferences

Projection Preferences set Library Substitution and Texture preferences that apply to all Feature Types. These preferences affect how library substitution is performed, and how texture is applied during manual and batch feature projection.

To set up Projection Preferences, select **GeoFeature/Projection Preferences**. The *Projection Preferences* dialog box appears.



Library Substitution Preferences

These preferences affect how library substitution is performed during manual and batch projection of features. Note that these preferences augment the **Library Substitution Preferences** in the *Feature Preferences* window (see "Feature Projection Preferences" on page 1138).

Model Path

Specifies the default directory path to the high and low resolution models that are available for library substitution. You can enable library substitution for individual Feature Types in the *Feature Preferences* window (**GeoFeature/Feature Preferences**). In this window, you choose specific library models for a particular Feature Type. If Creator cannot find the model in the directory path specified in the *Feature Preferences* window, it looks in the directory path you enter in **Model Path**.

Plant GeoFeature Models and Use External References

During library substitution, **Plant GeoFeature Models** inserts copies of the library models in the database. **Use External References** inserts library models as external references. See "Planted Models vs. External References" on page 1162 for a complete discussion of these options.

Texture Preferences

These preferences affect how texture is applied during manual and batch projection of features.

Apply GeoFeature Texture

Directs Creator to apply the textures specified in the Feature Preferences when cultural features are projected. If this option is set, features are textured after they are projected as 3D models. If this option is cleared, 3D models are created from feature data during projection, but the models are not textured.

Apply Linear Texture as if an Areal

Causes linears without height to be textured using strip face texture. *Strip face texturing* should be used to apply patterns that need to match along the center of consecutive faces, such as road stripes. If this preference is cleared, linear features without height (2D features) are textured with global texturing. *Global texturing* is used to apply a noise texture pattern (a texture pattern with no specific orientation) to a feature. See "3D Tops" on page 1136 for a complete description of global and strip face texturing.

Resolve Subfaces on Post Project

Cuts subfaces against each other. This option is set by default.

Creating Rules and Actions

When you project features automatically, Creator needs to know what features to extract from the data, how you want to project them, and in what order. This is done through rules and actions. Actions tell Creator what features to search for and how they should be projected. Rules, made up of one or more actions, tell Creator the order in which the features should be projected.

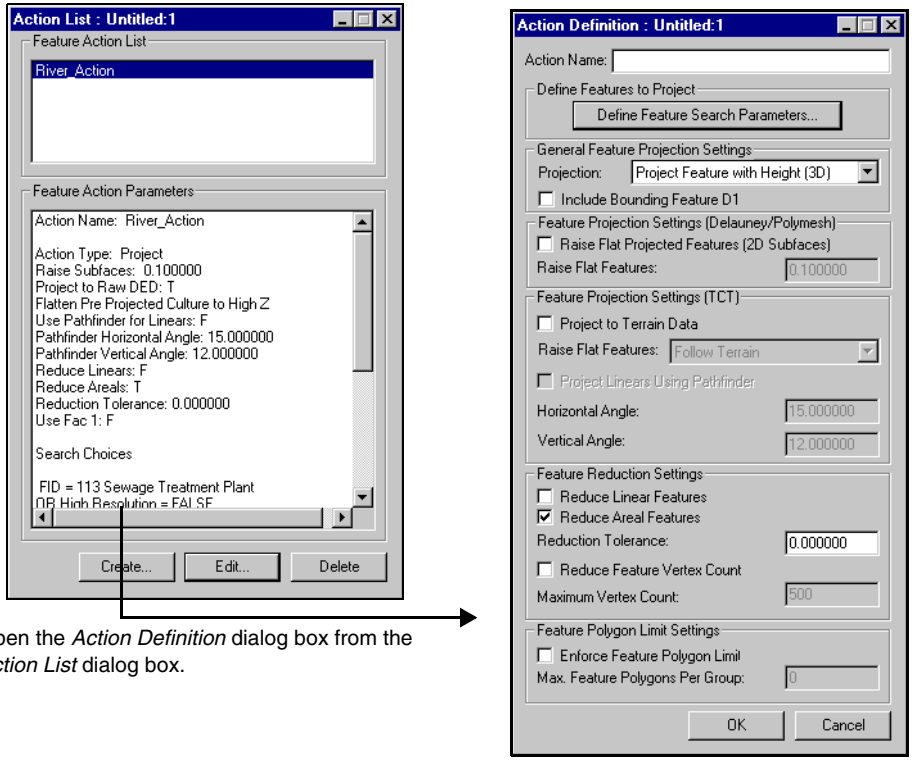
Creating Actions

An *action* defines a combined attribute search and feature projection or library substitution. Each action selects imported cultural features and projects them (or library equivalents) into a terrain database. Sets of actions are collected into *rules* and assigned to levels of detail for batch culture processing. (See “Creating Rules” on page 1153.)

To Create an Action

Actions are defined in the *Action Definition* window.

Step 1. In the database window, choose **GeoFeature/Rules and Actions/Feature Actions** to open the *Action List* dialog box, then click **Create** to open the *Action Definition* dialog box.



Open the *Action Definition* dialog box from the *Action List* dialog box.

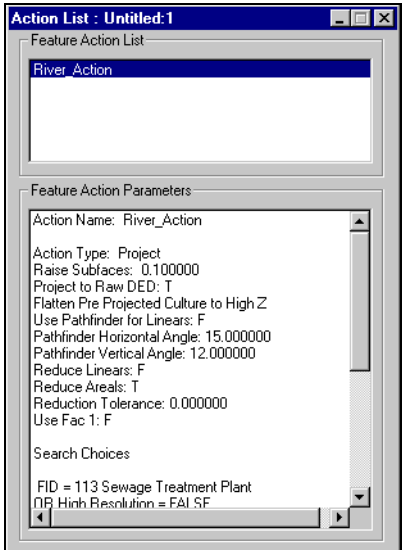
Step 2. Specify the **Action Name**. This name identifies the action when it is added to a rule. Now you can set the parameters that define the type of projection or library substitution and control the import of culture.

Step 3. Click **Define Feature Search Parameters** and set up an attribute search that selects features for import. (See "To Perform a Feature Attribute Search" on page 1123.)

Step 4. Click **OK** in the *Feature Search* dialog box. The action is saved as a feature preference and can be added to a rule, which is then assigned to an LOD.

Action List Preferences

Choosing **GeoFeature/Feature Action** opens a dialog box that lists each defined action. To display the parameters of an action, select it in the list.



Create

To create an action, click **Create** to open the *Action Definition* dialog box.

Edit

To modify an action, select it and click **Edit** to open the *Action Modify* dialog box. This dialog box is functionally identical to the *Action Definition* dialog box (see "Action Definition Preferences" on page 1148).

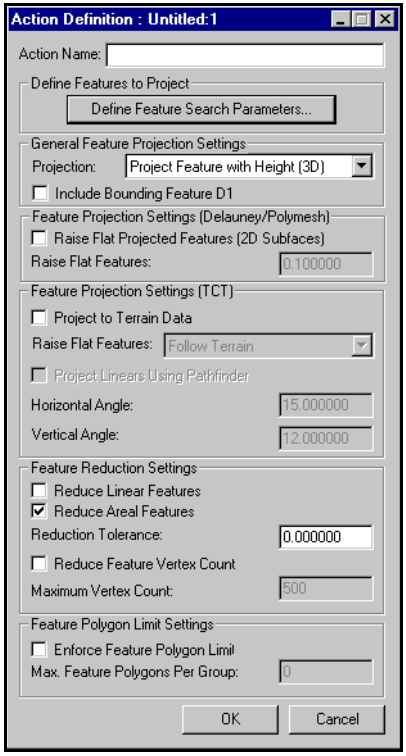
Delete

To delete an action, select it and click **Delete Action**.

If a rule references an action, the action must be disassociated from the rule before the action can be deleted.

Action Definition Preferences

Action Definition preferences control which features to look for and how they are to be projected.



Action Name

Defines the name of this action.

Define Feature Search Parameters

Opens the *Feature Search* dialog box to specify the features that will be affected by the action.

Projection

Selects the type of projection or library substitution to perform.

- **Project Feature with Height (3D)**
  Creates geometry representing each selected cultural feature (using its attributes including height), and projects the new geometry onto the underlying terrain

face. This can be used with TCT to project features to elevation data, or post project features to the terrain polygons.

- **Project Feature with Height (3D) and Cut In**

  Like **Project Feature with Height (3D)**, creates geometry representing each selected cultural feature (using its attributes including height), but also cuts the new projected geometry into the underlying terrain face. This is for post-projected features only.

- **Project Feature with Height (3D)/Cut In/Low Z**

  Like **Project Feature with Height (3D) and Cut In**, creates geometry representing each selected cultural feature (using its attributes including height), but projects the vertices of the geometry using the height (z value) of the lowest vertex, and then cuts the new projected geometry into the underlying terrain face. This is for post-projected features only.

- **Project Feature Walls Only**

  Creates geometry representing each selected cultural feature (using its attributes including height), and projects only the walls of the new geometry onto the underlying terrain face. This is for post-projected features only.

- **Project Feature Tops Only**

  Creates geometry representing each selected cultural feature (using its attributes including height), and projects only the tops of the new geometry onto the underlying terrain face. This is for post-projected features only.

- **Project Feature Flat (2D Subface)**

  Also creates geometry representing each selected cultural feature, but the new geometry is projected to the terrain without considering the height attribute of the cultural feature. This can be used with TCT to project features to elevation data, or post project features to the terrain polygons.

- **Project Feature Flat and Cut In**

  Like **Project Feature Flat (2D Subface)**, projects the new geometry to the terrain without using its height, and then cuts the geometry into the terrain. This is for post-projected features only.

- **Project Feature Flat/Cut In/Low Z**

  Projects the vertices of the new geometry to the terrain using the height of the lowest vertex, and then cuts the geometry into the terrain. This is for post-projected features only.

- **Substitute High Resolution Model/Substitute Low Resolution Model**

Replace each selected cultural feature with an OpenFlight model specified in the feature's Feature Preferences (see the following section for library substitution instructions).

Include Bounding Feature (D1)

Includes the first feature in the DFD file. The first feature is a rectangle that represents the extents of the DFD file, such as 902 Soil.

Feature Projection Settings (Delaunay/Polymesh)

The following projection preferences are only valid for projecting features onto Delaunay or Polymesh databases.

- **Raise Flat Projected Features (2D Subfaces)**
  Creates a sibling face during projection to the terrain polygon and raises the face above the terrain by the amount specified in **Raise Flat Features**. Raised faces can be combined into a T-strip (a realtime drawing structure). This preference is not recommended for z-buffer systems.
- **Raise Flat Features**
  Specifies the distance a subface is projected above its parent.

Feature Projection Settings (TCT)

The following projection preferences are only valid for projecting features onto TCT databases.

- **Project to Terrain Data**
  If selected, it projects the feature onto the elevation data before the terrain is generated around the feature. If this option is not selected, the feature is simply projected on top of the terrain skin.
- **Raise Flat Features**
  Determines how feature data is placed on the terrain. This checkbox only becomes active when **Project to Terrain Data** is set.

  Follow Terrain places the features on the terrain. Features are projected onto the terrain, then, using the given vertices, are broken up for vertical deviation and reduced using the reduction tolerance.

  Flatten to Low Z flattens the features to the lowest level in the terrain (within the feature). **Flatten to Low Z** could be used to cut a canyon in a terrain, for example.

For aerial features, such as lakes, this feature finds the lowest point on both the perimeter and inside the lake and puts all the vertices to that height. This is useful for setting lakes to the same elevation as the terrain.

For linear features, this finds the lowest latitude and longitude and sets all the vertices to the same height. This is useful for buildings, but should not be used for rivers, which must flow downhill. (Rivers are automatically set to flow downhill, if **Follow Terrain** is set.)

Flatten to Median Z flattens the features to the median level in the terrain (within the feature).

Flatten to High Z flattens the features to the highest level in the terrain (within the feature). Useful for setting the maximum height for cultural features like buildings.

If you set up an action that uses any of the flatten options, erroneous results can occur at junctions with other linears, as well as along area block boundaries. Use these options in situations like an airfield, for example, where the linear can be successfully projected with one of the flatten options. If you choose to apply flatten to miscellaneous linear features, mismatches will occur with abrupt changes in Z values.

- **Project Linears Using Pathfinder**

  Uses the Pathfinder algorithms to create Pathfinder entities from linear (roads and rivers and so on) and road features in the DFD data.

  Pathfinder roads are built to real world specifications. They are drivable roads that are appropriate for ground-based simulations.

  Project LInears Using Pathfinder can be used for linear features other than roads, such as rivers. For example, if this option is *not* selected, a river will curve only at the vertices of the linear. If the option is selected, the river will curve by the number of degrees specified in the **Horizontal Angle** field (see below). This creates more polygons, since the river edge and terrain edge must match at the edges.

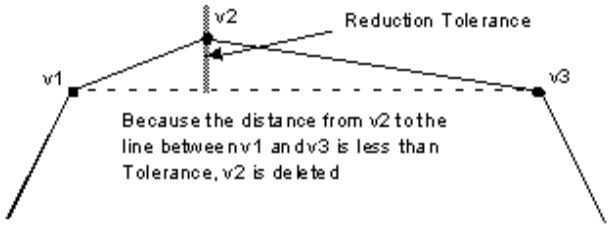
  Note: Project Linears Using Pathfinder only works with the TCT algorithm.

- **Horizontal and Vertical Angle**

  Specifies the angles for the polygonal break up of the linear features in the DFAD data when Pathfinder is used. Horizontal defines the degrees for a curve, and creates a new polygon when that limit is reached. Vertical is used for hills and valleys.

Feature Reduction Settings

- **Reduce Linear Features**
  Enables vertex reduction for linear features.
- **Reduce Areal Features**
  Enables vertex reduction for areal features.
- **Reduction Tolerance**
  Controls vertex reduction. If the distance between a vertex and the line segment connecting its two neighbors is less than **Reduction Tolerance** (in database units), the vertex is deleted.



- **Reduce Feature Vertex Count**
  Reduces each selected feature's vertex count by increasing **Reduction Tolerance** until the feature has no more vertices than the specified limit.

Feature Polygon Limit Settings

- **Enforce Feature Polygon Limit**
  Sets the specified limit on the number of polygons produced by this action to the number specified in the **Max. Feature Polygons Per Group** field. You can also place a polygon limit on rules. (See the *Online Help Summary; MultiGen Creator* for instructions on how to create Rules.)
- **Max. Feature Polygons Per Group**
  Sets the maximum number of polygons per group.

Creating Rules

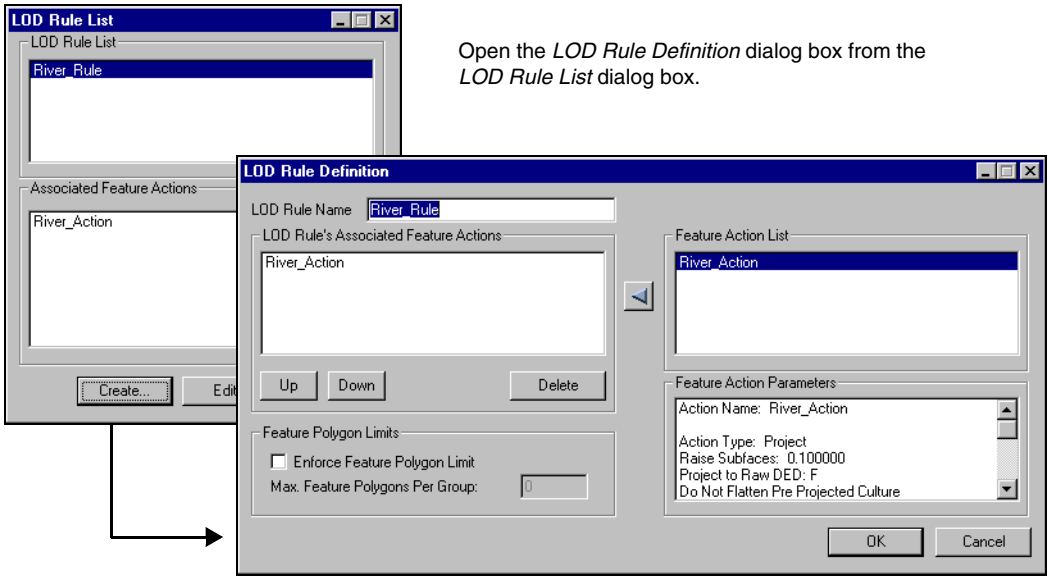
Rules are lists of actions that are performed, in order, during area block batch processing. You can assign one rule per level of detail in the *Terrain* window.

To Create a Rule

To define a new rule, do the following:

Step 1. Choose **GeoFeature/Rule List** to open the *LOD Rule List* dialog box.

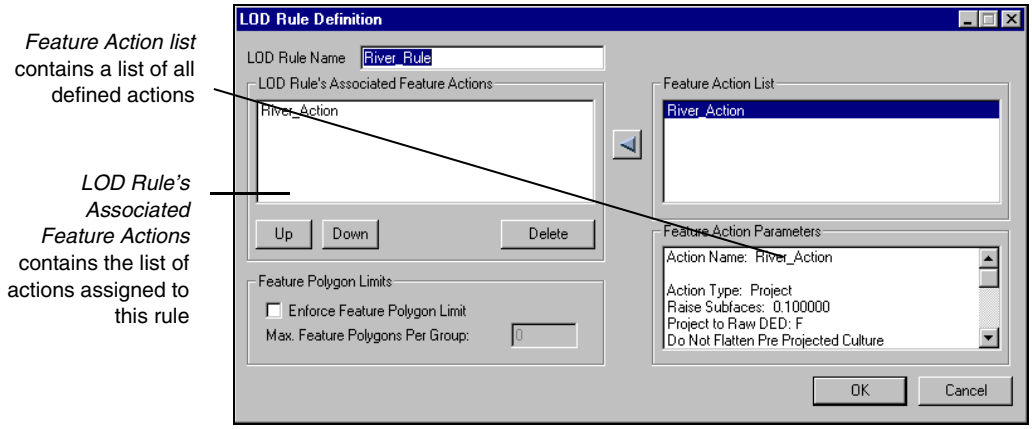
Step 2. Click **Create** to open the *LOD Rule Definition* window. Type a new name in the **Rule Name** field. This name identifies the rule when it is applied to an LOD during batch processing.



Step 3. Select one of the defined actions from the **Feature Action List**, and click the arrow to move the action to the **LOD's Rules Associated Feature Actions** list. Use **Up**, **Down** and **Delete** to set up the list of actions that will be performed, one after another, during batch processing.

LOD Rules Definition Preferences

The *LOD Rule Definition* dialog box allows you to create, modify, and delete rules by adding or removing actions to each rule.



LOD Rule Name

Specifies the name of the new rule.

Up

Moves the selected action up in the list.

Down

Moves the selected action down in the list.

Delete

Deletes the selected action from the **LOD Rule's Associated Feature Actions** list.

Enforce Feature Polygon Limits

Sets an upper limit on the number of polygons that can be created by executing the actions in this rule. Set the number in the **Max. Feature Polygons Per Group** field.

Insert Action Arrow



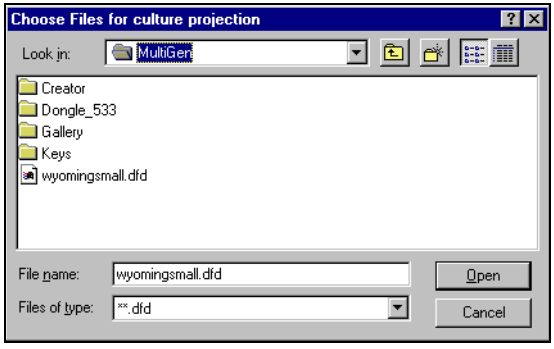
Inserts the action selected in the **Feature Action List** into the **LOD Rule's Associated Feature Actions** list.

Assigning Rules to LODs

The **Associated Rule** list in the *Terrain* window contains a list of all the rules you have defined. Each level of detail (LOD) can have one rule. As each LOD for the area blocks is processed, Creator uses the rule assigned to that level of detail to extract culture and add it to the terrain. To assign rules, do the following:

Step 1. Select **Enable Area Block Processing** and **Enable Feature Processing** in the *Terrain* window.

Step 2. In the Feature panel, Click **Browse** and select DFD files containing the cultural data.



Step 3. In the *Terrain* window, select one or more area blocks.

Step 4. Use the **LOD Number** list to choose a level of detail. LOD 0 is the highest resolution level of detail, LOD 1 the next highest, and so on. The number of levels of detail that will be generated during area block processing is determined by **LOD** preferences.

Step 5. Select a rule from the **Associated Rule** list. The selected rule is assigned to the indicated level of detail in all selected area blocks, and the name of the assigned rule appears below the **LOD Number**.

Step 6. To set up additional LODs for the same selected area blocks, repeat the previous two steps.

Document Rules and Actions

You can write the rules and actions to a text file by choosing **GeoFeature/Rules and Actions/Document Rules and Actions** and saving it with a file name of your choice.

This writes the current rules, actions, and LOD rule selections to a text file. The text file contains three sections: *Level of Detail List*, *Rule List*, and *Action List*. The *Level of Detail List* shows the rules assigned to each LOD. The *Rule List* shows the actions contained in each rule. The *Action List* shows the parameters contained in each action.

This text file is for your own information only, it cannot be imported back into Creator. However, you can reuse the dfadbat1.prefs file, which contains all of the rules and actions that you have defined for this project. See "The Project Panel" on page 1000.

Projecting Features Onto Existing Terrain

When you project features, you create OpenFlight models from the construction objects in the Feature layer, using the settings in the *Feature Preferences* window (**GeoFeature/Feature Preferences**), and under the control of the Projection preferences (**GeoFeature/Projection Preferences**). You can select features and then choose one of the commands in the **GeoFeature** menu to project it:

- **GeoFeature/Feature Projection/Project Feature(s)** creates 3D OpenFlight models from the selected feature data and places them on the terrain, in the LOD that currently is displayed. Features with height become walled objects. Features without height become subfaces of the terrain. The internal walls of areals are removed automatically, but the interior walls of multiple vertex linear features with height are not removed.
- **GeoFeature/Feature Projection/Project and Cut** creates 3D OpenFlight models from the selected feature data and places them onto the terrain polygons, in the LOD that currently is displayed. The feature data is then cut into the terrain.
- **GeoFeature/Feature Projection/Project and Cut Low Z** is similar to **GeoFeature/Feature Projection/Project and Cut**, except that the vertices of a feature are all projected using the height (z value) of the feature's lowest vertex. This is useful for features that are projected onto a sloped terrain, for example.
- **GeoFeature/Feature Projection/Project Walls Only** projects only the walls of 3D OpenFlight models from the selected feature data and places them on the terrain,

in the LOD that currently is displayed. This is useful for ground simulations or low levels of LODs in which the tops of the features will not be visible.

- **GeoFeature/Feature Projection/Project Tops Only** projects only the tops of 3D OpenFlight models from the selected feature data, in the LOD that currently is displayed. This is useful for flight simulations or low levels of LODs in which the walls of the features will not be visible.
- **GeoFeature/Feature Projection/Project Feature Flat** is similar to **GeoFeature/ Feature Projection/Project Tops Only**, except that the top of each feature is placed directly on the current LOD of the terrain. Choose **Project Feature Flat** in applications where height of features is not needed.
- **GeoFeature/Feature Projection/Project Flat and Cut** is similar to **GeoFeature/ Feature Projection/Project Feature Flat**, except that the features are also cut into the terrain.
- **GeoFeature/Feature Projection/Project Flat and Cut Low Z** is similar to **GeoFeature/ Feature Projection/Project Flat and Cut**, except that the vertices of each feature are all projected using the height (z value) of the feature's lowest vertex, and then the feature is cut into the terrain.
- **GeoFeature/Feature Projection/High Res Library Sub** and **GeoFeature/Feature Projection/Low Res Library Sub** place existing OpenFlight models on the current LOD of the terrain, instead of creating models from the feature data. The models that are substituted for projected feature are specified in the *Feature Preferences* window (**GeoFeature/Feature Preferences**).

This section describes how to use these manual methods of projecting features.

As an alternative, you can set up a batch feature projection in the *Terrain* window. This type of projection is controlled not only by the Feature Preferences file, but also by the Rules and Actions (see "Creating Actions" on page 1146 and "Creating Rules" on page 1153).

Note: If you select a Project and Cut tool on terrain that already has subfaces, all of the subfaces will be cut into the terrain.

Projecting Features Into Terrain

To project cultural data into a database, the geographical location of the terrain must correspond to the geographical area of the feature data.When you project features into a database, the software automatically correlates the position of these features with the geographical information in the database header attributes. Features are added to the

terrain group in which they are located, regardless of their places in the Feature construction tree. This insures that terrain and feature are arranged most efficiently for the realtime system.

To Project Features

Step 1. Display the terrain level of detail you want to populate and hide all other levels of detail. Projected features become part of the level of detail that currently is displayed. You must populate each level of detail separately.

Step 2. Select a set of features to project. You can do this manually or with a feature attribute search (see "To Perform a Feature Attribute Search" on page 1123).

Step 3. Choose one of the feature projection commands in the **GeoFeature** menu to add the feature to the database. See "Projecting Features Onto Existing Terrain" on page 1157 for descriptions of these feature projection commands.

Performing Library Substitution

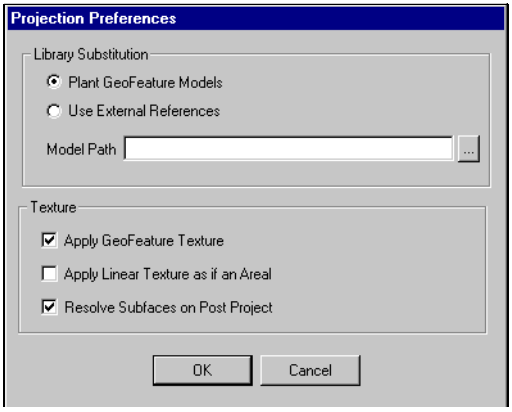
Library substitution replaces wireframe features with existing OpenFlight models instead of creating new 3D OpenFlight models from feature data attributes. Using library substitution to populate the terrain with hand-built models generally produces a more attractive database than projecting wireframe features. Two sets of preferences control library substitution: **Projection Preferences** and **Feature Preferences**.

Projection Preferences (**GeoFeature/Projection Preferences**) specify a default directory for library models, and determine whether the geometry of these models will be copied into the database during projection, or inserted as external references.

The **Library Substitution Preferences** in the **Projection Preferences** panel of the *Feature Preference* window (**GeoFeature/Feature Preferences**) determine which models are available for library substitution for a particular Feature Type, and whether the models should be scaled as they are projected into the terrain.

To Set Up Library Substitution

Step 1. Select **GeoFeature/Projection Preferences**. The *Projection Preferences* dialog box appears.



Step 2. In the **Model Path** text field, specify the directory path to the default high and low resolution models. Click the button next to the text field if you want to use the browser to find the correct directory.

This path will be used if Creator cannot find models in the directory specified in the *Feature Preferences* window. This is also the directory that appears when you click the Add button to add models in the *Feature Preferences* window.

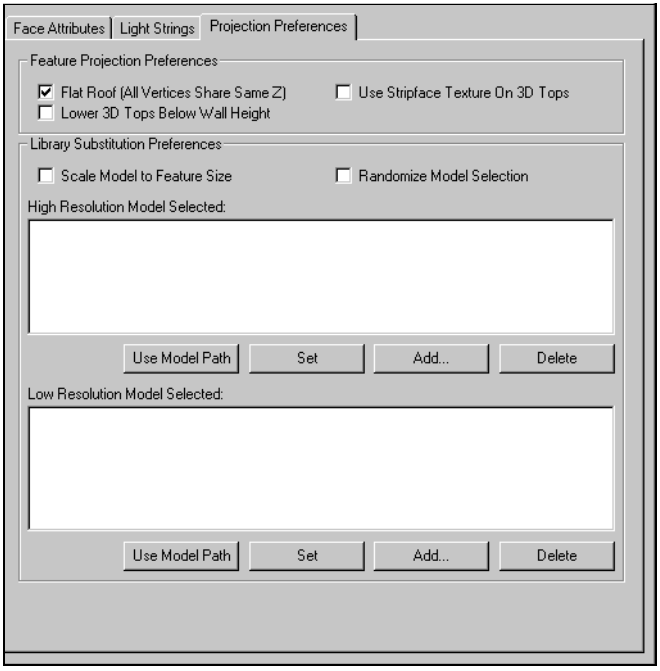
Step 3. Set **Plant GeoFeature Models** if you want to insert copies of the library models in the database. Set **Use External References** if you want to insert library models as external references.

Step 4. Click **OK** to close the dialog box.

Step 5. Select **GeoFeature/Feature Preferences**.

Step 6. In the *Feature Preferences* window, select the **Feature Type** for which you want to enable library substitution.

Step 7. Click the **Projection Preferences** tab to display its panel.



Step 8. Set **Scale Model to Feature Size** if you want to scale the model, as it is projected, to approximately the same size as the feature it replaces. (This is not always desirable; see "Scale Model to Feature Size" on page 1141.)

Step 9. Create lists of high resolution and low resolution models by clicking the **Add** button beneath the appropriate list and then selecting a model from the dialog box that appears. Repeat this step for each model you want to add. To delete a model from the list, select it and then click **Delete**.

Step 10. For each list, select a model and click **Set** to designate a single model that will be substituted for all features with the selected Feature Type. If you want Creator to select models at random for substitution, set **Randomize Model Selection**.

Step 11. Close the *Feature Preferences* window.

Step 12. In the database window, display the terrain level of detail you want to populate, and turn off the display of all other terrain faces.

Step 13. Select the wireframe features to be replaced with library models.

Step 14. Choose **GeoFeature/Feature Projection/High Res Library Sub** or **GeoFeature/Feature Projection/Low Res Library Sub**. The software uses the Library Substitution preferences of each selected feature to replace it with a model. Features that do not have appropriate Library Substitution preferences are ignored.

You can also perform library substitution during batch culture processing by creating an Action that specifies library substitution (see"Creating Actions" on page 1146.)

Model Rotation

Models substituted for point and linear features with two vertices are rotated as specified in the **Orientation** field of the feature's attribute page. Rotation is not applied to features with more than two vertices; instead, the model is placed in position with an orientation of 0.

Planted Models vs. External References

If the Projection Preference **Plant GeoFeature Models** is set (see "Plant GeoFeature Models and Use External References" on page 1145), library substitution opens each library model as it is needed, and inserts a copy of the model into the database. If **Use External References** is set, Creator uses the name of each library model to create an external reference to the model.

Keep in mind the following facts:

- **Using External References** requires less memory than planting (pasting in) feature models.
- The bottom vertices of external references are not planted to the terrain face, so externally referenced models may appear to float.
- External references use their own color, texture, and material palettes because the **Palette Override** external reference attributes are automatically set. You can force external references to use the palettes of the referencing database by manually clearing these options.

Planted Library Model Properties

The bases of polygons that result from library substitution are projected into the underlying terrain, but models are not instanced or clipped against adjacent polygons. If a feature intersects many terrain polygons, or if it lies above overlapping terrain polygons, this may produce geometry that needs to be corrected by hand.

Texture on library models is remapped during library substitution, as follows:

- If the model is scaled, or if planting changes its geometry, ***u***,***v*** values are modified to reflect the new geometry.
- Billboard faces are not retextured.

Planted library models use the texture, color, and material palettes of the master database. Use the **Attribute Resolver** function, described in the *MultiGen Creator User's Guide*, to ensure the palettes of all database files are consistent.

Pre-Projected Features

When features are pre-projected, the terrain is generated around them, so that they are cut into the terrain rather than being placed on top of it, as in post-projection. With TCT, post projected features will not project onto pre-projected features. For example, pre-projected roads will cut through a post-projected town.

Two types of features are used for pre-projection: OpenFlight models that are either cutout or footprint files, and Pathfinder features. You can pre-project cutout and footprint files, and Pathfinder features using TCT terrain generation, and only cutout files with CAT.

Cutout Files

Cutout files are OpenFlight models that have geocoordinates. The cutout files' longitude and latitude must correspond to the longitude and latitude of the terrain, so that it is positioned properly on the terrain. For example, to add an airport to a terrain, the airport's OpenFlight file coordinates in longitude and latitude must lie within the area defined by the terrain skin (you must change the latitude and longitude information in the *Header Attributes* window of the cutout file).

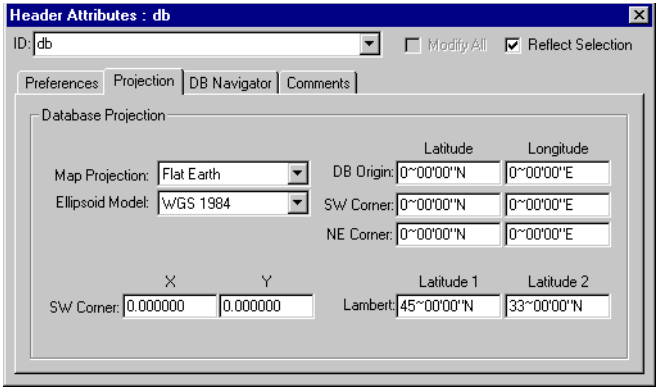
In CAT databases containing cutout features, terrain faces morph between levels of detail, but the feature faces are static. Because LOD switching is only performed on terrain, cutout features are generally used for modeling simple elements, or more complex things on simple ground planes such as airports.

To Change a Cutout File's Geocoordinates

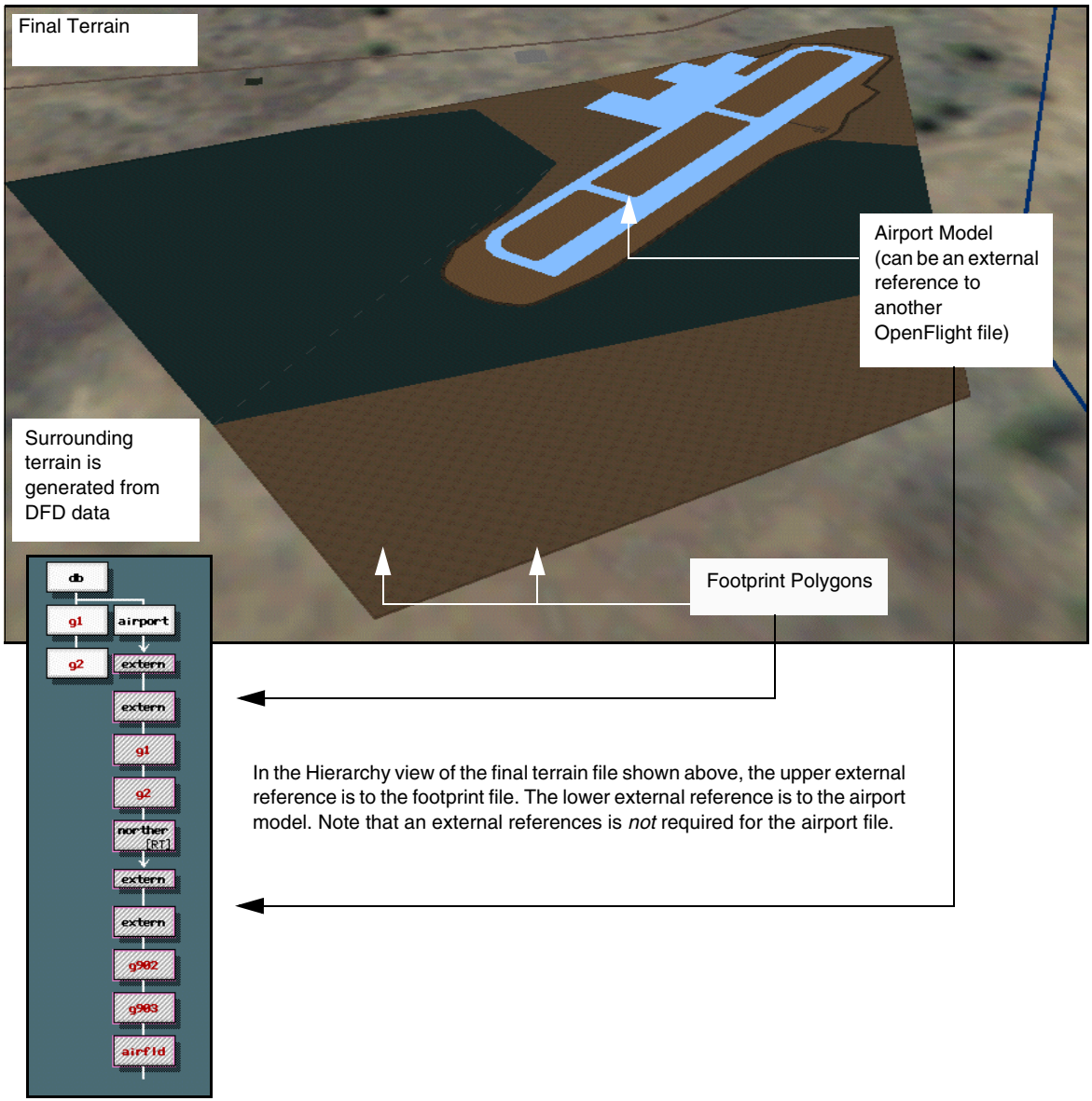
Step 1. Click the DB node in the Hierarchy view in the Database window.

Step 2. Choose **Attributes/Modify Attributes** to display the *Header Attributes* window.

Step 3. Click the **Projection** tab to display the Projection panel.



Step 4. Change the values in the **Database Projection** section, including the **Map Projection**, **Ellipsoid Model**, and **Latitude** and **Longitude** fields, to the same values for the terrain.



In the Hierarchy view of the final terrain file shown above, the upper external reference is to the footprint file. The lower external reference is to the airport model. Note that an external references is *not* required for the airport file.

Selecting Cutout Files for Processing

The **Cutout Feature List** in the Triangle panel of the *Terrain* window specifies OpenFlight files to include during processing, so that both the features from the feature data file and the objects in the OpenFlight files are cut into the terrain.

To add or delete a file in the **Cutout Feature List**:

Step 1. Click **Add** in the **Cutout Feature List** section to open the *Choose Files for TCT* (or CAT) *Cutout* dialog box.

Step 2. Browse the directory containing your cutout files, select a file, and then click **Open** to add it to the **Cutout Feature List**. To add multiple files, hold down Ctrl and click each file, then click **Open**.

To remove a file from the **Cutout Feature List**, select it and click **Delete**.

Footprint Files

TCT can also incorporate footprint files if library substitution is enabled through the Rules and Actions. A footprint file is a library substitution model to which a ground face has been added. This polygon's face attribute is set as a footprint. When defining the feature action, you set the **Project to Terrain Data** option, which tells Creator to pre-project the feature. When the terrain is generated, the file's footprint is cut into the terrain rather than placed on top of the terrain as in normal library substitution.

Unlike cutout files, footprint files do not depend on geocoordinates for placement. They derive their elevation from the terrain.

Pathfinder Features in TCT

Any linear feature data, such as a road, river or stream, can be automatically projected either as a linear or Pathfinder feature during TCT conversion. The Pathfinder option is set in the TCT **Feature Projection Settings** section of the *Action Definition* window.

Linear features are polygons that make up the feature's surface. Pathfinder features are splines with centerlines that can be driven using the **Drive Roads** command. During TCT processing, both linear and Pathfinder features are automatically connected to junctions and bridges as they are created.

Roads projected as Pathfinder features cannot be modified with the current Road Tools option. They are also expensive in terms of polygons, and should only be used in simulations that need them as drivable roads. Since the TCT algorithm allows only one LOD for feature data, you should insert your road in the LOD in which it will be used.

Note: You cannot post-project Pathfinder features onto a TCT terrain. Post-projected features can only be simple polygonized linears.

Creating Pathfinder Features

To project linear feature data as Pathfinder features:

Step 1. Choose **GeoFeature/Rules and Actions/Feature Actions** to open the *Action List* dialog box.

Step 2. Click **Create** to display the *Action Definition* window.

Step 3. Select **Project Linears Using Pathfinder**.

Project to Terrain Data must be selected to enable the **Project Linears Using Pathfinder** option.

Step 4. In the **Horizontal Angle** field, enter the maximum number of degrees that the road will change in azimuth before a new polygon is created.

Step 5. In the **Vertical Angle** field, enter the maximum number of degrees in pitch to bend before a new polygon is created.

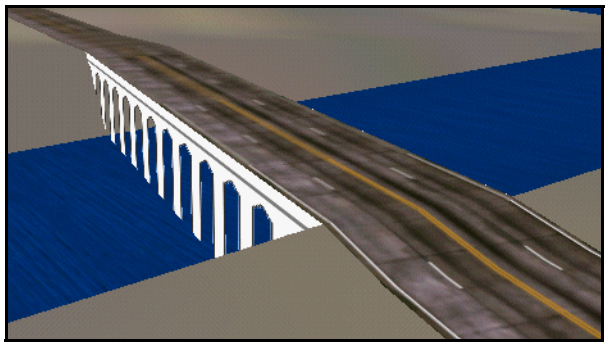
Step 6. Define any other parameters for the action.

Step 7. Include the action in a rule, and apply it using the procedure described in "Assigning Rules to LODs" on page 1156.

Applying Texture to Pathfinder Roads

When two Pathfinder roads intersect, one road is cut against the other. If a road crosses a river, a bridge is automatically built, using the texture for the 3D wall in the preferences file.

The texture you specify for the 3D Tops in the Feature Preferences window is the texture that is applied to the road surface. The 3D Walls texture is applied to the sides of elevated roadways and to texture the sides of bridges. By default, the top of the texture and bridge sides is set to the same height as the road.

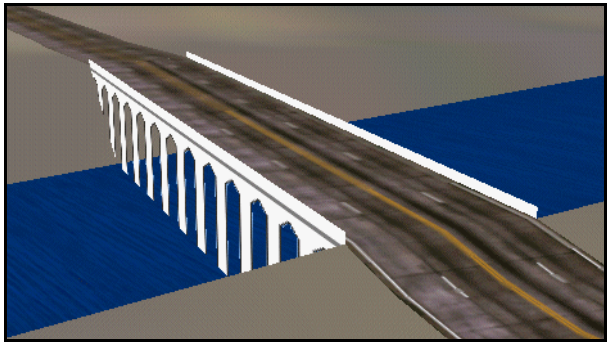


By default, the bridge sides are set to the same level as the road surface.

The total height of this texture is 128. The V pivot value is set to 128 by default.

The bridge sides here are set slightly higher than the road surface using the technique described below.

The texture's V pivot value in this example is 118.



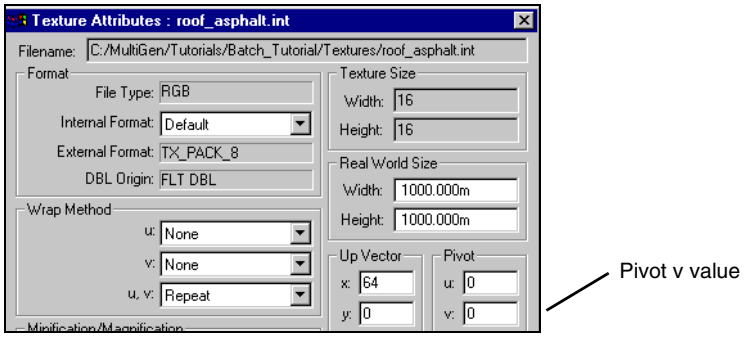
To position the texture higher or lower, change the **Pivot v** value in the texture attributes file. The **Pivot v** value specifies the number of texels placed below the polygon. Using the example above, if the total height of the bridge is 128, setting the **Pivot v** value to 118 will place 118 texels below the bridge and 10 texels above it.

To set the **Pivot v** value in the Texture palette:

Step 1. Open the Texture palette.

Step 2. With the bridge side texture selected, choose **Edit/Modify Attributes** (Ctrl+=).

Step 3. In the **Pivot v** field, enter the pixel value.



Step 4. Regenerate the terrain. The bridge side texture should now be aligned correctly with the road side. You may have to experiment to place the texture exactly where you want it.

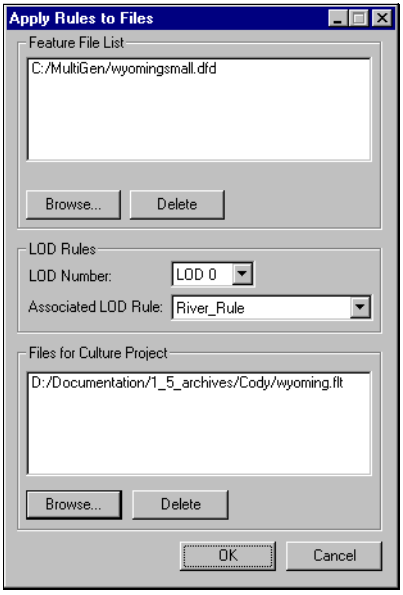
Batch Culture For Existing Terrain

Adding culture to existing terrain files is *not* done in the *Terrain* window. The **GeoFeature** menu in the database window contains menu items that allow you to extract culture and project it into one or more terrain databases in batch mode without performing area block processing.

To Add Features to Terrain Databases

To add features to one or more existing databases, do the following:

Step 1. Choose **GeoFeature/Apply Rules to Files** to open the *Apply Rules to Files* dialog box.



Step 2. Click **Browse** (**under the Feature File List**) to open the *Choose Feature Files* dialog box.

Step 3. Select the DFD files that contain the cultural features you want. Click **Open** to add the files to the **Feature File List**.

Step 4. Select an **LOD Number**.

Step 5. Select a rule in the **Associated LOD Rule** list. As in area block culture processing, only one rule can be assigned to each level of detail.

Step 6. To set up additional levels of detail, repeat the previous two steps.

Step 7. Click **Browse** (under **Files for Culture Project**) to open the *Select Files for Culture Project* dialog box.

Step 8. Select the terrain databases you want to populate with culture and click **Open**. You can select multiple consecutive files by holding down the Shift key while selecting them. Hold the Ctrl key to select non-consecutive files.

Step 9. Click **OK** to begin batch culture processing.

32 Batch Processing

You can create your terrain as either a single OpenFlight file or in a Batch process. The single file method is generally used for small areas of terrain. Batch processing was developed for large-area terrain databases.

The Batch process divides the terrain data, and the terrain that it forms from this data, into a grid of evenly-spaced area blocks. When you import data into the *Terrain* window, you can select as many or as few area blocks for processing as you want. Creator processes each area block as a separate OpenFlight file, and creates a single master file which contains external references to each of the single files. Batch processing can be used for small areas of terrain, but is particularly useful for large areas of terrain for the following reasons:

- The system does not have to process all the data at once - each area block has its own LOD structure and can be paged in differently.
- Each file created is the same size because the area blocks are the same size.
- Edge matching is preserved with adjacent files.
- Each area block can have its own local origin rather than being offset from the southwest corner of the elevation data. This prevents the flickering that can occur in the realtime system when offsets become so large that significant digits are truncated.

Batch processing is available only with the TerrainPro option. You can choose to batch process terrain only, terrain plus features, or features only. The following topics provide more information on these options. For a comprehensive explanation of how to create a terrain database, refer to the book *Creating Terrain For Simulations*, which is included with Creator in printed form, and as an Acrobat (.pdf) file on the Creator CD.

Batch Processing Basics

Area Block Batch Processing proceeds as follows:

- Import the terrain file to be processed. In the *Terrain* window, set the **Map** and **Triangle** preferences as described in Setting Panel Preferences, and set the batch processing preferences as described in To Create Terrain With Features. Then select the area blocks to be processed. An area block is a geographical partition

of the DED file (the terrain post file). **Batch** preferences in the *Terrain* window control the definition, selection, and processing of area blocks. When you have selected all the area blocks to process, click **OK** or **Apply** to convert the terrain.

- Each area block is processed into a separate terrain file and written to a specified directory. A master file is also created, which contains external references to each of the area block files. The master file takes the name of the terrain DED file, and has a .flt extension (for example, gcanyon.flt). If that name is already used, subsequent master files from the same DED file will be named gcanyonv1.flt, gcanyonv2.flt, and so forth.
- Each terrain file created contains its own geometry, LODs, and properties defined by the terrain preferences. You view them by opening the OpenFlight files (using either **Read Select** in the *Terrain* window or choosing **File/Open**), or by opening the master file to view the entire terrain (all processed area blocks).
- Area block preferences are saved using the directory path and the name of the DED file, the number 1, plus the extension .bpf (block preference file). For example, the block preference file for the terrain file gcanyon.ded will be gcanyon1.bpf. If gcanyon1.bpf already exists, a backup copy of the file will be made with a .bak extension and the new information will be written to gcanyon1.bpf.

Zooming in Batch Mode

Normally, when you select an area and set **Zoom** in the *Terrain* window, the selected area is resized to fit the whole **Gaming Area**. When you clear **Zoom**, it goes back to its original size. However, in Batch mode, zoom works differently.

When **Enable Area Block Processing** is selected, the mouse selects cells for processing, not for zooming. Therefore, **Zoom** doesn't do anything *unless you have used it to select an area prior to setting* **Enable Area Block Processing**. The following procedure shows you how to do this.

Step 1. In the *Terrain* window, select **Batch** and clear **Enable Area Block Processing**.

Step 2. Use your mouse to select an area in the **Gaming Area**.

Step 3. Set **Enable Area Block Processing**.

Step 4. Click **Zoom** to zoom in and out.

Step 5. To change the size of the zoomed area, you must clear **Enable Area Block Processing**, re-select the zoomed area and then set **Enable Area Block Processing** again.

Understanding Master Files

A master file is created each time you generate a terrain database in batch mode. The master file contains external references to each of the terrain databases generated from the corresponding DED file.

You can save the master file and the terrain databases in separate directories by defining a directory path for the terrain databases in the **Prefix** field or by selecting a different path for the master file in the *Browse for Folder* dialog box or the **Output Directory** field of the **Project** panel in the *Terrain* window.

The first time you generate a batch of terrain databases, the master file name is the same as the DED file. For example, bay.ded becomes bay.flt. Each time you generate a new terrain file, the master file name is incremented by one number. For example, bayv1.flt becomes bayv1.flt, bayv2.flt, and so forth. *Each new master file includes all the previous files generated from that DED file.* To start a new master file, copy the DED file into a new directory before you begin a new Batch process.

Working with Large Area Databases

Many graphics displays use a floating point value when switching terrain in and out of the eyepoint. Because the size of the area that can be switched in is limited by the floating point value, this can created some jerkiness in the display when you are flying over large areas of terrain. When the **Use Local Origin** checkbox is set, a local origin is assigned to each area block tile that sets the center of the tile's bounding box to 0,0. This results in much smaller floating point values and facilitates smoother and faster switching.

When processing terrain as a batch with **Use Local Origin**, in addition to the master file that is always created with a batch process, Creator also produces a Vega™ ADF file. The ADF file contains references to the actual locations of each area block. If Vega is installed on your system, you can double-click the ADF file to automatically launch the Vega LynX module, and then select **Preview/Active Preview** to launch a window with

mouse controls for moving the model. The ADF file may also contain motion, fog, and AOI information that is activated when LynX is launched.

Information relative to the actual real world location of each area block tile are also stored in the database's header for access by other realtime systems.

Area Block Naming Conventions

When processing area block tiles you can use one of four naming conventions for the output files:

- Each database is named with the actual latitude and longitude coordinates of the tile's lower left corner.
- Each database is named with a specified prefix, followed by a number corresponding to the tile's sequence in the batch process.
- Each database is named with the specified prefix, the row number and the column number of the area block within the geocell.
- Each database is named with the specified prefix, the column number and the row number of the area block within the geocell.

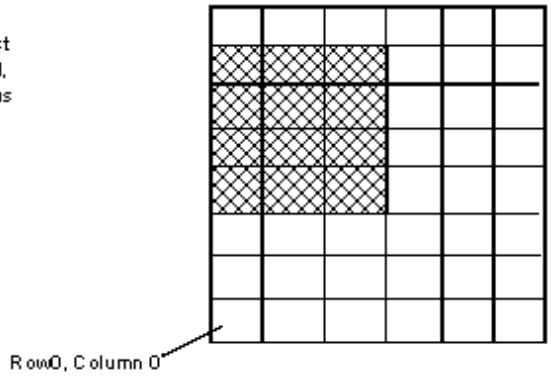
Additionally, when processing several batches of area blocks you can designate what the starting row and column number will be in the naming scheme (1st row:, 1st col: fields). The 1st row value is the row number assigned to the first area block in the lower left corner; the default value is 0. The 1st col value is the number assigned to the extreme left column; the default value is 0. The values entered can be either positive or negative. When 1st row and 1st col values are not specified, the area block in the lower left corner is considered Row 0, Column 0.

The example below assumes a **Use Local Origin** batch process using the prefix wy_e:

When using prefix/row/col, and 2st Col and 1st Row are not specified, the area blocks will be processed as follows:

wy_eRow003Col0000.flt to

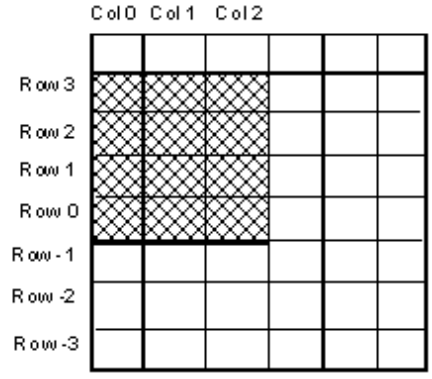
wy_eRow005Col0020.flt



When 2st Row is set to -3 and 2st Col is set to 0, the naming scheme would be as follows:

wy-eRow003Col0000.flt to

wy-eRow003Col0002.flt



To Create a Batch Terrain

Step 1. Choose **Terrain/New Project** and select a .ded file to import.

Step 2. Set up the colors, textures and materials that will be used in the new databases (as shown in Pro Terrain Conversion).

Step 3. Set the **Map**, **Triangle** and other conversion preferences (as described in Setting Panel Preferences).

Step 4. Select **Batch** in the *Terrain* window.

Step 5. Set **Enable Area Block Processing**.

Step 6. Specify the size of each area block in the **Area Block Size** fields.

Step 7. Set **Minutes** or **Posts** to control the units of **Area Block Size**.

Step 8. Set **Show Grid** to display a grid of area blocks in the gaming area. This step is optional.

Step 9. Set **Calculate Shading** to add dynamic shading properties to the new databases. Vertex colors are calculated whether or not **Calculate Shading** is enabled.

Step 10. Specify an **Area Block File Naming** method for the new databases. See "Area Block Naming Conventions" on page 1174 for more information about naming during batch processing.

Step 11. If using the **PrefixRow/#Col#** or **PrefixCol#Row#** naming method, enter a row and column number in the **1st Row** and **1st Col** fields.

Step 12. Select the area blocks to process in batch mode.

To select an area block, click it. Click it again to deselect it. Selected area blocks are overlaid with a red diagonal mesh.

- To select all area blocks, click **Select All**.
- To deselect all area blocks, click **Deselect All**.

Step 13. Click **OK** or **Apply**. (**OK** closes the *Terrain* window.) If you have not chosen an **Output Directory** path (on the **Project** panel), Creator prompts you for it the first time you run a batch file.

Note: If you type in a path in the **Prefix** field (which controls the location of the terrain databases), you may want to select the same path in the *Browse for Folder* dialog box so that the master file is placed in the same directory as the terrain databases.

Area blocks that have been processed appear faded in the gaming area. To open the database generated from one of these area blocks, select it and click **Read Select**.

Creator also creates a master file containing external references to all databases processed from this DED file in batch mode.

Batch Terrain With Feature Projection

Adding cultural features to area blocks processed in batch mode is different than when files are processed individually. With batch feature processing, Creator populates terrain databases as they are created from area blocks by extracting *features* (cultural data) from DFD files and then projecting the features into each database.

Since Creator is doing this automatically, it needs a set of feature actions and rules, to control the extraction and projection. *Actions* are projections and/or library substitutions that extract features according to attributes. *Rules* are lists of actions to be performed for each LOD during block processing. Rules can contain more than one action and are needed because only one rule can be applied to each LOD.

These are the general steps for creating terrain with features with cross-references to tools and techniques with which you should be familiar before starting. For more details, see To Create Terrain With Features.

1. Set up block batch processing for terrain. (See Batch Processing Basics.)
2. Set the Feature preferences that control feature projection and library substitution. (See "To Set Up Feature Preferences" on page 1131 and Performing Library Substitution.)
3. Open the *Action Definition* dialog box and set up search conditions that select actions for processing. (See Creating Actions.)
4. Open the *LOD Rule List* dialog box and set up rules that apply a sequence of actions to feature data. (See Creating Rules.)
5. In the *Terrain* window, select **Enable Feature Processing**, and use the **Feature** panel preferences to assign rules to selected levels of detail. (See Assigning Rules to LODs.)
6. Select area blocks to process and click **OK** or **Apply**.

To Create Terrain With Features

To perform feature conversion during batch processing:

Step 1. Select **Batch** in the *Terrain* window to display area block batch processing preferences.

Step 2. Set **Enable Area Block Processing**.

Step 3. To display a grid of area blocks in the contour display, set **Show Grid**.

Step 4. Specify the size of each area block using the **Area Block Size** fields. **Minutes** and **Posts** specify the unit of measure used. The value you enter in **Area Block Size in X** and **Area Block Size in Y** changes the size of the grid in the **Gaming Area**.

Step 5. To shade all new databases (by calculating vertex normals), set **Calculate Shading**. Vertex colors (for Gouraud shading) are calculated whether or not this checkbox is enabled. Choosing **Calculate Shading** at this point insures that the shading in all the databases will match.

Step 6. Specify a **Area Block File Naming Method** for the new databases.

Step 7. Select the **Feature** tab, and set **Enable Feature Processing (Batch Only)**.

Step 8. Click the **Browse** button and select a .dfd file whose coordinates correspond to the elevation data.

Step 9. Set up actions and rules, and assign culture conversion preferences to each area block.

Step 10. Select the **Batch** tab and then select the area blocks to process in batch mode.

- To select/deselect an area block, click it with the left mouse button. Selected area blocks are overlaid with a red diagonal mesh.
- To select all area blocks click **Select All** in the batch preferences.
- To deselect all area blocks click **Deselect All** in the batch preferences.

Step 11. Click **OK** to process the area blocks and close the *Terrain* window, or click **Apply** to process the area blocks and leave the *Terrain* window open.

You can also use rules to add cultural data to a set of existing terrain databases. This is done from the **GeoFeature** menu. (See "To Project Features Without Terrain" on page 1179.)

Projecting Features Without Terrain

When a linear feature, such as a road, comes too close to an aerial feature, such as an airport, the terrain between the road and the airport may show gaps in the terrain skin. Errors can also occur due to library substitution, when the model that is substituted is larger or wider than the actual feature data.

The solution is to correct the features before the terrain is processed.

- Correct the feature data that is projected from feature data, save the corrections by exporting the modified data to a new DFD file, and then run the triangulation.
- If the conflicting features are in a single cutout file, correct the file prior to the triangulation.
- If the conflicting features are in different cutout files, combine the cutout files prior to the triangulation.

Using the TCT algorithm's **Project Features Only (No Terrain Polygons)**, you can project features without generating terrain polygons and view them in the Graphics view. Features that overlap or appear too close to other features appear in red so they can be easily seen. You can use Creator's tools to move and modify the feature data and then export the corrected data to a new DFD file.

Note: When projecting features without terrain, bridge polygons are automatically assigned a FID of 260 Bridges (General). When a conflict occurs between features and one of the features has a FID of 260, the polygons do not appear in red.

To Project Features Without Terrain

This procedure generates an OpenFlight file of projected features only.

Step 1. Choose **Terrain/New Project**.

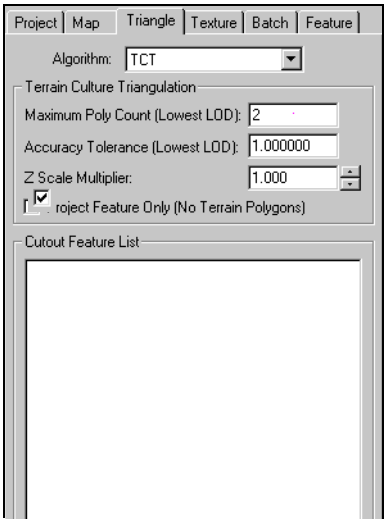
Step 2. In the *Open Terrain File* dialog box, select your DED file and click **OK**. The *Terrain* window opens and displays the Project panel preferences.

Step 3. In the **Project** panel, set the paths for Features, Rules and Actions, and any palettes to be used.

Step 4. Click the **Triangle** tab and set **Algorithm** to **TCT**.

Step 5. Set the **Accuracy Tolerance** and **Maximum Poly Count** fields, if desired, or leave them at the default settings.

Step 6. Set **Project Feature Only (No Terrain Polygons)**.



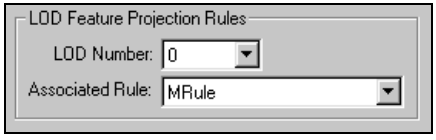
Step 7. Click the **Batch** tab and click **Show Grid** to display the grid in the **Gaming Area**.

Step 8. In the **Gaming Area**, choose the section of the terrain to process by selecting and deselecting area blocks.

Step 9. Click the **Feature** tab and set **Enable Feature Processing**

Step 10. Click **Browse** and choose the DFD files from which to extract features.

Step 11. Set the rule(s) for LOD 0 (zero). First, click **LOD Number** (if necessary) and select 0 (zero). Then select an **Associated Rule**.

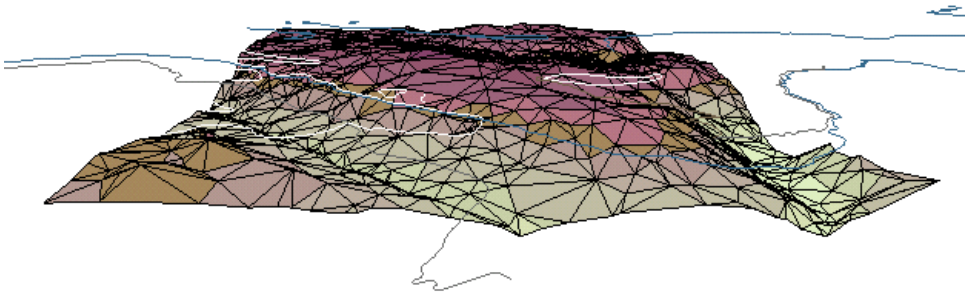


Step 12. Click **OK** or **Apply** to process the selected area blocks.

Step 13. Open the OpenFlight file (open the master file if you processed multiple area blocks) to bring feature data into the Graphics view and locate the conflicts. Only the feature data will be shown.

If the selected features are not found in the DFD files, no database window will be opened. Change your search parameters and try again.

Step 14. Choose **GeoFeature/Import/Import DFD** and import the DFD file used in the terrain generation.so that the feature data overlaps the OpenFlight file.



Step 15. Modify the feature data as needed using Creator tools.

Step 16. Choose **GeoFeature/Export/Export DFD** to export the feature data. Be sure the data is exported with a.dfd file extension.

Step 17. Re-generate the terrain as either TCT or CAT, using the modified DFD file.

33 Applying Texture to Terrain

Textures add visual realism to your terrain database. A detailed texture pattern can be used in place of actual features in lower levels of detail when the database scenario doesn't require interaction (because collision detection can't be programmed into textures). This can save polygons that can be used elsewhere in your simulation.

There are several ways to apply texture to a terrain database. One way is to assign a texture to each elevation in the Contour palette. There are easier ways to apply texture patterns to terrains of all sizes, however, by specifying the texture files to be applied and the areas to which they should be applied. This can be accomplished by:

- Applying a single indirect texture or geospecific texture when generating the terrain database
- Combining a set of geospecific textures into one large image, and generating clip textures, mipmaps, and high resolution insets for Infinite Reality systems
- Applying a series of geospecific textures to specified OpenFlight files in a batch process

The following topics describe these methods and how to use them.

Applying Texture With the Contour Palette

You can apply a texture pattern to one or more elevation contours in the **Elevation Scale** in the *Terrain* window. These textures are then applied to the polygons at corresponding elevations when the terrain database is generated.

To apply texture to an elevation, do the following:

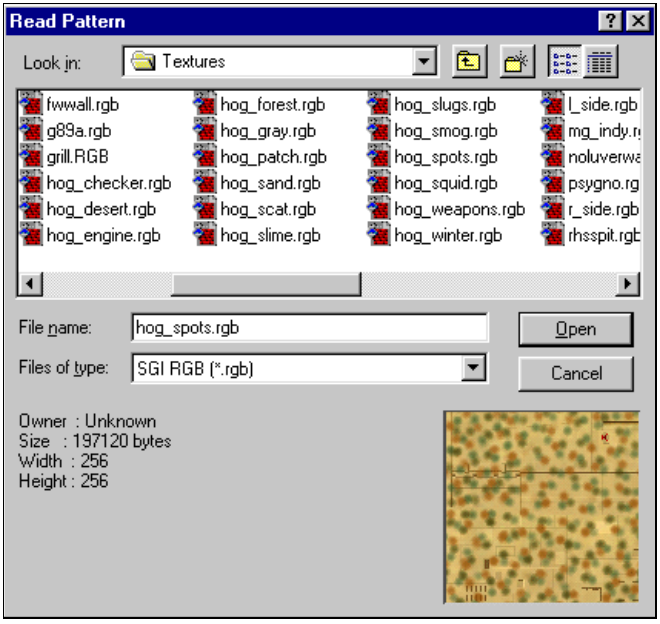
Step 1. In the *Terrain* window **Gaming Area**, drag-select the terrain to be converted by fencing it with the mouse.

Step 2. Select **Texture** from the **Contour Properties** menu.

Step 3. Double-click a contour in the **Elevation Scale** to open the Texture palette, or select a contour and then open the Texture palette by clicking the **Current Texture** button in the Toolbar.

Step 4. Choose **File/Read Pattern**. The *Read New Pattern* dialog box appears.

Step 5. Click **OK** to accept the default pattern placement and display the list of textures.



Step 6. Select a texture pattern and click **Open**. The selected texture is placed in the Texture palette and also in the **Elevation Scale**.

Step 7. Repeat Steps 3 through 6 to assign more textures to contours, if desired.

Step 8. Set the **Triangle**, **Map**, **Project**, **Batch** and **LOD** parameters as desired, and then click **OK** to create the database.

Step 9. In the Database window, open the View panel and click **Show Texture** to view the textures.



Using Indirect Texture

Indirect texture is a method for assigning texture, color, and material properties to terrain using a bitmapped image as a color key and an ASCII text file that associates each color in the image with a specific group of properties.

You can apply an indirect texture to USGS material class usage maps or satellite photographs, and create richly textured landscapes from relatively low-resolution images. You can also use indirect texture to paint imaginary worlds for a gaming environment.

You must create two files before applying an indirect texture:

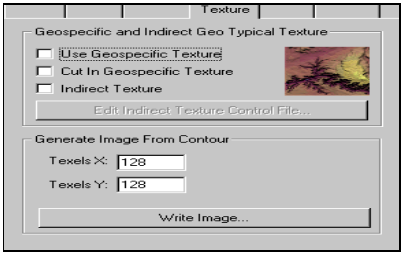
- An image file that defines where specific colors, textures, and materials to use
- An indirect file. This is an ASCII file that defines how the colors, textures and materials will be applied to the polygons in the terrain file being imported.

  Each color in the indirect texture refers to a line in the ASCII file that calls out a specific texture, color, material, and LOD to be assigned to that area of the terrain.

Both the indirect texture and the user-defined ASCII file must be located in the same directory.

Once you create these files, you apply them by selecting the **Indirect Texture** checkbox in the Texture panel in the *Terrain* window. When this checkbox is set, the current texture file in the Texture palette is used as the indirect mapping image. Clicking the Indirect Texture sample box opens the Texture palette where you can choose the image file you want to apply.

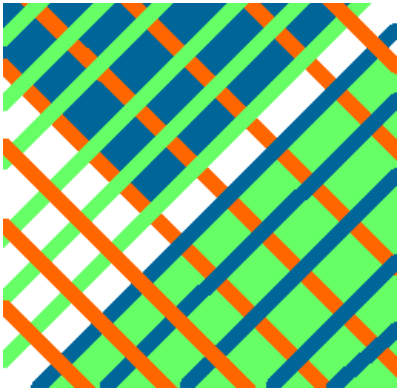
The image file is fit to the exact limits of the terrain file being imported and is projected onto the created polygons. When two or more vertices of a polygon fall on the same color, that color is used to assign properties to the polygon.



To Create an Indirect Texture

Step 1. Use a painting application to create a bitmapped image of the geographic area that will be generated as terrain, or open an image file of a land class usage map.

Use different colors to paint each area where you will be assigning different groups of properties. Record the RGBA values for each color, and note the texture, color and material properties you want each color to represent.



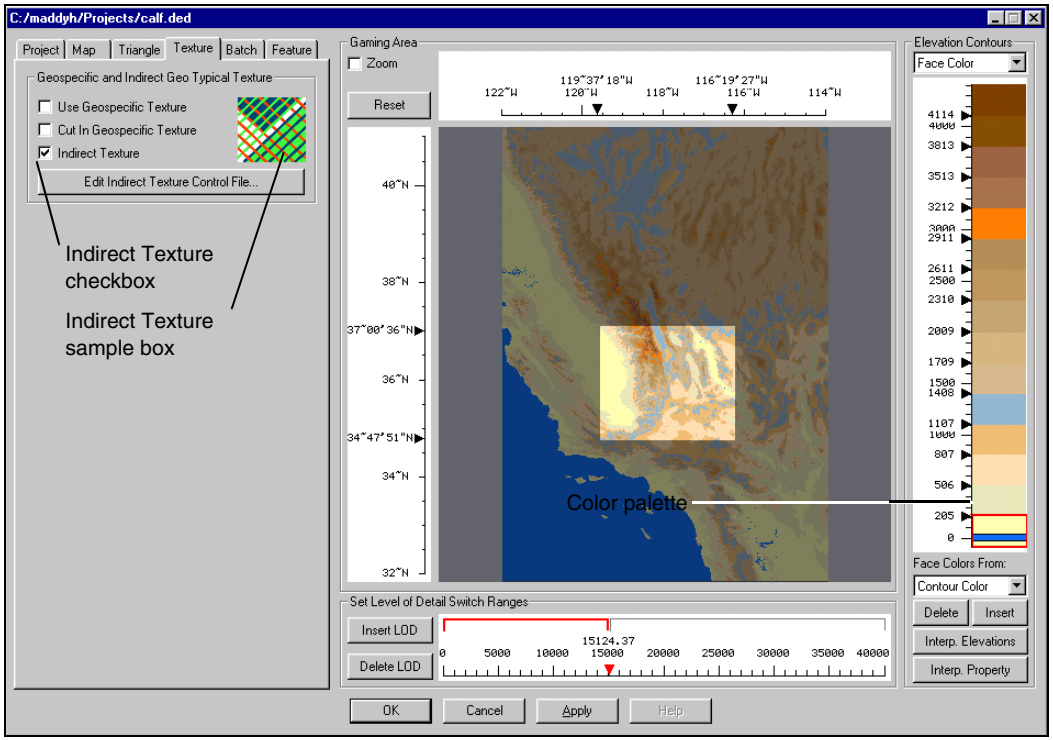
Step 2. Save the image. CTS supports most commonly used image formats.

To Apply an Indirect Texture

Step 1. Choose **Terrain/Import Elevation**. A dialog box opens and prompts you to select a terrain file to open.

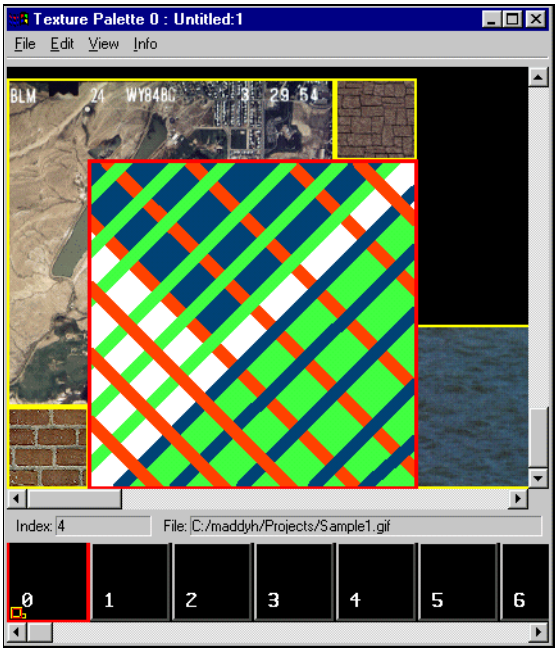
Step 2. Choose a DED file from the list, and click **OK**. The *Terrain* window opens.

Step 3. Check the **Indirect Texture** box and then click the **Indirect Texture** sample box beside it to open the Texture palette.

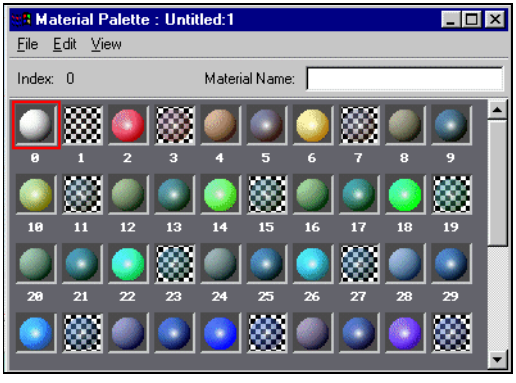


Step 4. Load all the textures you want to assign to the terrain into the same Texture palette. Record the palette index for each texture; you will need them when you prepare the table in Step 8.

Step 5. Load the indirect texture file into the Texture palette. This must be the selected texture when you generate the terrain.



Step 6. Open the Material palette and load all materials you want to assign to the terrain. Record the palette index for each material; you will need these indices when you prepare the table in Step 8.



Step 7. Optionally, you can save the Color, Texture, and Material palettes so that they can be reused.

Step 8. Use a text editor to create an ASCII file using the following as an example.

;Indirect texel color				Texture	Texel	Texel	Color	Matl.	LOD
;R	G	B	A	index	sizex	sizey	index	index	
0	255	255	255	0	20.0	20.0	127	-1	0
220	96	0	255	2	20.0	20.0	127	-1	0
0	135	64	255	3	20.0	20.0	127	-1	0
0	0	255	255	1	20.0	20.0	762	-1	-1
220	96	0	255	-1	20.0	20.0	3327	-1	-1
0	135	64	255	-1	20.0	20.0	62175	-1	-1

Colors in Indirect Texture

Properties to be assigned

Note: The Texel sizex and Texel sizey values override the Texture attributes' Real World values.

A -1 in the Texture, Color, or Material index column indicates the absence of that property. A -1 in the LOD column indicates that properties listed on that line are to be used in all LODs that are not otherwise specified.

Each color's RGBA value is associated with the properties that the color represents. Each line of the file must contain the following fields, in this order:

- Texel color red value
- Texel color green value
- Texel color blue value
- Texel color alpha value (if using only RGB values, alpha value is always 255)
- Texture palette index for the texture you want to assign to the area (-1 is no assignment)
- Texel sizex of the texture you want to assign to the area

- Texel sizey for the texture you want to assign to the area
- Color palette index for the color you want to assign (127 is white)
- Material palette index for the material you want to assign (-1 is no assignment)
- Level of detail to which the properties are assigned (-1 means all LODs)

Step 9. Save the ASCII file in the same directory as the indirect texture. Use the same root name that you assigned to the indirect texture, and assign a suffix of `indirect`. For example, if you named the indirect texture `texture_code.rgba`, then name the ASCII file `texture_code.indirect`.

Step 10. Click the **Triangle** tab and choose either the Polymesh or Delaunay terrain conversion algorithm

Step 11. Set LODs and any other terrain conversion preferences in the *Terrain* window, if desired.

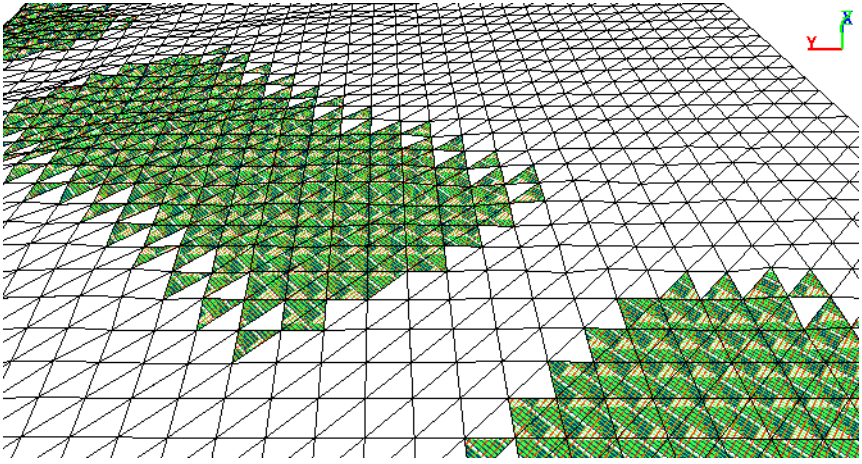
Step 12. Click **OK**. The *Write Terrain Database File* dialog box opens prompting you to specify a directory and file name to write out the terrain file.

Note: To ensure that the indirect texture is correctly applied to the terrain, your sampling of source data should be high enough to create sufficient polygons to cover the area you want textured. Experiment on a small area until you achieve satisfactory results. When using Delaunay, experiment with error correction multipliers until the area is sufficiently triangulated to reflect your indirect texture. Use caution when applying indirect texture to multiple LODs: Your textured area can change drastically with a reduction of polygons at lower LODs.

Step 13. Set the path and enter a name in the **File Name** field, then click **Save**. The terrain appears in the Graphics view at the lowest level of detail.

Step 14. Click **Show Texture** to view the textures you assigned to the terrain with Indirect Texture.





Step 15. Choose **More Detail** (Ctrl+M) and **Less Detail** (Ctrl+L) from the **LOD** menu to switch levels of detail (if you generated more than one level of detail).

Applying a Geospecific Texture

A *geospecific texture* contains geographical attribute information (the image's latitude and longitude) that indicates where the texture pattern belongs. Typically, geospecific textures are used to apply satellite photographs to large terrain areas. Because CTS automatically correlates geospecific attributes with database coordinates, a geospecific texture can be applied automatically during terrain conversion.

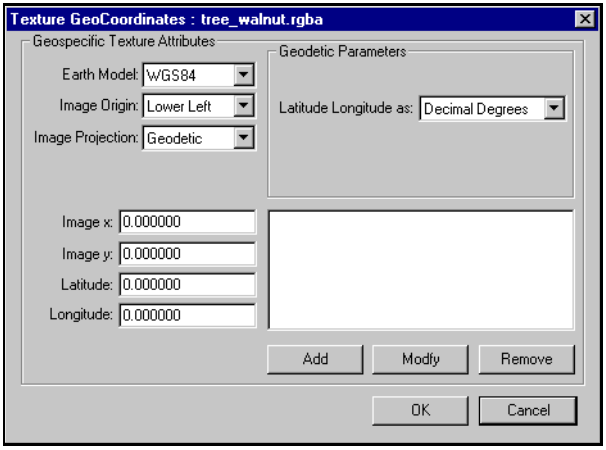
The geospecific texture is loaded into the Texture palette as the current texture. Although you can use a government terrain map (generally in TIF format), which has latitude and longitude information associated with it, your geospecific texture file can actually be any bitmapped image format such as TIF, RGB, RGBA, GIF, or INT. However, the texture pattern's geospecific coordinate attributes must match those of the terrain in order to be properly mapped.

Geospecific textures can be applied in three ways:

- In the *Terrain* window at the time the terrain is generated
- Using the **GeoPut Texture** tool
- Using **Batch GeoPut** in the **Terrain** menu

Setting Geospecific Information

In order for a geospecific texture to be applied correctly, the texture's geospecific information must exactly match the terrain to which it is being applied. You do this by modifying the texture's attributes. Choose **Edit/Modify Attributes**, in the Texture palette menu, then click **Geospecific Coordinates** in the *Texture Attributes* window to display the geospecific information.

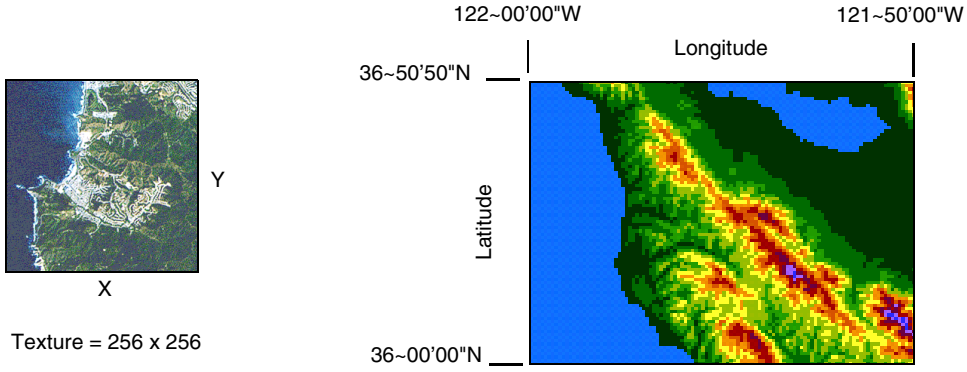


The texture is mapped to the terrain using a 3-Point Put Texture; that is, three control points that correspond to points on the terrain. The information you must provide is the starting origin, and the latitude and longitude for the three control points.

The discussion that follows uses the following examples:

- A texture having an image size of 256 x 256

- A terrain having a latitude of 36~00′00"N to 36~50′00"N and a longitude of 122~00′00"W to 121~50′00"W



The location of the origin (*x*, *y*) for each point depends on the size of the texture. If the mapping is to start from the lower left (southwest) corner of the terrain, the origin would be x=0, y=0. The coordinates of the terrain's southwest corner are 36~00′00"N latitude by 122~00′00″W longitude. So the geodetic parameters for the first point are: X=0.000; Y=0.000; Latitude=36~00′00"N; Longitude=122~00′00"W.

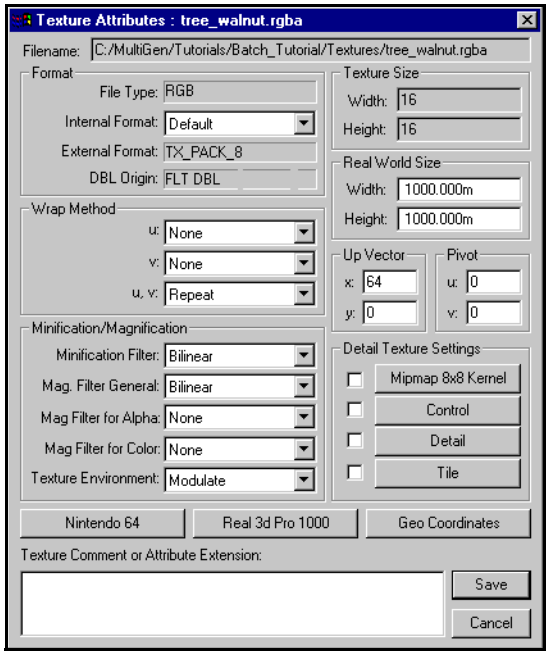
The texture's lower right corner (x=256, y=0) becomes the origin for the terrain's southeast coordinates. The terrain's coordinates are 36~00′00"N latitude by 121~50′00"W longitude.

The texture's upper right corner (x=256, y=256) is the origin for the terrain's northeast coordinates. The coordinates are 36~50′00"N by 121~50′00"W.

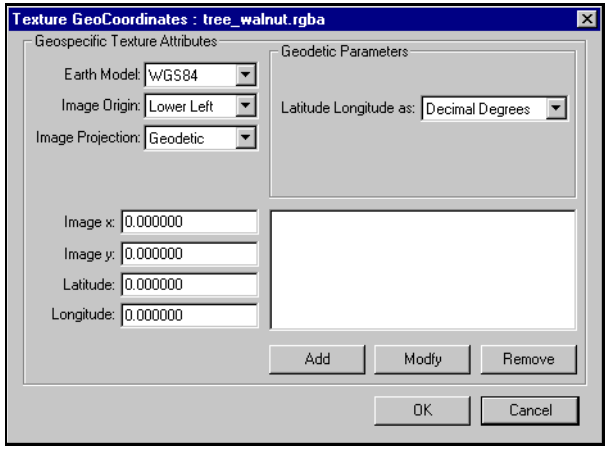
To Set a Texture's Geospecific Coordinates

Step 1. Load the geospecific texture you want to apply into the Texture palette. This must be the current texture.

Step 2. Choose **Edit/Modify Attributes** from the Texture palette menu. The *Texture Attributes* window opens.



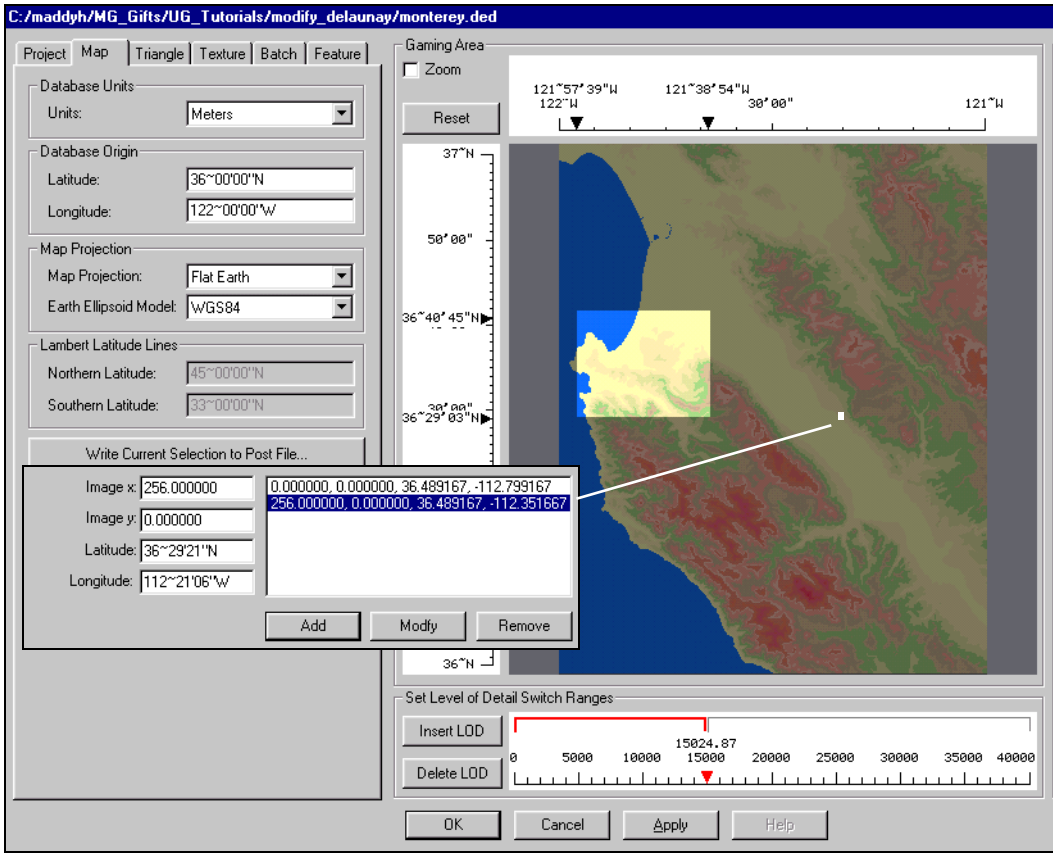
Step 3. Click **Geospecific Coordinates** to open the *GeoCoordinates* dialog box.



Step 4. Set **Earth Model** and **Image Projection** to match the selected terrain (**Map** in the *Terrain* window). Leave **Image Origin** set to **Lower Left**.

Step 5. Set **Latitude Longitude** to **Deg Min Sec**.

Step 6. Enter the geospecific coordinates of the terrain's southwest corner in **Image X**, **Image Y**, **Latitude** and **Longitude**, click **Add** to add the coordinate (see "Setting Geospecific Information" on page 1192). The information appears in the box to the right of the **Image X** field.



To change geodetic information after it has been added, select it. The information reappears in the **Image X**, **Image Y**, **Latitude** and **Longitude** fields.

Edit the information, and then click **Modify**. To delete a coordinate, select it and click **Remove**.

Step 7. Repeat Step 6. for the southeast and northeast corners of the terrain, and then click **OK** to set the geospecific coordinates and return to the *Texture Attributes* window.

Note: When **Image Origin** is set to **Upper Left**, **Latitude** and **Longitude** must correspond to the northwest, northeast, and southeast corners of the terrain.

Step 8. Click **Save** to save the changes to the texture attributes. You are now ready to apply your geospecific texture.

Applying Texture in the Terrain Window

Applying a geospecific texture during terrain generation consists of setting parameters in the Texture panel of the Terrain window. The texture is applied to all levels of detail. Geospecific textures are frequently used together with a "noise" texture covering areas of the database that do not require a lot of visual detail.

To Set Terrain Parameters

To apply a geospecific texture in the *Terrain* window:

Step 1. Choose **Terrain/Import Elevation**. The *Import Terrain File* dialog box opens.

Step 2. Select a `.ded` file to import, then click **Open**. The *Terrain* window opens.

Step 3. Select the portion of the terrain to be converted in the **Gaming Area**.

Step 4. If you are using other texture patterns along with the geospecific texture, open the Texture palette and load them.

Step 5. Assign properties to the Contour palette if you wish. Texture patterns in the Contour palette are applied to faces that fall outside the geographical area of the geospecific texture pattern.

Step 6. Click the **Texture** tab, click the texture thumbnail to open the Texture palette and choose the geospecific texture to be applied. Close the Texture palette.

Step 7. In the Texture panel, set **Use Geospecific Texture**. To slice faces in the database along the boundaries of the geospecific texture, set **Cut In GeoSpecific Texture**. This prevents the texture from only covering part of a face.

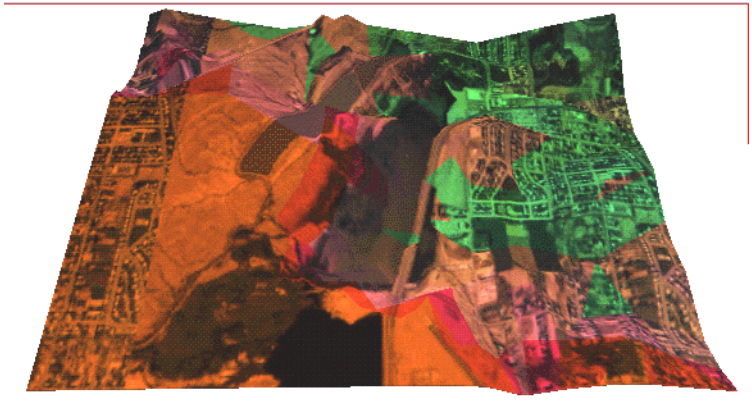
Note: **Cut In GeoSpecific Texture** is only available for Polymesh and Delaunay.

Step 8. Choose a conversion algorithm (Polymesh, Delaunay, or CAT), and set the LOD information, **Map**, **Triangle** preferences, and any **Batch** information, if necessary.

Step 9. Click **OK** or **Apply** to generate the terrain database with geospecific texture.



Step 10. Turn on **Show Texture** (T) to view the geospecific texture.



The Batch GeoPut Utility

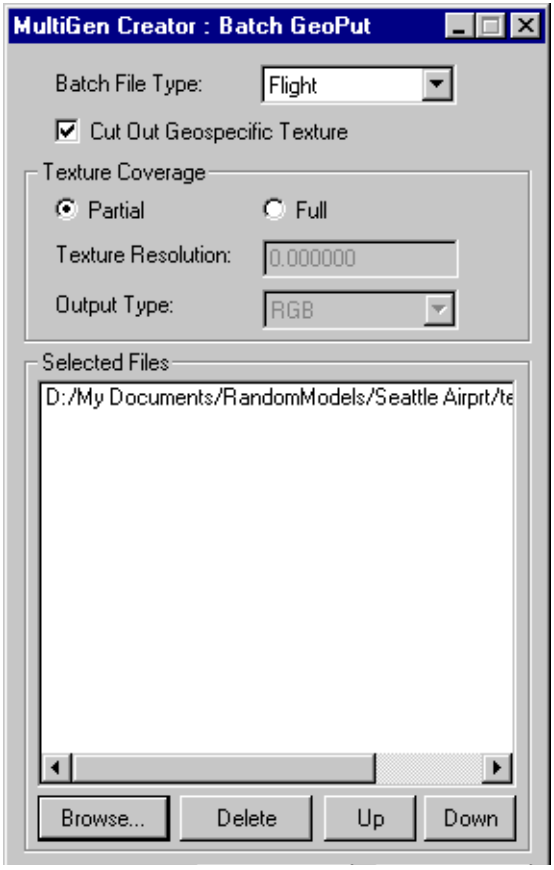
Batch GeoPut is a utility in the **Terrain** menu that allows you to apply multiple geospecific textures to a specified set of OpenFlight files. The textures are applied to all levels of detail.

The *Batch GeoPut* window requires you to add files to two lists. The first is a list of geospecific textures, and the second is a list of the OpenFlight files to which you will apply the textures. Batch GeoPut opens each specified OpenFlight file and applies all textures in the list that have geospecific coordinates that fall within the coordinates of the OpenFlight file database. The textures do not have to be loaded into the Texture palette first.

With Batch GeoPut, you do not have to select faces in your database to apply geospecific textures – you do not even have to open the OpenFlight file.

To apply a texture using **Batch GeoPut**:

Step 1. Choose **Terrain/Batch GeoPut**. The *Batch GeoPut* window appears.



Step 2. Make sure **Flight** is selected in the **Batch File Type** list box.

Step 3. Choose an OpenFlight file that you want to apply geospecific textures to, and then click **Open**. To select multiple files, hold down the Ctrl key as you click individual file names. To select a range of files, hold down the Shift key as you click the first and last file names in the range.

Note: If you have created a series of OpenFlight files using the **Batch** Terrain option, you must select those files, not the master file containing external references to them. **Batch GeoPut** will not apply texture to external references.

Step 4. Continue to add files until your list of OpenFlight files is complete. The files in the **Selected Files** list do not have to be in the same directory.

Step 5. Change the **Batch File Type** list box selection to **Texture**, and then add the geospecific texture files you want to apply.

Note: The order of the files in this list is important if you choose textures that will overlap either fully or partially after they are applied. For any overlapping areas, the texture that is applied last will overlap all previously applied textures.

Step 6. Choose a **Texture Coverage** method:

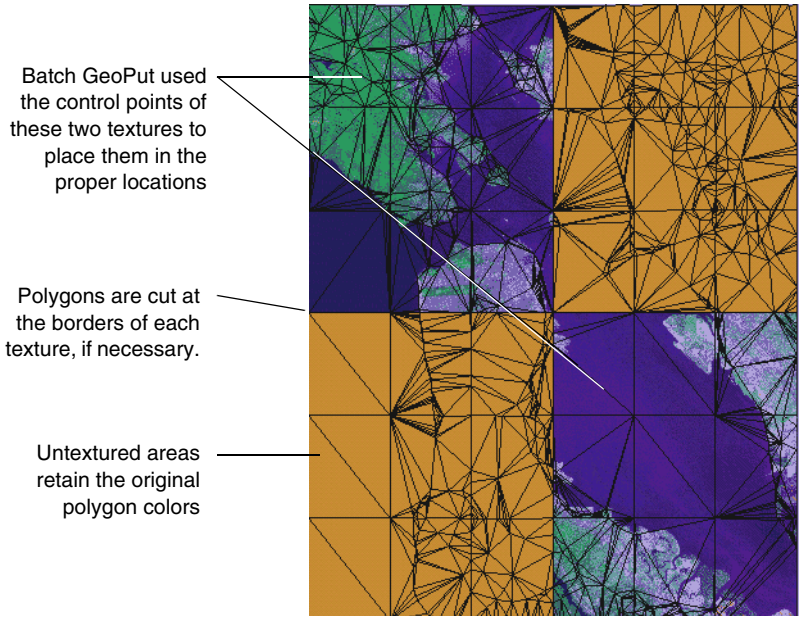
Partial results depend on whether or not **Cut Out Geospecific Texture** is selected.

IMPORTANT! Do not use **Cut Out Geospecific Texture** with CAT databases. CAT tesselation is carefully calculated to produce smooth morphing between LODs. When you use **Cut Out Geospecific Texture**, you change the terrain tesselation, resulting in CAT databases that do not morph correctly.

Partial combined with **Cut Out Geospecific Texture** applies each texture only over the area determined by its geospecific control points. With this meth-

od, the number of polygons is increased, unless the borders of each texture coincide with the edges of polygons.

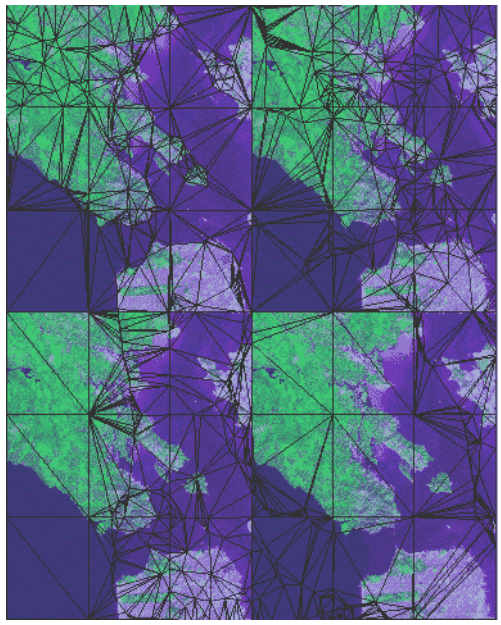
Partial with Cut Out Geospecific Texture



When **Cut Out Geospecific Texture** is not selected, **Partial** repeats a single texture to cover the entire terrain of the OpenFlight file. If you choose more than one texture, **Batch GeoPut** overwrites each texture, until the last tex-

ture in the list is applied. This texture is the only one you will see when you open the OpenFlight file, and it is repeated to cover the entire terrain.

Partial Without Cut Out Geospecific Texture

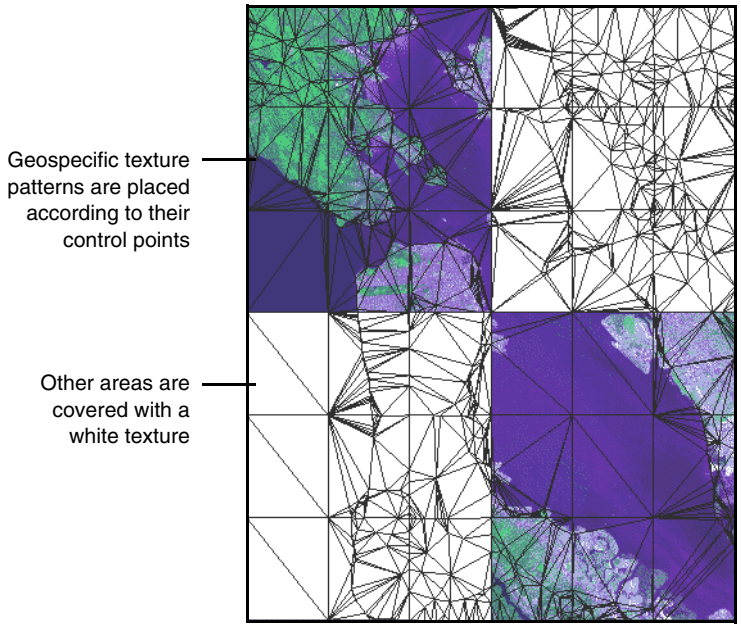


A single geospecific texture pattern is repeated until the entire OpenFlight file is covered with texture.

Full applies the geospecific textures in the order in which they were selected (see the previous step), and then applies a white texture pattern to any

areas of the database that are not covered by geospecific texture. **Cut Out Geospecific Texture** is ignored when **Full** is selected.

Full Coverage



Note: If the face color of a polygon in the OpenFlight file is not white, the face color and the overlying texture color will combine. For example, a red polygon that is covered with white texture will appear pink. To eliminate this effect, make sure the face attribute **Draw Textured Face White** is selected for each polygon in the file.

Step 7. Click **OK** to begin Batch GeoPut. As Batch GeoPut applies each texture, it places a new entry in the *Texture Mapping* palette of the affected file.

The GeoPut Texture Tool



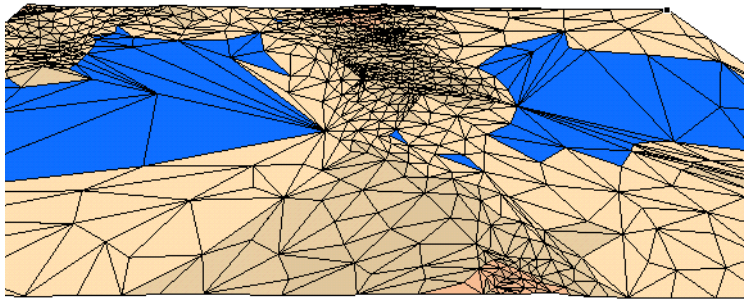
The **GeoPut Texture** tool in the **Map Texture** toolbox applies a geospecific texture pattern to an existing terrain database having the same geospecific coordinates. You can use

USGS or third party bitmap images that have geospecific coordinates associated with them, or you can use any texture pattern and apply geospecific coordinates to it by modifying its attributes. The texture can be applied to a single level of detail, or to all levels of detail, depending on what has been selected.

To apply a geospecific texture using the **GeoPut Texture** tool:

Step 1. Load the geospecific texture pattern into the Texture palette. If it has already been loaded, click it to make it the current texture. See "Setting Geospecific Information" on page 1192 for information about setting geospecific coordinates.

Step 2. Open the database.



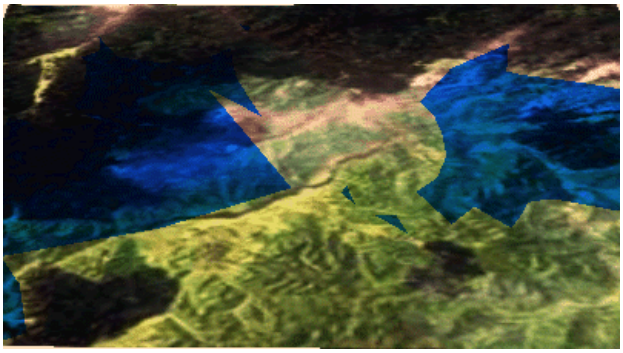
Step 3. In **Group** mode, select the database to select the currently displayed level of detail, or choose **Select/Select All** (Ctrl + A) to select all the levels of detail in the database.



Step 4. Open the **Map Texture** toolbox and click **GeoPut Texture**.



Step 5. In the View panel, click **Show/Hide Texture** to view the texture.



Creator Terrain Studio

Creator Terrain Studio (CTS) Image tools divide large textures into smaller tiles that are more readily managed in memory when the textures are loaded into your realtime system. CTS combines a set of geospecific orthographic textures into a single large image that is composed of small image tiles for display on realtime systems. In previous versions of Creator TerrainPro, the mgmosaic utility created clip textures and high-resolution insets.

Because only part of the texture is in view at a given time, only those texels in view need to be loaded into memory. Using CTS, a large image can be tiled or clip textured so that only the visible image tiles are loaded into texture memory. See the CTS online help for more information about using CTS.

34 Conversion Utilities and Filters

Elevation data comes in many different formats, such as USGS DEM and NIMA DTED. These original source data formats represent elevation maps for specific geographic regions, but with different format specifications. The Digital Elevation Data (DED) format is a uniform elevation map format that MultiGen-Paradigm developed for the Creator terrain generation tools to read. Because the terrain generation tools do not read source data in their original format, the original source data must first be converted to the DED format.

Creator includes the following utilities for converting terrain and image data in other formats to DED:

- *readdma* converts NIMA Digital Terrain Elevation Data (DMA DTED) to DED
- *readusgs* converts United States Geological Survey Digital Elevation Model (USGS DEM) data (also known as "raw" data) to DED
- *float2ded* converts data in binary float format to a 1 degree by 1 degree cell of DED
- *catdma* combines multiple DED files into a single file
- *mgattredit* lets you edit header attributes for large textures
- *mgutm2geo* converts UTM image attributes to geodetic attributes
- *Convert Database to DED* utility (located in the **Terrain** menu) converts an OpenFlight database to a DED file

The terrain and image conversion utilities can be found in the Creator installation directory.

Creator also includes the DED Builder tool, which uses the same functionalities as the readusgs, readdma, and catdma tools, to convert terrain elevation data to DED. You can use this tool to combine elevation files of different file types into a single DED file. You can edit the geographical information in an elevation file, either in the DED Builder tool or in the DED Header viewer, before you convert the file.

Import filters convert data from other formats to OpenFlight. Creator includes:

- *3D Studio* file import (*.3ds files)
- *AutoCAD* file import (*.dxf files)
- *Wavefront Object* file import (*.obj files)
- *Stereo Lithography* file import (*.stl files)

Export filters convert data from OpenFlight to other formats. Creator includes:

- *AutoCAD* file export (*.dxf files)
- *VRML* file export (*.wrl files)
- *Stereo Lithography* file import (*.stl files)

These import and export filters are installed with the default Creator installation.

Related Topics

Using the Conversion Utilities On Windows Systems

Step 1. From the Windows **Start** menu, open an *MS-DOS command prompt* window. (For most installations you choose **Programs** from the **Start** menu, and then drag to select **Command Prompt** from the submenu.) The window opens with a prompt that shows the current drive. For example:

C:>

Step 2. Most of the conversion utilities must be run from the directory that contains the files you want to convert. If the files are on a different drive from the one shown in the *command prompt* window, type the drive letter and a colon, and then press Enter. For example, to change from drive C to drive D, type the following at the C:> prompt:

d: [Enter]

Note: MS-DOS commands are not case-sensitive. You could enter d: or D: to change to drive D.

Step 3. Change to the directory that contains the files you want to convert. To do this, type cd, plus the path to the files, and then press Enter. For example:

cd \terrain\usgsfiles [Enter]

Step 4. Utilities are installed at the top level of the directory that contains the Creator program files. To start a utility, type: <drive letter>:\<`Creator` directory name>\ <utility name> and then press Enter. For example, if you installed Creator in the Creator directory, type the following to start the Readusgs utility:

\MultiGen\Creator\readusgs [Enter]

Step 5. Answer the prompts given by the utility. These are explained in the following sections. Parameters enclosed in <> are optional.

To discontinue processing by a conversion utility at any time, press Ctrl+C.

Using the Conversion Utilities on IRIX Systems

Step 1. Utilities are installed at the top level of the directory that contains the Creator program files. To start a utility, type: /<path to `Creator` directory>/<utility name> and then press Enter. For example, to start the readdma utility type:

Note: /usr/local/multigen/creator/readdma [Enter]

Step 2. Answer the prompts given by the utility. These are explained in the following sections. Parameters enclosed in <> are optional.

To discontinue processing by a conversion utility at any time, press Ctrl+C.

The readdma Utility

The National Imagery and Mapping Association (NIMA), a United States agency formerly known as the Defense Mapping Agency, produces CD-ROMs containing elevation data in DMA DTED format. Elevation data on the CD-ROM is stored in *cells*, or files that provide data for one-degree latitude by one-degree longitude areas of the Earth. Each cell includes an ASCII header that contains information describing the terrain data, including the geographic coordinates of the cell origin, elevation spacing, data accuracy specifications, and the horizontal and vertical limits of the data. The readdma utility converts data in this format to Creator's DED format.

To Convert DTED Data to DED Format

Step 1. From a command prompt, change to directory where the readdma utility is located, type readdma, and then press Enter. (See "Using the Conversion Utilities On Windows Systems" on page 1206 or "Using the Conversion Utilities on IRIX Systems" on page 1207 for detailed instructions.)

readdma responds:

DMA DTED Raw Data Preprocessor Utility - Version <version number>.

From 0-Tape or 1_Disk?

Step 2. Type the appropriate number for the unit and press Enter.

On Windows systems, readdma can only read files from disk or CD-ROM, so you would type 1.

The following prompt appears:

Input filename:

Enter the name of the file you want to convert and press Enter. For example: n27.dt1 [Enter].

readdma responds:

1-Level1 or 2-Level2?

Step 3. Enter the level number that corresponds to the DMA level of your file and press Enter.

readdma responds:

WARNING: VOL, HDR records not found!

Output file name:

Step 4. Ignore the warning – it appears because you are not reading the data from tape – and enter a name, including the path, for the DED file you want to create.

readdma responds:

Default setting:

VOL=0 HDR=0 UHL=80 DSI=648 ACC2700 DATA 2414 EOF 4 UTL=5 Tape_Mark=0

0-Use default or 1-Specify manually?

Step 5. Type 0 and press Enter, unless your file does not follow standard DMA DTED format.

readdma responds:

Number of cells on tape/file:

Step 6. Type 1 and press Enter. (Only files on tape have more than one cell per file.)

readdma issues file data messages during processing. When it is finished, you see the message Done! The new DED file can be imported into Creator.

The readusgs Utility

The United States Geological Survey (USGS) distributes digital elevation model (DEM) data for various locations across the United States and some surrounding areas of Mexico and Canada. USGS DEM files are freely available on the internet. The readusgs utility converts these files to DED format, so that they can be imported into Creator.

To Convert USGS Data to DED Format

Step 1. From a command prompt, enter the path to the readusgs utility, type readdma, and then press Enter. (See "Using the Conversion Utilities On Windows Systems" on page 1206 or "Using the Conversion Utilities on IRIX Systems" on page 1207 for detailed instructions.)

readusgs responds:

USGS DEM Raw Data Preprocessor Utility - Version <version number>.

Enter Input DEM File Name?

Step 2. Type the name of the file you want to convert (include the path, if necessary), and then press Enter. For example:

gcanyon.raw [Enter]

readusgs reads the file, issuing status messages in the process, and then prompts:

Enter Output .ded File Name?

Step 3. Type a name for the output file (gcanyon.ded, for example) and then press Enter. Include a path, if you want to create the output file in a different directory than the one that holds the source file (\converted_files\gcanyon.ded, for example).

readusgs prompts:

Does <filename> specify a file name or directory name on the target (F=file, D=directory)?

Step 4. Type f to specify a file name and press Enter. readusgs creates the DED file, which can be imported into Creator.

Note: If you enter d, readusgs creates a directory with the name you specified, and then creates a file named ssys.ded.tmp in that directory. You must rename the temporary file, because subsequent conversions will overwrite the file.

The float2ded Utility

The float2ded utility converts an array of data in *float* format into a 1 degree by 1 degree area of elevation data in DED format. The DED file that is produced can be imported into Creator, where the data will be used to build terrain.

The float2ded input file must be a 4 byte binary array of float values. (You can create this file using the standard C fopen function.) To run float2ded, you will need to enter the following information: the name of the input file, the number of values you want in the width and the number of values you want in the length of the array that will become a grid of elevation points. Starting at the southwest corner, float2ded creates a grid of elevation points from your float data. For example, if you specify a width of 300 and a height of 300, float2ded fills up the first column in the array with the first 300 values in the float file, fills the second column from the bottom up with the next 300 values, and so forth, until the number of columns matches the width you specified.

The following illustration shows how float2ded fills in the elevation points when it is given a width of 10 and a height of 10. (Typically, your file will have thousands of values; this is just for illustration purposes.)

float_10	float_20	float_30	float_40	float_50	float_60	float_70	float_80	float_90	float_100
float_9	float_19	float_29	float_39	float_49	float_59	float_69	float_79	float_89	float_99
float_8	float_18	float_28	float_38	float_48	float_58	float_68	float_78	float_88	float_98
float_7	float_17	float_27	float_37	float_47	float_57	float_67	float_77	float_87	float_97
float_6	float_16	float_26	float_36	float_46	float_56	float_66	float_76	float_86	float_96
float_5	float_15	float_25	float_35	float_45	float_55	float_65	float_75	float_85	float_95
float_4	float_14	float_24	float_34	float_44	float_54	float_64	float_74	float_84	float_94
float_3	float_13	float_23	float_33	float_43	float_53	float_63	float_73	float_83	float_93
float_2	float_12	float_22	float_32	float_42	float_52	float_62	float_72	float_82	float_92
float_1	float_11	float_21	float_31	float_41	float_51	float_61	float_71	float_81	float_91

- If your input file does not have enough values to satisfy the width and height you enter, float2ded inserts zeros in the vacant positions.
- If your input file has too many values, float2ded fills the positions up to the specified width and height, and ignores the additional data when creating the .ded file.

No matter how many values you enter for your width and height, the resulting file represents elevations for a 1 degree by 1 degree area of the earth.

To Convert Float Data into DED Format

Step 1. From a command prompt, type the path to the float2ded utility, the input file name, a space, the array width, a space, and the array height, and then press Enter. (See "Using the Conversion Utilities On Windows Systems" on page 1206 or "Using the Conversion Utilities on IRIX Systems" on page 1207 for detailed instructions.)

For example, for Windows type:

```
C:\multigen\creator\float2ded sfbayflt5.dat 3075 2890
```

For IRIX type:

```
/multigen/creator/float2ded sfbayflt5.dat 3075 2890
```

float2ded displays messages about the DED file it is creating, and then saves the new file in the same directory, and with the same root name as the source file. For example; sfbayflt5.dat is processed into the DED file sfbayflt5.ded.

The catdma Utility

The catdma utility concatenates multiple .ded files into a single .ded file, allowing you to create large terrain databases. Areas between the regions described by the input .ded files are given sea-level elevation (zero).

To Use catdma

Step 1. From a command prompt, change to the directory where catdma is located.

Step 2. Type catdma, and then press Enter. (See "Using the Conversion Utilities On Windows Systems" on page 1206 or "Using the Conversion Utilities on IRIX Systems" on page 1207 for detailed instructions.)

catdma responds:

DMA Cell Data Concatenation Utility - Version <version number>.

Output File Name:

Type a name for the output file (N_Calif.ded, for example) and press Enter. Include a path, if you want to create the output file in a different directory than the one that holds the source file (\converted_files\N_Calif.ded, for example).

catdma prompts:

Input File Name?

Step 3. Type the name of the first DED file you want to include in the concatenated file, and then press Enter. For example:

monterey.ded

catdma reads and processes the file, and then prompts:

More? (y/n)

Step 4. Type y and press Enter.

catdma prompts:

Input File Name?

Step 5. Type the name of the next DED file you want to include, and then press Enter.

Step 6. Continue to answer y to the prompt and enter file names. When you have entered all the names of the files you want to concatenate, type n and press Enter in response to the More? (y/n) prompt. catdma concatenates the files and creates the file with the name and directory path you specified.

The mgattredit Utility

Mgattredit lets you modify a texture attribute file without loading the file into the Creator Texture palette.

To use mgattredit, see the *MultiGen Creator Release Notes*.

The mgutm2geo Utility

The utility utm2geo converts an input UTM ortho image into a geodetic ortho image. The input ortho image must have an attribute file associated with it, which contains at least three specified corner points, the earth model and the UTM zone in which it is located.

To Use mgcvt_utm2geo

Step 1. From a command prompt, enter the following:

path\utm2geo [-h-r] [-o output_file] [-earth earth_model_type] image_file

Where:

- **path** is the path and directory where mgcvt_utm2geo is located.
- **-h** prints an explanation of the parameters to the screen.

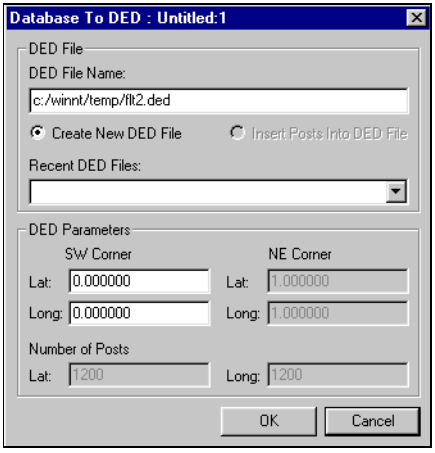
- **-r** converts a flat earth projection to UTM.
- **-o output_file>** specifies the name of the output file.
- **-earth earth_model** specifies the earth model type. The options are: **clrk66** for Clark 1966; **WGS84** for WGS 84; **WGS72** for WGS 72; and **bessel** for Bessel
- **image_file** is the UTM image.

Convert Database to DED

Convert Database to DED, in the **Terrain** menu, creates a gridded terrain file that represents the highest resolution level of detail of an OpenFlight database. This is a useful utility for creating elevation data when the original file has been damaged or is not available. The new file covers an area of one cell (one degree by one degree).

Converting Databases to DED Files

Select **Convert Database to DED** in the **Terrain** menu. The *Database to DED* dialog box opens, where you can set your conversion options and specify a file name.



Convert Database to DED provides two conversion options:

- **Create New DED File** creates a single DED file for each OpenFlight database. When this option is set, reusing the same DED file overwrites existing information.
- **Insert Posts Into Existing DED File** inserts additional post data into an existing DED file. For example, if you have several OpenFlight databases of a large area of terrain (as with area block batch processing), you can convert one of the databases to a DED file and add the rest of them to create one large terrain file.

Any feature data, textures, color or material properties applied to the OpenFlight database will be ignored by **Convert Database to DED**. Only the terrain polygons will be converted.

Creating a DED File

Step 1. Open the OpenFlight database you want to convert.

Step 2. Choose **Terrain/Convert Database to DED**. The *Database To DED* dialog box opens.

Step 3. Enter a path and file name for the new DED file in the **DED File Name** field.

The OpenFlight database's southwest and northeast coordinates display in the **DED Parameters** section. You can change the southwest latitude and/or longitude coordinates as long as they fall within the database's geospecific area.

When the OpenFlight database has been converted, the file name will be added to the **Recent DED Files** list.

Step 4. Set the **Create New DED File** option.

Step 5. Enter new **Lat** and **Long** coordinates for the southwest corner, if necessary.

Note: Minutes must be calculated as a decimal (37~ 40′ becomes 37.66)

Step 6. Click **OK** to create the DED file.

Overwriting Existing DED Files

Step 1. Open the OpenFlight database you want to convert.

Step 2. Choose **Terrain/Convert Database to DED**. The *Database To DED* dialog box opens.

Step 3. In **Recent DED Files**, select the existing DED file you want to overwrite. Once selected, this file name will also appear in the **DED File Name** field.

Step 4. Set the **Create New DED File** option.

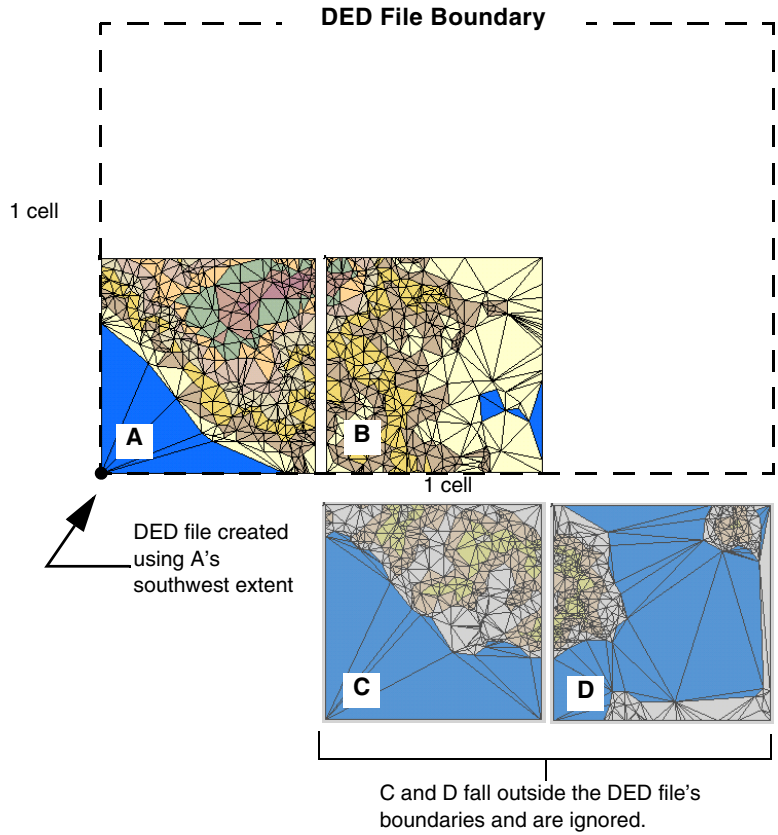
Step 5. Click **OK** to convert the data. A message box asks you to confirm overwriting the existing DED file.

Step 6. Click **OK** to overwrite the file.

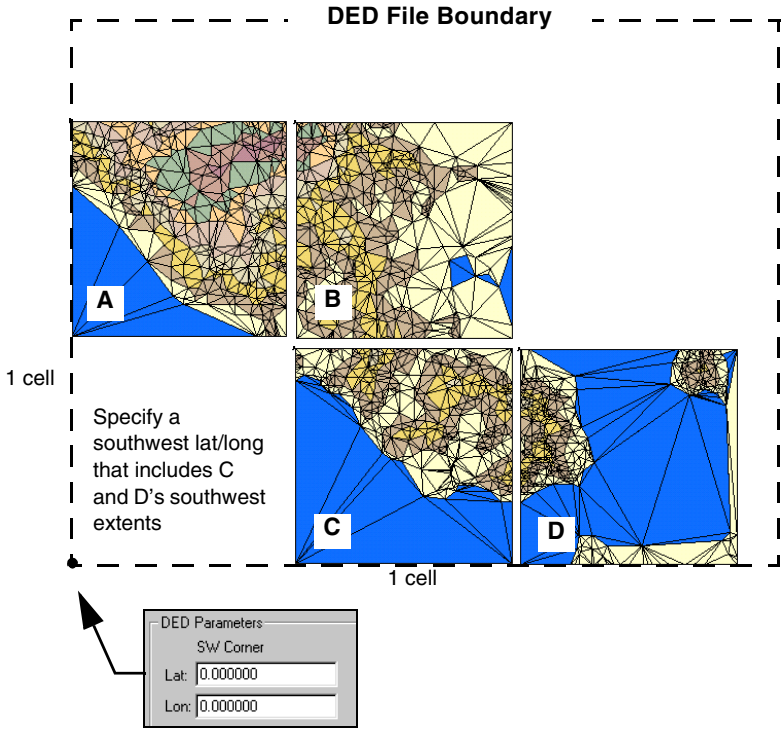
Inserting Data Into an Existing DED File

By setting the **Insert Posts Into Existing DED File** option, you can add post data to an existing DED file. Note that the southwest corner of the database being added must fall within the DED file's extents. If it does not, the data will not be inserted and an error message will display.

Consider the figure below. In order for a DED file to contain post data from files A, B, C, and D, you would open one of the files, A for example, create a DED file by setting the **Create New DED File** option, and then open and insert B, C and D into it by setting the **Insert Posts into Existing DED File**. Using A's southwest corner as the DED file's origin will cause C and D to fall outside the file's boundary. Their post data will not be inserted, and a message will inform you that the database lies outside the DED file's extents.



You must modify the **DED Parameters** section of the *Database to DED* dialog box and specify **Lat** and **Long** values for the new DED file that include the southwest boundaries of C and D, as shown in the figure below.



Creating a DED file from a highly detailed database and then inserting post data from a database covering the same area, but at a lower level of detail will give you a greater number of posts to sample. It does not, however, change the DED file's overall elevation.

To insert post data into an existing DED file:

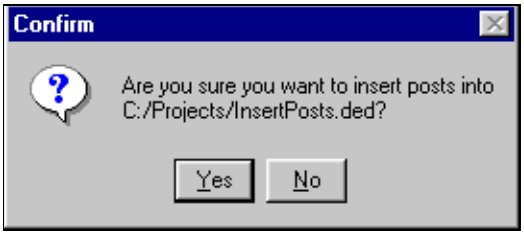
Step 1. Load the OpenFlight database to be added.

Step 2. Choose **Terrain/Convert Database to DED** to open the *Database To DED* dialog box.

Step 3. In **Recent DED Files**, select the existing DED file you will be adding data to. Once selected, this file name will also appear in the **DED File Name** field.

Step 4. Set the **Insert Posts Into Existing DED File** option. This option inserts the current database's post data into the DED file without overwriting existing data.

Step 5. Click **OK**. You will be asked to confirm that you want to add post data to the existing DED file.



Step 6. Click **OK** to add the data.

The DED Builder

The **DED Builder** tool in the **Terrain** menu converts terrain elevation data from their original file formats to MultiGen Creator's DED file format. The DED Builder provides a single graphical interface to import elevation files of different file types, preview the contour maps of elevation files, change their geographical information, and combine the files into a single DED file.

The DED Builder converts each non-DED elevation file to a temporary DED file (intermediate file) that is stored on your computer. When you generate the output DED file, these intermediate DED files are combined with any imported DED files to create a single DED file.

Note: All files that you import must have the same coordinate projection system (UTM or Geodetic) to load correctly.

You can import these types of elevation files:

- USGS DEM
- USGS GTOPO30
- NIMA Digital Terrain Elevation Data (DTED)
- USGS STDS DEM
- MultiGen-Paradigm Digital Elevation Data (DED)

Related Topics

"DED File Cells" on page 1220

"To Import and Convert Elevation Files" on page 1221

DED File Cells

A DED file contains one or more *cells* of elevation information. These cells are visually represented in the DED Builder after you import the elevation files. Each cell corresponds to a geographic region and is composed of an *elevation map* and a *cell header*. The elevation map contains elevation values (posts) at evenly-spaced intervals within the geographic extents of a cell. The cell header contains a description of the cell, including:

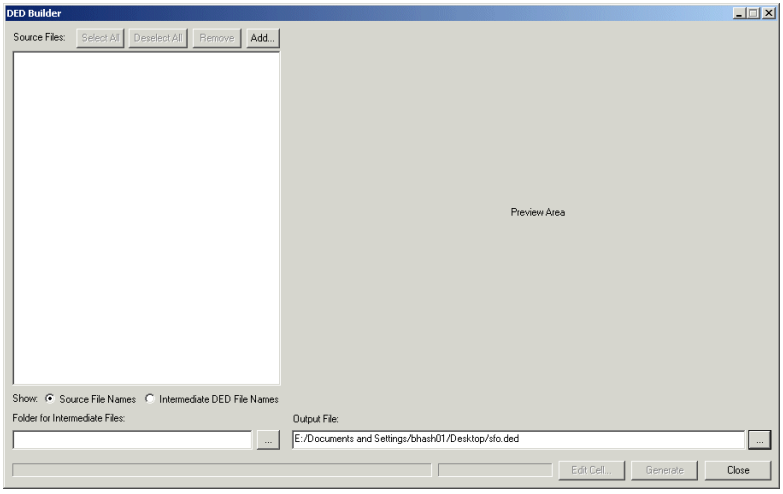
- The extents (latitude and longitude points) of the geographic region covered by the cell
- The number of elevation sample points in each dimension (longitude and latitude)
- The distance between sample points in each dimension

Within the DED Builder, you can view and edit the cell header information. You can translate the elevation map contained in a DED file to another geographic area by changing longitude and latitude points in the cell header. The DED Builder will convert files with changed header information. Header information only changes in the new DED file; the original source file does not change.

Typically, the cells in a DED file are related and are geographically near each other. Because Creator operates on a single DED file, combining neighboring cells into one big DED file lets you process larger areas in Creator.

To Import and Convert Elevation Files

Step 1. Choose **Terrain/Build DED**. The *DED Builder* dialog box appears.



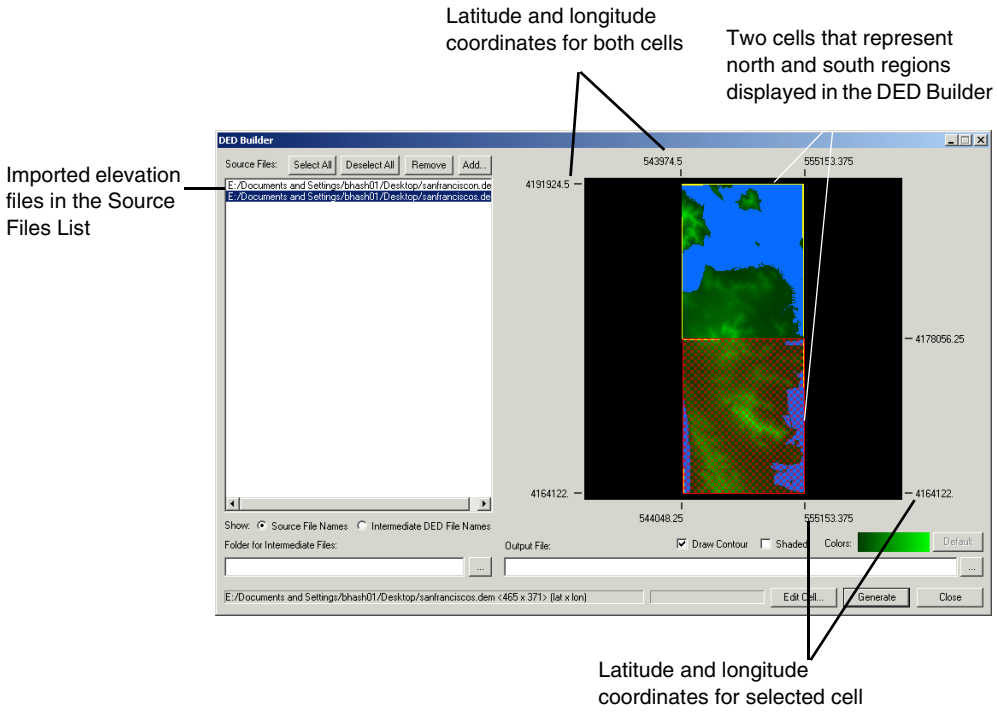
Step 2. If you are importing non-DED elevation files, click the browse button next to the **Folder for Intermediate Files** field to specify the name of the folder where you want the intermediate files saved. The intermediate DED files will be saved in the same folder where the original elevation files are located if you do not specify a folder. You must specify a folder if you are importing elevation files from a CD-ROM or other read-only device.

Note: The DED Builder does not delete the intermediate files automatically. If you want to delete these files, delete them only when Creator is closed to be sure that Creator does not need to access them.

Step 3. Click the **Add** button. In the *Open* dialog box, select the type of elevation files that you want to import in the **Files of Type** field, and select the files.

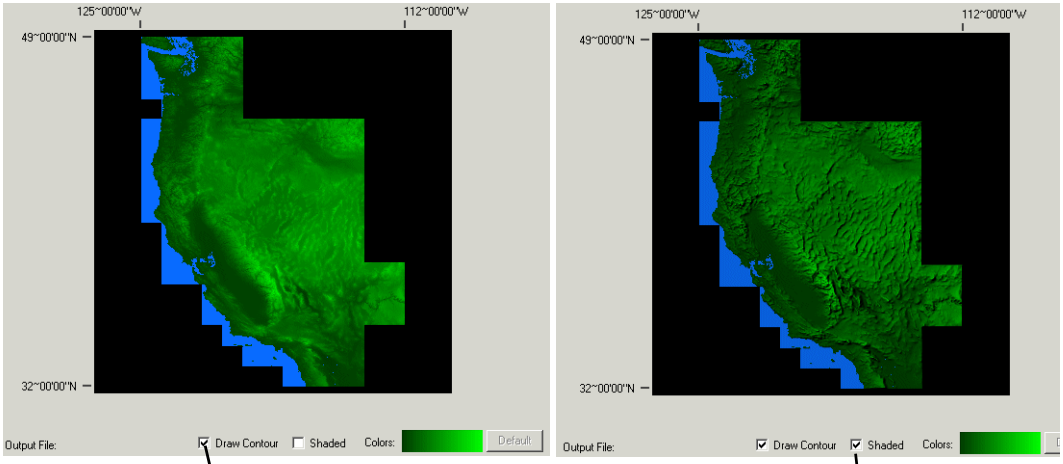
Step 4. Click the **Open** button. The files appear in the **Source Files** list in the *DED Builder* dialog box. In the contour display, each cell in an elevation file is represented by a solid checkerboard pattern over the geographic area where the cell is located. The extents for a selected cell are displayed at the right and bottom sides of the contour display; the extents for all cells are displayed at the top and left sides.

Note: Elevation data displays in a real-world orientation, so if cells are geographically far away from each other, they will appear very small to fit in the contour display. Areas with no elevation data appear blank in the contour display.



Step 5. Select the **Intermediate DED File Names** option to view intermediate files in the **Source Files** list.

Step 6. Set the **Draw Contour** checkbox to view the contour map of each cell. Selected cells are overlaid with a red checkerboard pattern. You can set the **Shaded** checkbox to display the contour maps as shaded reliefs.

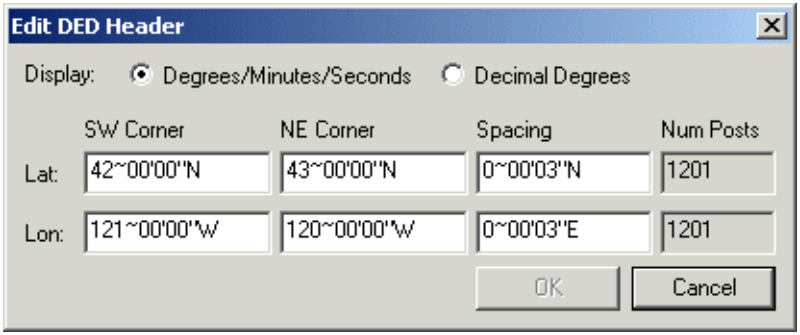


Step 7. To change the colors used to draw the contours, click the left side of the Colors swatch to change the lowest elevation color and the right side of the Colors swatch to change the highest elevation color. The Windows *Color* dialog box appears in which you can pick a color. All elevations in between are calculated by interpolating between the low and high colors.

Step 8. Select one or more elevation files in the **Source Files** list to process using one of the following methods:

- Click a file in the **Source Files** list with the left mouse button
- Click the **Select All** button to select all source files
- Click or fence an area of the contour map with the left mouse button. Hold down Shift to either add or deselect a selected file. Hold down Ctrl to deselect all files.

Step 9. To change the header information for a cell, select a cell, either by selecting a file in the **Source Files** list or clicking a cell in the contour map, and click **Edit Cell**. The *Edit DED Header* dialog box appears.



Step 10. Select the **Display** option to display latitude and longitude coordinates in either units of degrees/minutes/seconds or decimal degrees. You can change the latitude and longitude coordinates for the southwest or northeast corners of the cell. You can also change the spacing between posts in each dimension in the **Spacing** fields.

Step 11. Click **OK** after you have made your edits.

Step 12. In the *DED Builder* dialog box, click the browse button next to the **Output File** field to specify the name of the DED file that you want to create and the location where you want it saved.

Step 13. Click **Generate** to generate a single DED file from the selected elevation files. The status message at the bottom of the DED Builder dialog box indicates if the DED file was created successfully.

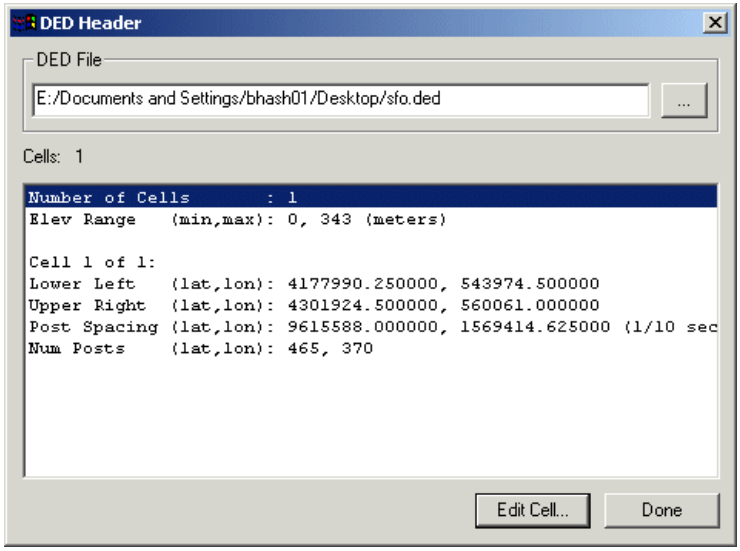
Note: If there are several warning or error messages during processing, only the last message will appear in the status field at the bottom of the *DED Builder* dialog box. Choose **Status Log** in the Creator **Info** file menu to view a complete history of all warning and error messages.

Viewing DED Headers

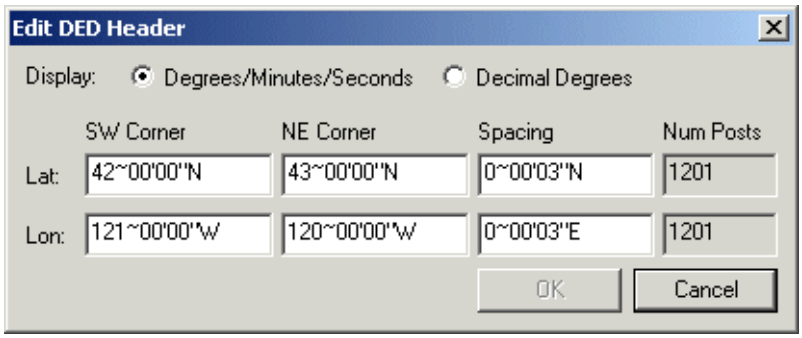
The **DED Header** viewer in the **Terrain** menu lets you preview the DED file's header information, which includes the geographic extents that a DED file covers. You can edit this information to relocate the elevation map to another geographic area on the Earth. Any edits are saved in the DED file.

To View a DED Header

Step 1. Choose **View DED Header** in the **Terrain** menu to open the *DED Header* dialog box. Information such as the number of cells in the DED file, the latitude and longitude coordinates of the lower left and upper right corners of the cell, the spacing between posts, and the number of posts in each direction, is listed.



Step 2. Click **Edit Cell** to change this information in the *Edit DED Header* dialog box.



Step 3. Click **OK** when you have made your edits. The information is updated in the *DED Header* dialog box and saved in the DED file.

Step 4. Click **Done** to close the *DED Header* dialog box.

Import Filters

Creator includes plug-in modules that allow you to import 3D Studio .3ds files, Stereolithography .stl files, Wavefront Object .obj files, and AutoCAD .dxf files into Creator. To import a file, choose **File/Import** to open the *Import Database File* window. Select the file type from the **Files of type** list and then choose the file to open.

> **Note:** To import files for creating a terrain database, you use the **Terrain/Import Elevations** command (see "Reading and Importing Elevation Data" on page 1063).

Import/export filters must be located in the Plug-Ins folder within the directory in which you installed Creator. The default installation procedure places them in the correct location.

3D Studio Import Filter

The 3D Studio plug-in imports files from 3D Studio version 3.0 or later and converts them into Creator databases.

Compatibility Notes:

- Geometry

  Geometry is fully supported (3ds files are all triangles)

- Materials

  The following 3D Studio material information is used to construct the Creator material palette:

  Ambient color

  Specular color

  Diffuse color

  Shininess strength

  Self-illumination percentage

  Transparency

- Face color

  3D Studio objects get their color from the combination of material and texture colors, so the face colors are set to white on import, and the object must be shaded to see the true colors in Creator.

- Texture

  The MultiGen texture palette is loaded with the primary texture patterns used in the 3D Studio materials. Any image format supported by Creator can be used.

  Texture patterns are applied using 3D Studio texture vertices. Texture environment attributes are approximated according to 3D Studio settings.

- Light source nodes

  Creator uses the following 3D Studio information to construct light source nodes:

  Ambient light source color/intensity

  Omnilight position, color/intensity

  Spotlight position, color/intensity, direction, falloff.

- Unsupported features

  Complex texture effects

  Animation of any kind

Cameras

AutoCAD Import Filter

The AutoCAD plug-in imports files from AutoCAD .dxf format and converts them into Creator databases.

Compatibility Notes:

- Each “layer” of the AutoCAD model is given its own group node.
- Each use of a “block” in the AutoCAD model becomes an instance node that is inserted as a child of the group (layer) in which the block is used.
- AutoCAD models do not include vertex normals, nor do they distinguish the front from the back side of the face. Therefore, some faces may appear backfaced. To correct this, you can reverse individual faces manually, or set the face attribute **Both Sides Visible**.
- The AutoCAD Text entity is ignored.

Wavefront Object Import Filter

The Wavefront Object file plug-in imports Wavefront Object files in .obj format and converts them into Creator databases. The OBJ file format stores geometry and other properties of 3D objects.

Stereolithography Import Filter

The Stereolithography plug-in imports Stereolithography files in .stl format and converts them into Creator databases. The STL file format stores information about 3D models on CAD/CAM systems. It is often used for producing 3D physical prototypes from digital data.

Export Filters

Creator includes plug-in modules that allow you to export AutoCAD .dxf files, Stereolithography .stl files, and VRML .wrl files from Creator. To export a file, choose

File/Export to open the *Export Database File* window. Choose the file format in which to save the file from the **Save as type** list. Then name the file and click **Save**.

Import/export filters must be located in the Plug-Ins folder within the directory in which you installed Creator. The standard installation Creator places them in the correct location.

AutoCAD Export Filter

The AutoCAD plug-in exports Creator database files and converts them into AutoCAD .dxf format.

Compatibility Notes:

- The only .dxf entity used in the output file is 3DFACE.
- The database hierarchy is lost due to the limited hierarchy of the .dxf format.
- Polygons with more than three sides are split into triangles.

VRML Export Filter

The VRML export filter converts Creator database files to VRML 1.0 format.

Stereolithography Export Filter

The Stereolithography export filter converts Creator database files to STL format. The STL file format stores information about 3D models on CAD/CAM systems. It is often used for producing 3D physical prototypes from digital data.

35 Road Tools

The Creator Road tools option creates realistic roadway models for driving simulation, training, and accident reenactments, as well as completely imaginary roads and paths for virtual worlds. Developed in conjunction with the University of Iowa College of Engineering, Road tools readily duplicate complex 3D surfaces used in real-world road construction, including superelevated curves with spiral transitions and parabolic vertical profiles.

Using Road tools, curves, hills, and arbitrary combinations of both are easily defined and adjusted in interactive 3D views, and can be accurately modeled after engineering design standards.

In addition to creating quality visual database models, Road tools generate correlated analytic roadway data for use in traffic simulation and vehicle dynamics computations.

Road Tools Overview

Road tools can be broken down into three basic operations: Road construction, road tessellation, and scenario data.

Road construction defines the type of road or road section to be created, its start and end points, and its curve or slope. All new or existing road alignments and design parameters can be interactively manipulated or modified, with dynamic update of the resulting 3D geometry.

Road tessellation defines and applies the road's attributes, including its appearance at various levels of detail. Regularly spaced roadway features, such as light poles and reflective markers, can be placed along the roadway automatically. An unlimited number of levels-of-detail (LODs) can be generated, each with its own features, cross section, texture, and optimized curvature-based polygonization.

Scenario data is created for use in driving simulation. This data includes lane and centerline definition. However, only road construction and tessellation are necessary to build a realistic road, if it is not to be used for simulation.

Roads can be modeled using background map images and/or full 3D terrain skins as references.

Constructing Road Sections

Using the Construction Tool, you can create three types of road sections: curves, hills, or straight roads.

Road sections are built using a combination of the control points you enter on the tracking plane, the parameters you enter in the *Road Construction* dialog box, and the values provided by the setup file. If you lay down control points for a new section and the section turns red, it means the specified control points cannot be combined with the parameters in the dialog box to produce a valid (mathematically feasible) section.

Before you create the first road section, set the tracking plane as follows:

Step 1. Turn on the **Grid** display and set the grid units to an appropriate value (100 meters is a good starting point).

Enlarge the grid and zoom out as needed to view a large area (about 40 squares in each direction, for example).

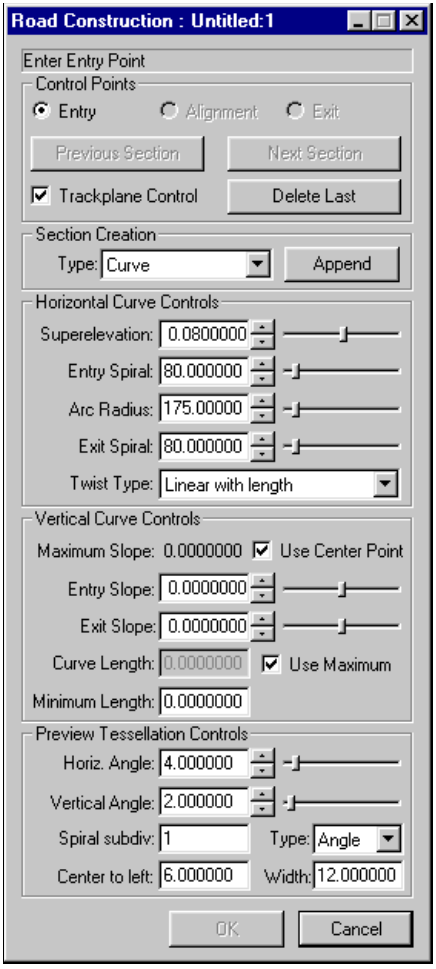
Step 2. If you are creating road sections that follow uniform or regular spacing, set the grid units to a value that allows all control points to fall on a point where grid lines intersect, and turn on **Grid Snap**. If you are creating road sections with non-uniform spacing, which is often the case in real-world roads, turn off **Grid Snap**.

Step 3. Set the tracking plane to an appropriate alignment for your first road section. To make it easier to enter control points, and to ensure continuity of the road's alignment, the Construction Tool automatically adjusts the tracking plane as you create additional sections.

Note: You can prevent automatic realignment of the tracking plane for each new section by clearing the **Trackplane Control** checkbox in the *Road Construction* dialog box. If you repeatedly require the same tracking plane orientation, you can save the orientation in the **Trackplane** menu, and reapply it. You may also want to save the eyepoint. These features are useful when you want to create symmetrical hills or curves with the same orientation. Refer to the *MultiGen Creator User's Guide* for instructions on saving tracking plane and eyepoint settings.

Using the Construction Tool

The **Construction Tool** is used to create road sections. You access the Construction Tool by choosing **Road/Construction Tool**. The *Road Construction* dialog box opens.



In the *Road Construction* dialog box, you define the type of road section to be created, and then set parameters to define the road's curve, superelevation and slope. You indicate the location of the control points for the road section in the Graphics view, and

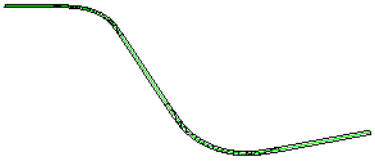
the road section appears. If the section is legal (mathematically feasible), it appears green. If it is not feasible it appears red.

You can select and drag any of the control points or change any of the road section's parameters and view the effect in the Graphics view. You can add initial tessellation to your road using the **Preview Tessellation** controls.

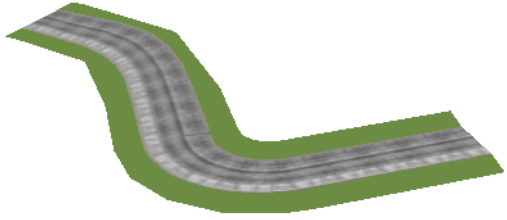
When a road section is satisfactory, click **OK** to build the road section, or click **Append** to build the next section. If you decide that you want to change the last section you were editing, or realize that you did not want another section at all, you can delete it by clicking **Delete Last**. Any section in the current road model can be edited by selecting it with the **Next Section** and **Previous Section** buttons.

> **Note:** While the **Construction Tool** is active, the road appears either green or red in the Graphics view. When you click **OK** to build the road, the Current color is applied.

Once the road is built, it appears in the Graphics view as a flat ribbon. Its initial width and centerline has been defined by the **Preview Tessellation** controls, but it has no other attributes such as texture or color.You must define its attributes and any features using the **Tessellation Tool**.



Before Tessellation Tool



After Tessellation Tool

Road sections are placed as road nodes at the group level in the Hierarchy view. Each road section is comprised of at least one path node and a number of road subsections, represented by group nodes, that are children to the road node.

Construction Tool Parameters

The following subsections describe each of the parameters in the *Road Construction* dialog box.

Control Points

The **Control Points** parameters define the limits of the road, specify the target road section for editing, and delete road sections.

Entry

Defines the section's beginning point in space (note that this is not necessarily where curvature begins). The *Road Construction* dialog box prompts you for the **Entry** point *only* when you are building the first section of a road. For subsequent sections, the Construction Tool assumes the **Entry** point is the same as the third point of the previous section. You can always modify a section's assumed **Entry** point. If you do, the previous section's **Exit** point changes to match, ensuring continuity.

Alignment

Defines the intersection of the tangents leading into and out of the curve in the horizontal dimension. The **Alignment** point can also be used to define the incoming and outgoing slopes of the section.

Exit

Defines the end point of the section.

Previous Section

Selects the road section built before the current section for editing. The previous section then becomes the current section.

Next Section

Selects the road section built after the current section for editing. The next section then becomes the current section.

Trackplane Control

Prevents the tracking plane from automatically realigning for each new road section. By default this option is set.

Delete Last

Deletes the last road section built.

Section Creation

Section Creation

Defines the type of road section that you are creating and allows you to add new sections to existing ones.

Type

Specifies what type of road section to create. You can create straight sections, curves or hills.

- **Curve** – Creates simple horizontal curves and horizontal and vertical curve combinations. Horizontal road curves are constructed using a radial arc with spiral (Euler) transition sections at either end to provide easement. A curve section's horizontal alignment and vertical profile are defined by inputting three control points and setting the curve's design parameters.
- **Hill** – Creates parabolic vertical curves with no horizontal curvature.
- **Straight** – Creates sections with no horizontal or vertical curvature. Note that straight sections are not often required; when you create curves and hills, the Construction Tool automatically constructs the straight segments necessary to provide a smooth transition between each adjacent section. You need to construct a straight section only when the section does not provide a transition between successive curves or hills. For example, you would use straight sections to build a network of straight streets.

Append

Adds the next section of road to the end of the section just created.

Horizontal Curve Controls

The **Horizontal Curve Controls**, together with the three control points, define the horizontal alignment of a curve.

Entry Spiral and Exit Spiral smooth the transition between a circular arc segment and a straight segment (or the adjacent road section, if the curve does not have a straight

segment). This includes providing easement between a superelevated (banked) curve and a flat road section.

Superelevation

Determines the maximum amount of bank in a curve, expressed as a slope (rise/run.)

Entry Spiral

The distance, in database units, through which the curvature and superelevation are introduced into a curve.

Arc Radius

Specifies the radius of the circle that defines the center line of the circular part of the curve. Every curve is defined by a circle. The circular arc segment conforms exactly to the circumference of this circle. The spiral segments in the curve form transitions between the circular arc segment and the previous and subsequent road sections.

Exit Spiral

The distance, in database units, through which the curvature and superelevation are reduced when exiting a curve.

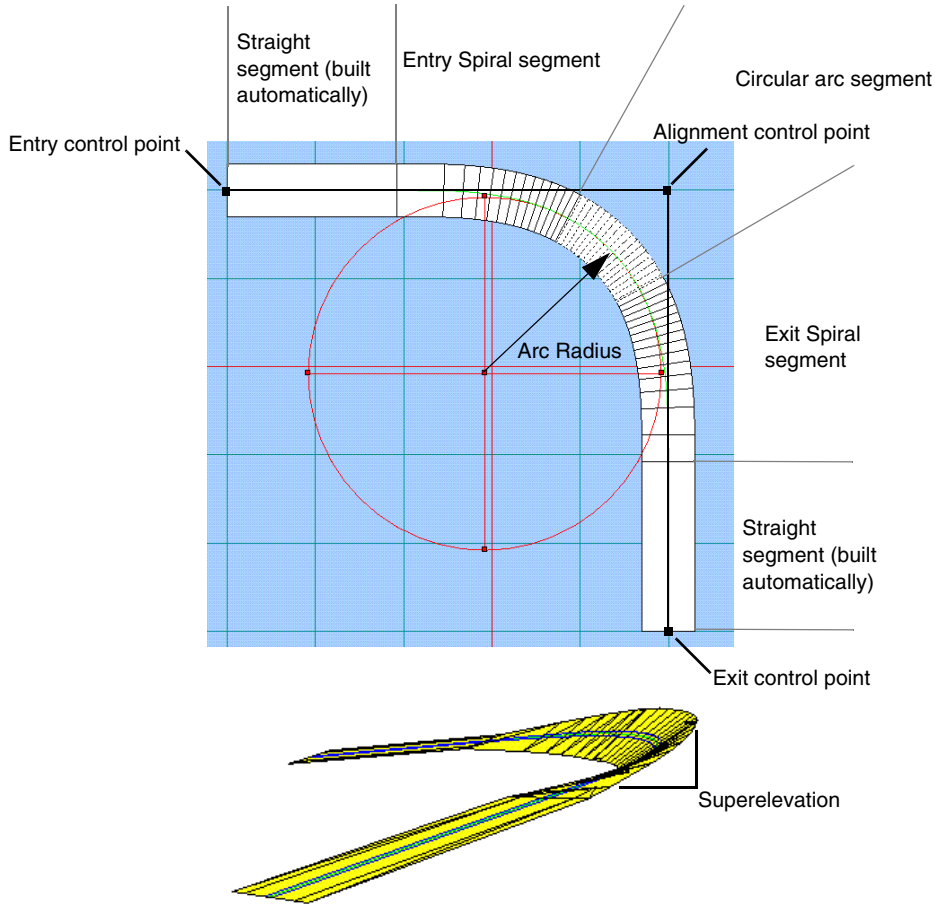
Twist Type

Controls the way in which superelevation (banking) is distributed in spiral sections.

- **Linear with Length** – Indicates that superelevation is added to the surface of the spiral at a linear rate with respect to the distance into the spiral. For example, at a point 80 meters into a 100-meter spiral, 80% of maximum superelevation has been attained. This is the method used in the design of many real roads. When your road has dramatic spirals that attain unusually high superelevation in short distances, this method can result in very abrupt transitions, producing a bumpy road surface. The other two methods may be better when your road has such radical banking.
- **Linear with Curve Angle** – Computes superelevation as a function that increases linearly with angular deflection into the spiral. As the rate of curvature increases more rapidly toward the arc-end of a spiral, so would the rate of twist or introduction of superelevation increase more rapidly toward that end. This concentrates most of the twist at the arc end of the spiral segment.

- **Cosine Function with Length** – Computes superelevation based on a cosine function with length along the spiral, giving very gradual twisting rates at both ends of the segment, and concentrating the twisting in the spiral segment's midsection.

  The following illustration shows how the **Horizontal Curve Controls** are used in the construction of a simple curve.



Vertical Curve Controls

Curves can have both horizontal and vertical curvature; hills contain only vertical curvature. **Vertical Curve Controls** for curve sections are the same as those for hill sections. Note that the vertical curvature is defined and computed independently of the horizontal alignment; vertical and horizontal curvature can be constructed separately or can occur together (either fully or partially) within a road section.

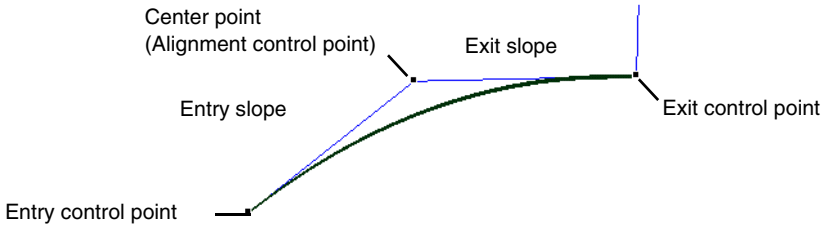
A curve's vertical profile is defined by the elevations and slopes at the beginning and ending control points, and by the **Vertical Curve Controls**.

Maximum Slope

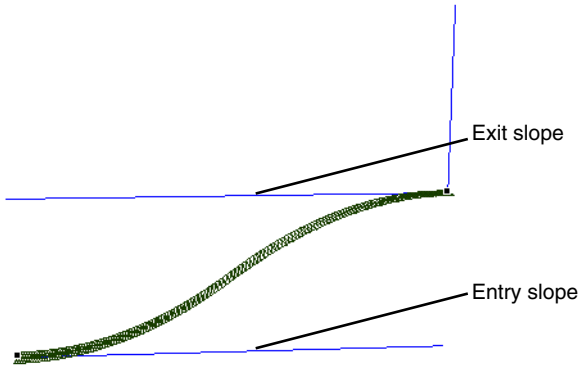
Displays the maximum slope of a hill or curve. This field is informational and cannot be modified.

Use Center Point

Defines the **Entry Slope** and **Exit Slope** from the lines connecting the first and third points to the second (center) point. In the following illustration, the curve is constructed with **Use Center Point** set.



To define the ending slope manually, clear **Use Center Point** and adjust the **Exit Slope** slider or type in a new value. A parabolic curve that is tangent to both the beginning slope and the ending slope is constructed. The parabola will be centered around the point of intersection of the two slopes, if they intersect between the beginning and ending control points. If the slopes do not intersect within the section, a set of two parabolas will be constructed to achieve the requested slopes, and **Using 2 Curves** is displayed in the dialog box (next to **Minimum Length**). For example, if you specify the same value for the starting and ending slopes, the slopes are parallel, and two curves are constructed.



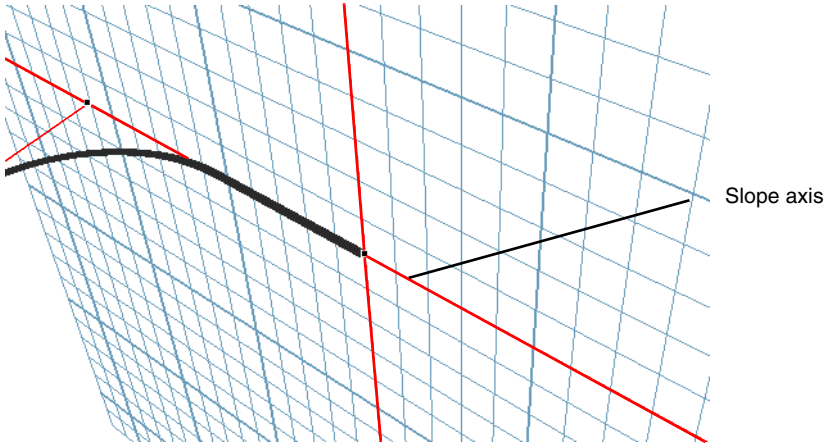
Entry Slope

Defines the slope at the first control point. The **Entry Slope** of the current section must equal the **Exit Slope** of the previous section. If you enter a different **Entry Slope**, the **Exit Slope** of the previous section automatically changes to match, ensuring that road sections are continuous.

Exit Slope

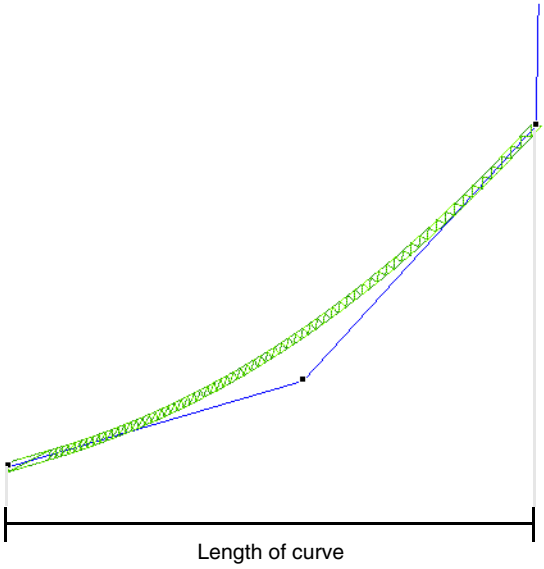
Defines the steepness of the slope at the third control point.

If you begin to create an additional section, the Construction Tool reorients the tracking plane, as shown in the following illustration, after you click **OK**. To prevent the **Exit Slope** from changing as you begin to define the next section, clear **Use Center Point**, and make sure you place the second control point on the slope axis by using the center mouse to click on the grid.

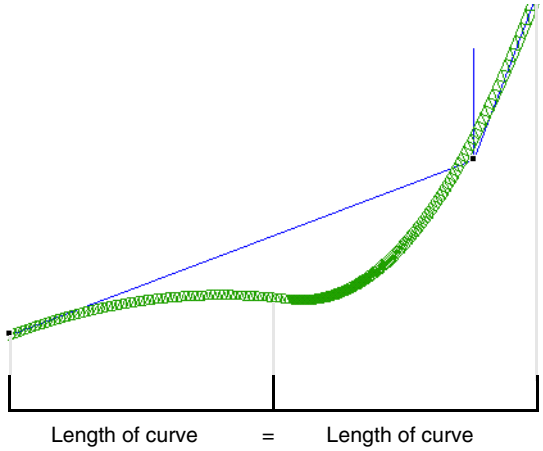


Curve Length

Specifies the horizontal distance covered by the vertical curve.



If the curve specifications force two curves, it is constructed from two parabolas of equal length. The **Curve Length** is the length of one of these parabolas.



In highway design, the length of the curve is dictated by design regulations and depends on the design speed of the road. If you are building a road that simulates real-life conditions, clear **Use Maximum** and enter a specific value for the **Curve Length**.

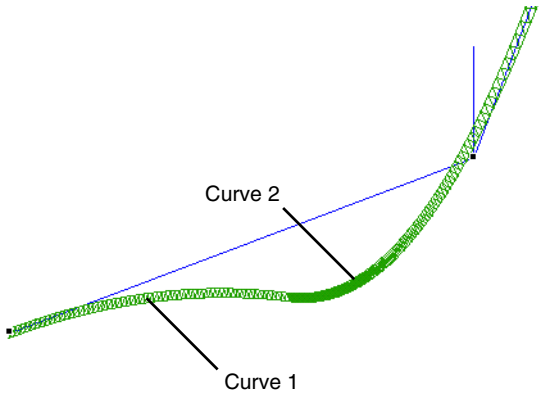
Use Maximum

Specifies that the vertical height should be based on the longest parabola that is tangent to the beginning and ending slopes and falls within the defined section.

Minimum Length

Applies when the intersection of a section's beginning and ending slopes occurs very close to either the first or third control point. This condition can severely restrict the length of the vertical curve, because the maximum length of a curve is twice the distance from the slope intersection to the nearest end point. If the vertical curve is restricted to a length that is less than the minimum length you specify, two parabolas of equal length are used to construct the curve section, thus allowing the use of longer parabolas.

The following illustration shows a vertical curve constructed from two curves when the **Minimum Length** was not achievable with one curve.



Preview Tessellation Controls

The **Preview Tessellation Controls** in the *Road Construction* dialog box allow you to define the road's initial width and centerline location so that you can view the road while it is being built. You can enter numbers in the various fields or use the sliders and view the road as it automatically updates the information. The information in these fields is applied to the entire road. However, you must use the **Tessellation Tool** to further define your road's attributes (texture, color, levels of detail, and features) or load a setup (.rds) file.

Horiz. Angle

Controls the number of horizontal polygons to create smoother transitions into and out of curves. As **Horiz. Angle** increases, the number of polygons decreases.

Vertical Angle

Controls the number of vertical polygons to create smoother or rougher curves and hills. As **Vertical Angle** increases, the number of polygons decreases.

Spiral Subdiv.

Allows a smoother transition through the spiral sections of curves, by dividing road surface polygons into smaller polygons. To create smaller polygons within the spiral segment, the **Horiz.Angle** and **Vertical Angle** values are divided by the subdivide value. For example, a subdivide value of 2 would double the number of polygons used to

represent the spiral section for that level of detail. While this results in a smoother driving surface, the extra polygons will impact system performance.

Center to Left

Defines the distance between the left edge of the road's cross section and the centerline of the road's construction.

Type

Specifies how the spiral segments are divided into polygons. This has no effect on the basic geometry of the spiral, but only on how polygons are used to surface that geometry.

- **Angle** – Creates polygons according to the **Horizontal Angle** curvature restraint in the *Road LOD attributes* dialog box. Using this method, more polygons are generated for areas of high curvature and fewer polygons are generated for areas of low curvature.
- **Distance** – Uses the same length along the centerline for each polygon in the spiral segment. This length is equal to the length of the polygons in the curved arc segment.

Width

Sets the distance from the left edge of the road to the right edge, on a vector that is perpendicular to the centerline of the road. This width is expressed in database units and does not include side areas.

Creating Curves

Curve creates simple horizontal curves, and horizontal and vertical curve combinations. Curves are defined by three points: The curve's beginning point, a control point, and the curve's end point; and by **Horizontal Curve Controls** and **Vertical Curve Controls** that define slope, amount of curve and transition lengths.

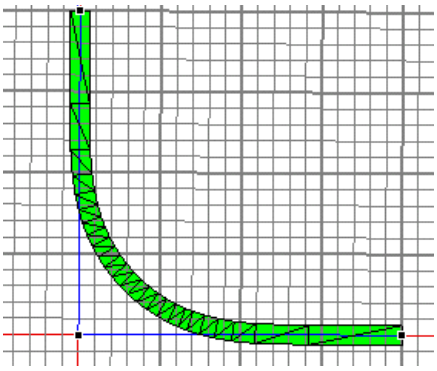
To Create a Simple Curve

Step 1. Choose **Road/Construction Tool**. The *Road Construction* window opens.

Step 2. Set **Type** to Curve.

Step 3. Specify the **Entry**, **Alignment**, and **Exit** points on the trackplane to define the curved road section. The road section displays in bright green.

Once you enter the third point, the curve displays in bright green. To reposition a point, hold down Ctrl and click the point to select it, then specify the new position or drag the point to the new position.



Step 4. Specify the **Horizontal Curve Controls** and **Vertical Curve Controls**. You can enter information into the fields, use the arrow buttons to increase or decrease the fields' values, or drag the sliders left or right. The road immediately updates with the new values. See "Horizontal Curve Controls" on page 1236 and "Vertical Curve Controls" on page 1239 for information about each of these controls.

Step 5. In the *Road Construction* dialog box, choose a section creation type and click **Append** to begin constructing the next section, or click **OK** to create the road.

Special Curves

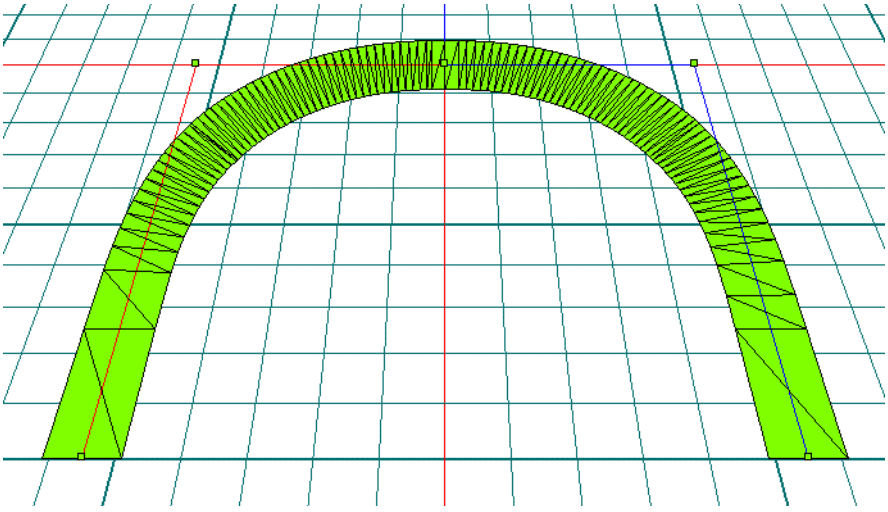
Special cases of curve sections include construction of curves of 180 degrees or more, compound radial curves, and any other construction requiring zero-length spiral transitions.

To build a curve of 180 degrees or more, you must define two curve sections, with the first section ending in an arc, and the second section beginning with an arc, as shown below.

Step 1. To create the first section, set its second spiral length to zero, and adjust its design parameters and control points such that the length of the ending straight piece is zero. You can either calculate the exact control point positions and design parameters (**Arc Radius** and **Entry Spiral**), or create the curve with a zero spiral length and eliminate the straight piece manually.

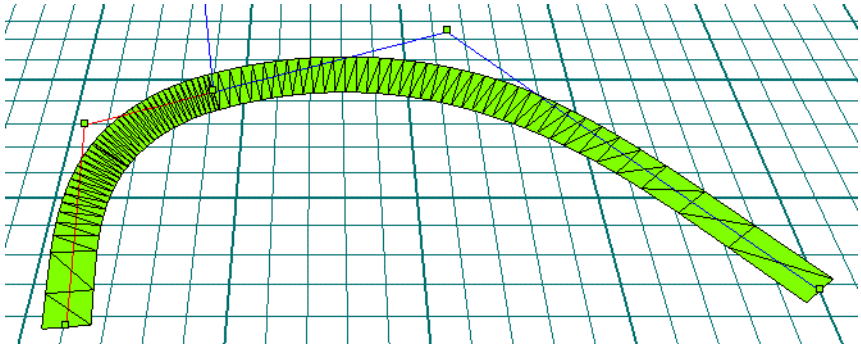
Step 2. To create the second section, set the **Entry Spiral** to zero and the **Exit Spiral** to its design length. Adjust design parameters and control points such that the length of the beginning straight piece is zero.

180° curve

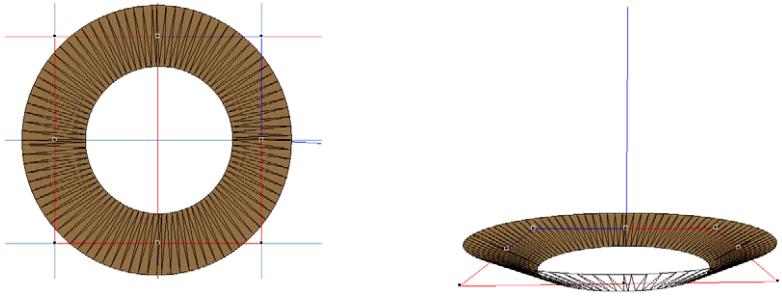


Compound Radial Curves

Compound radial curves are constructed in the same manner, but use tangent arcs of different radii, as shown below.



To construct a circular road, use four 90-degree connected sections of the same arc radius, all with zero-length beginning and ending spirals. To make it easier to place the control points, turn on **Grid Snap**. It is important to note that straight portions that continue out from an arc without a transition are built at the same cross slope (superelevation) as the arc. The following figure shows the top and side elevations of a circular road, including control points and **Entry Slope** and **Exit Slope** lines.



Creating Hills

Hill sections model simple vertical curves that have no changes in horizontal alignment. A hill's profile is defined by the **Vertical Curve Controls** and three control points—a beginning, a slope intersection, and an ending.

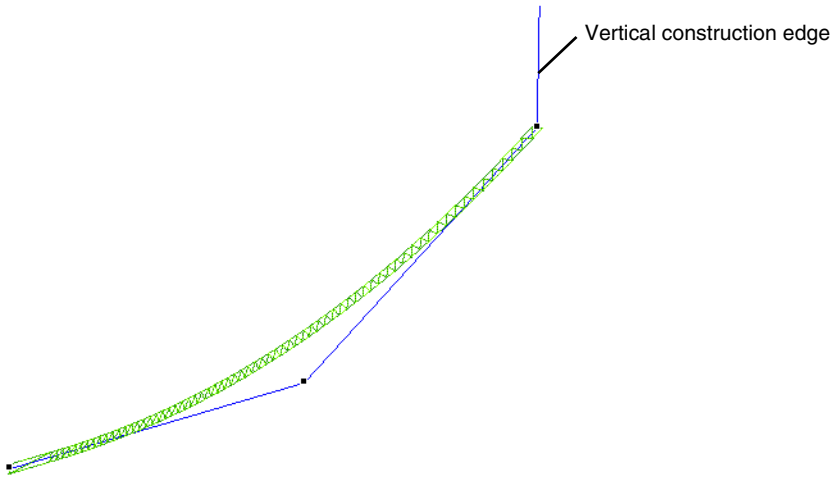
To Create a Hill

Step 1. If this hill is the first section in the road, position the tracking plane to an alignment that is appropriate for the curve you want to draw, and zoom out the eyepoint. the tracking plane automatically adjusts for subsequent sections (unless **Trackplane Control** is cleared).

Step 2. Choose **Road/Construction Tool**. The *Road Construction* dialog box opens.

Step 3. In **Section Creation**, set **Type** to Hill.

Step 4. Specify the **Entry**, **Alignment**, and **Exit** points on the tracking plane. The road section appears in the Graphics view. If the road section is valid it displays in bright green. If it is not a valid road, it displays in red.



To reposition a point, hold down Ctrl and click the point to select it, then specify the new position or drag the point to the new position.

Step 5. Grab the vertical construction edge with the middle mouse button and raise the elevation to form the hill you want.

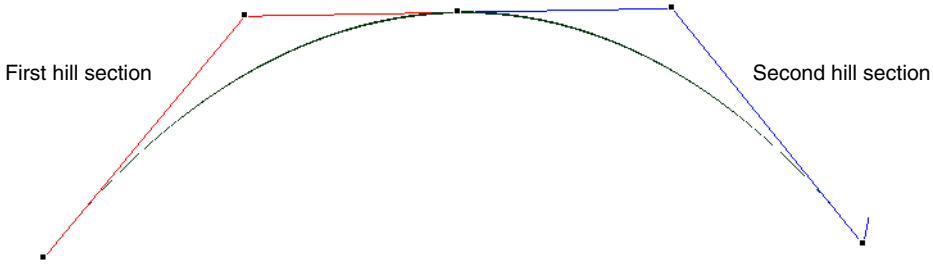
Step 6. Enter values for the **Vertical Curve Controls** or drag the sliders to modify the entry and exit slopes, curve length, and maximum curve. See "Vertical Curve Controls" on page 1239 for more information.

Step 7. Click **OK** to build the road.

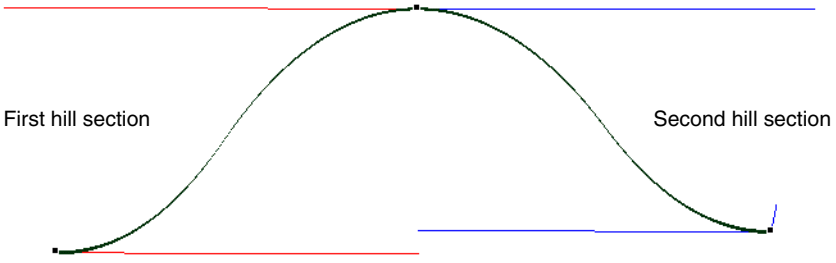
If **Auto Tessellate** is selected, the current road style definition is automatically applied when the road is built. If **Auto Tessellate** is not selected, you must open the **Tessellation Tool**, define the road style, and then apply it to the road section.

Special Hills

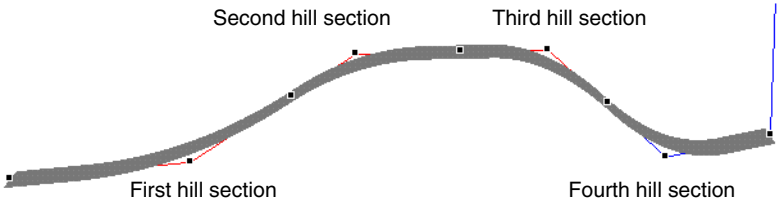
To construct a hill that goes up and down, you must create at least two consecutive hills.



If two curves are used to construct each hill section, the result is an up-and-down hill with sags and crests.



In the following illustration, four hill sections are used to construct an up-and-down hill, precisely controlling the location and amount of crest and sag.



Creating Straight Sections

To create a straight section, define the **Entry** (beginning) and **Exit** (ending) points. If the section is a continuation from other road sections, only the **Exit** point is required.

Modifying Roads

You can modify a created road by selecting any part of it (including a face, edge, or vertex) and using the Construction Tool to change parameters for the sections you wish to change, or to add new road sections. The road appears in the Graphics view with the control points drawn along its entirety. Use **Next Section** or **Previous Section** to select the section of road you want to modify.

Related Topics

"To Modify a Road Section" on page 1250

To Modify a Road Section

Step 1. Select an existing road section for editing and choose **Road/Construction Tool**. The road's control points are drawn, and the selected section of the road is active for modifying.

Step 2. The same operations originally used to create the road can be used to edit it. All design parameters can also be modified. The parameters you can modify depend upon the type of section selected (they will not be available if you cannot modify them). In other words, for straight sections you can only modify the **Entry** and **Exit** points; for curves, you can modify all

controls points and both **Horizontal** and **Vertical Curve** controls; and for hills, you can modify all control points, but only **Vertical Curve** controls.

To reposition a point, hold down Ctrl and click the point to select it, then specify the new position or drag the point to the new position

Use **Previous Section** and **Next Section** to select other sections and modify them. You can also append new road sections to the end of the road.

Step 3. Click **OK** when modifications and additions are complete.

Tips for Good Road Design

There are a few things to consider when designing a road. The speed of cars using the road and the climate of the area in which the road is being built will affect the arc radius, spiral transition and superelevation values.

Superelevation

Nearly all curves on modern roadways employ some level of superelevation. The rate of superelevation is a function of both the radius of the curve and its design speed. Proper values for superelevation can be determined using formulas or curve tables in a highway design manual such as the Policy on Geometric Design, published by the American Association of State Highway and Transportation Officials (AASHTO).

The maximum superelevation allowed on most public roads is 0.12. This is a maximum 12% bank for a highway in a warm climate. The recommended maximum superelevation for roads that are subject to snow or icy conditions is 8%, because at a low speed, cars slide off a road with a 12% bank.

Spiral Transitions

Spiral sections provide easement for horizontal curves, introducing curvature and banking. A large bank combined with a short spiral results in an abrupt transition.

Spiral lengths are typically the distance covered in four seconds of travel time at the design speed of the road. For example, at 65 m.p.h. (approximately 29 meters per

second), 4 seconds imply 116 meters of spiral transition at each end of the curve. Spiral lengths are measured along the centerline of the road.

Balanced spirals (the same spiral length at each end of the curve) are desirable, but not required. In real life, the terrain, easements, and cost of property play an important role in determining the layout of curves (and all other road sections).

If you have a dramatic spiral, you can use **Spiral Subdivision** in the Tessellation Tool's *Road LOD* dialog box to make a smoother polygonal model. When you cut more polygons, the road surface is less angular, but rendering additional polygons is more taxing on a realtime system. The default method for mapping the road surface is with curvature-based polygonization. The *Road LOD* dialog box also provides a **Spiral Type** option for custom control of the geometry and polygonization of spirals.

Arc Radius

The arc radius for a curve at a given speed is determined by a table of standards. For a given design speed and maximum superelevation, there are safety regulations that dictate what the minimum arc radius can be. For example, if a freeway has a maximum superelevation of 8% and a design speed of 120 km/hr, AASHTO recommends a minimum arc radius of 655 meters. For details on this and other design regulations, refer to an appropriate highway design policy or manual.

A short arc radius results in a tighter curve. For a given design speed, a tighter curve requires a greater rate of superelevation than a more gradual curve. In gaming simulation, a curve with a short arc radius and high superelevation offers a more dramatic driving experience.

Road Tessellation

When you construct a road using the Construction Tool, you have only defined its length and any horizontal or vertical characteristics (curve and slope). It appears in the Graphics view as a flat ribbon. You must use the Tessellation Tool to define the road's attributes (such as its width and the width of its side area), define its appearance at different levels of detail, add features such as lightposts, traffic signs, or guard rails, and then apply these definitions to your road.

You can create these definitions before creating a road and apply them later, or you can immediately apply them to any road section created using the **Auto Tessellate** menu option.

You can save the road attribute definitions in a setup file (*.rds), which can be loaded at any time, edited, and applied to new or existing roads.

Selecting Road Sections

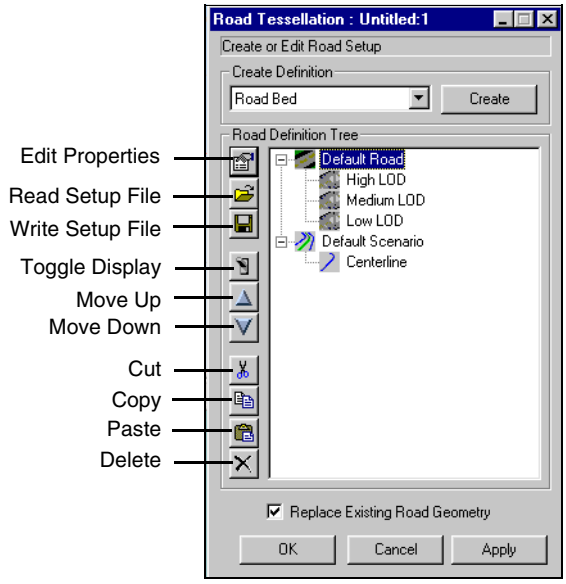


You can tessellate a section of created road, or the entire road, depending on how much you select. The **Road** mode tool allows you to select a road section, but you can select any part of a road using any mode in the hierarchy under **Road** mode (for example, Object or Face mode), or by selecting a road node in the Hierarchy view.

To select an entire road for modifying, select a portion of the road, and then choose **Road/Select Road**.

Using the Tessellation Tool

Selecting **Tessellation Tool** from the **Road** menu opens the *Road Tessellation* dialog box. The road components appear in a tree structure. Buttons to the left side of the Road Definition Tree allow you to create and edit definitions, save them, move the components up and down the hierarchy, and toggle the display of components on or off.



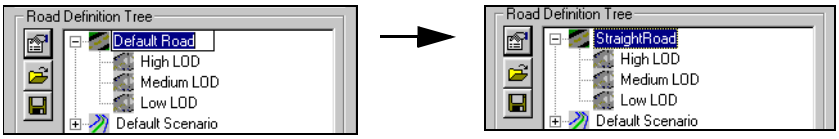
You can change the name of any road component to be more meaningful.

Each branch of the **Road Definition Tree** can be saved independently in its own setup file. You can use the **Cut**, **Copy**, and **Paste** functions to create new definitions.

To Rename a Road Component

Step 1. In the **Road Definition Tree**, click a road component name once to highlight it. Click it again to activate edit mode, and then type a new name for the road. Spaces are allowed, but backslashes (\) and colons (:) are not allowed.

Step 2. Click anywhere in the **Road Definition Tree** box to accept the change. Pressing Enter accepts the new name and opens the road component's definition setup.



Tessellation Tool Controls

This section describes the Tessellation Tool controls that are used to create and apply attributes to a road.

Create Definition

A list box of road components that can be added to the **Road Definition Tree**.

Create

Adds the selected road component to the **Road Definition Tree** and opens the corresponding component's dialog box.

Edit Properties



Opens the setup definition of the selected road component.

Read Setup File



Loads a saved road definition setup file (*.rds).

Write Setup File



Saves the current road definition in a setup file (*.rds).

Toggle Display



Turns the display of the selected road component and its children on or off (an LOD with Features, for example). When a component is toggled off, attributes are not applied during tessellation. The road definition setup file preserves this setting.

Move Up



Moves the selected road component to the next higher position in the hierarchy (if it is legal).

Move Down



Moves the selected road component to the next lower position in the hierarchy (if it is legal).

Cut



Removes the selected road component from the hierarchy and places it on the clipboard.

Copy



Copies the selected road component and places the copy on the clipboard.

Paste



Places the road component on the clipboard at the specified location in the **Road Definition Tree**.

Delete



Removes the selected road component from the Road Definition Tree.

Replace Existing Road Geometry

When set applies all road definitions to the selected road (or road section). This is the default setting. If this checkbox is clear, it applies only the new definitions. For example, if you created a series of lightposts along the road, clearing **Replace Existing Road Geometry** would only apply the lightposts. Setting **Replace Existing Road Geometry** would reapply all the definitions as well as the lightposts.

Apply

Applies the tessellation to the selected road section and allows you to view the effect without exiting the Tessellation Tool.

Defining the Road Style

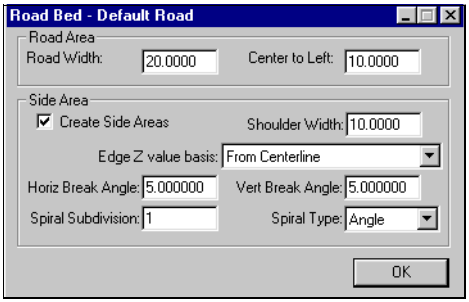
Road Bed attributes define the general characteristics of the road. These settings are overridden by the settings in the LOD definitions.

To Create a Road Definition

Step 1. Choose **Road/Tessellation Tool**. The *Road Tessellation* window opens.

Step 2. Click **Default Road** in the **Road Definition Tree** and click **Edit Properties**. The *Road Bed* dialog box opens.





Step 3. In the **Road Area** section, set the **Road Width** and **Center to Left**.

Step 4. In the **Side Area** section, set **Create Side Areas** if you want your road to have a shoulder, specify a **Shoulder Width**, and then click **OK**.

Road Bed Parameters

This section describes the *Road Bed* dialog box parameters that define the appearance of a road.

Road Width

Width of the road, in database units.

Center to Left

Distance from the center of the road to the left edge, in database units.

Create Side Areas

Specifies whether the road has a shoulder on each side.

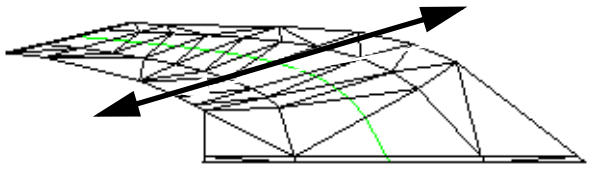
Shoulder Width

The width of the shoulder area. This field is not available if **Create Side Areas** is not set.

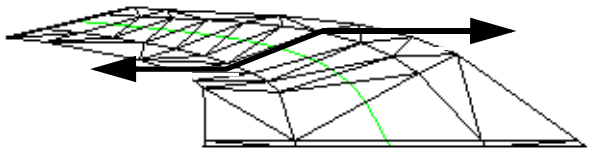
Edge Z Value Basis

On superelevated roads, determines whether the side areas will be banked, and how the banking will occur.

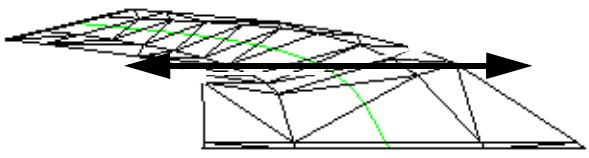
- **No change (level with road)** - If the road is banked (superelevated), the side areas will be banked to remain at the same angle as the road.



- **From Road Edge** - The side areas extend out flat from the edge of the road.



- **From Centerline** - The far edges of the side areas are set to the *z* value of the road's centerline.



Horizontal Break Angle and Vertical Break Angle

Controls the construction of the polygons that form the surfaces of horizontal and vertical curves in the side area. The smaller the angle you specify (in degrees), the larger the number of polygons produced.

> **Note: Horizontal Break Angle**, **Vertical Break Angle**, **Spiral Subdivision**, and **Spiral Type** values must be larger than the lowest LOD tessellation used.

Spiral Subdivision

Smooths the transition through the spiral section of a curve by dividing the side area into smaller polygons. To create smaller polygons within the spiral segment, the **Horizontal Break Angle** and **Vertical Break Angle** values are divided by the **Spiral Subdivision** value.

Spiral Type

Specifies how the spiral segments are divided into polygons. This has no effect on the basic geometry of the spiral, but only on how polygons are used to surface that geometry.

- **Angle** – Creates polygons according to the **Horizontal Break Angle** curvature restraint. This generates more polygons for areas of high curvature, and fewer polygons for areas of low curvature.
- **Distance** – Uses the same length along the centerline for each polygon in the spiral segment. This length is equal to the length of the polygons in the curved arc segment.

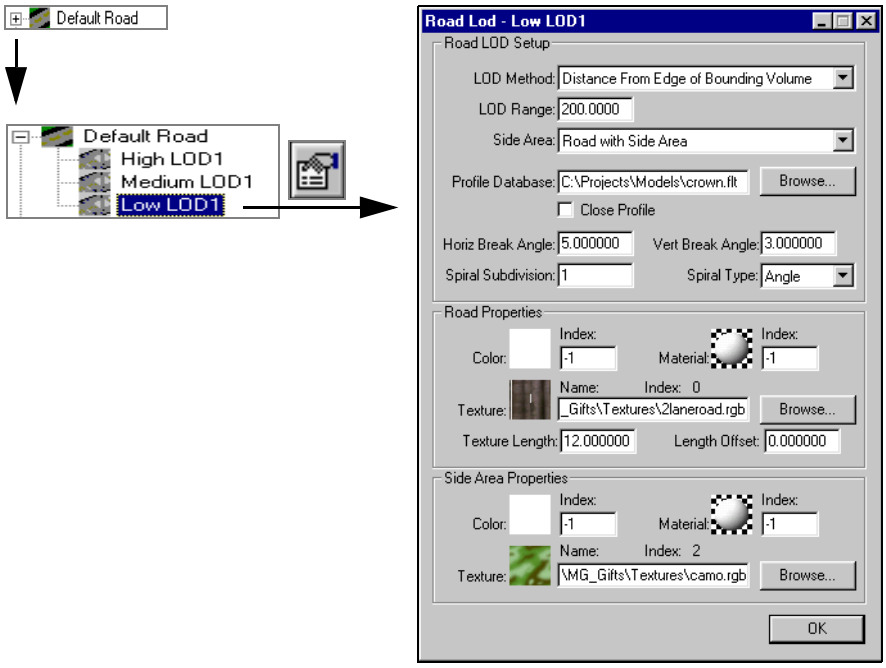
Defining Levels of Detail for Roads

Levels of detail permit a realtime system to switch between more or less complex versions of a model, depending on the distance from the model to the eyepoint. For more information about levels of detail, see “Levels of Detail” on page 707.

You can define any number of levels of detail for each section of road you create.

To Define a Road Level of Detail

Step 1. Click the cross to the left of **Default Road** to extend the tree, then select **Low LOD** and click **Edit Properties**. The *Road LOD - Low LOD* dialog box opens.



Step 2. Set the **LOD Method** and **LOD Range**, or accept the default setting. See "Road LOD Parameters" on page 1261 for descriptions of **LOD Methods** and their relation to the **LOD Range** setting.

Step 3. In the **Road Properties** section, specify a **Color**, if desired. To do this, right-click to choose a material from the Quick palette (see the "Using Quick Palettes" on page 358for information about using Quick palettes). If you want, you can also double-click the sample to open the Color palette. Choose a color, and click the sample again to apply the new current color to the **Road Properties**.

Step 4. Specify a **Material** property, if desired, by clicking the sample to apply the current material to the **Road Properties** or by choosing a material from the Quick palette.

Step 5. Choose a texture pattern from the Quick palette, or click **Browse**, select an appropriate texture pattern in the Texture palette, and then click **Open**.

Step 6. Set **Texture Length** and **Texture Offset**.

Step 7. In the **Side Area Properties** section, specify **Color** and **Material** properties, if desired.

Step 8. Choose a side area texture pattern from the Quick palette, or double-click the **Texture** sample to open the Texture palette, select an appropriate texture pattern to apply, and then click the **Texture** sample again to apply the texture.

Step 9. Set **Texture Length** and **Texture Offset**.

Step 10. Click **OK** to accept the new LOD definition.

Road LOD Parameters

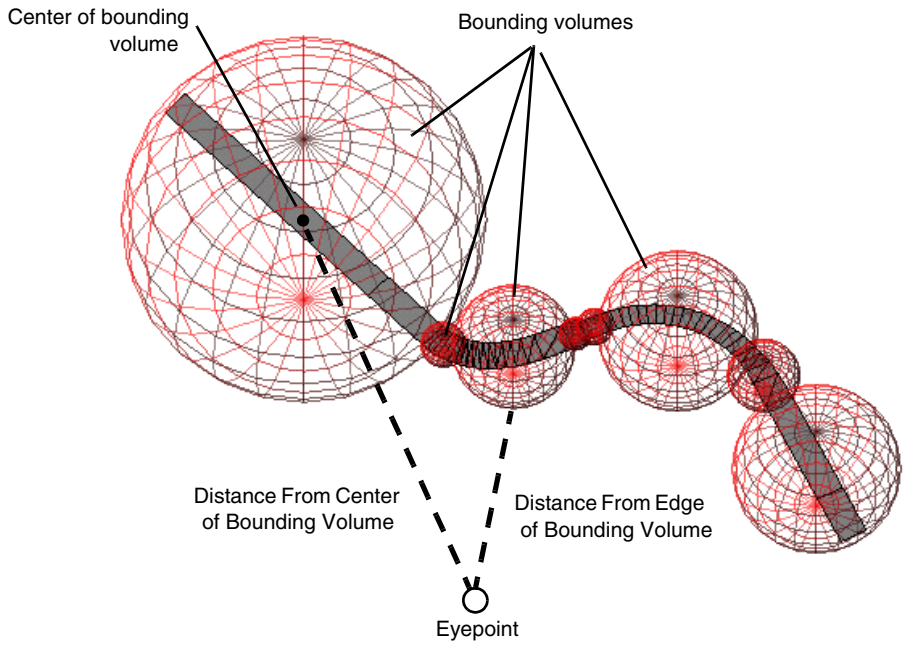
This section describes the parameters in the *Road LOD* dialog box.

The dialog box is divided into two sections: Road LOD Setup and Road Properties. Road Properties specify which color, material and texture properties are applied to the road surface, and control how the texture is applied.

LOD Method

Specifies how the level of detail will be switched in.

- **Distance From Center of Bounding Volume** – Switches in at the **LOD Range** setting by measuring the distance from the eyepoint to the center of the bounding volume. This is the normal switch in method for levels of detail.



- **Multiple of Bounding Volume Size** – Multiplies the size of the bounding volume by the **LOD Range** value.
- **Distance From Edge of Bounding Volume** – Calculates the distance from the eyepoint to the edge of each bounding volume.

LOD Range

Used together with LOD Method to calculate the farthest distance from the eyepoint at which the level of detail is displayed (switch-in distance).

Side Area

Defines how the road and side areas will appear when the LOD switches in.

- **Road with Side Area** - Appears as a road with side areas
- **Road Extends into Side Area** - The road's width includes the side areas. Uses the Road Bed Side Area definition for tessellation. Any LOD tessellation definitions for side areas are ignored.

- **Side Areas Only** - Only the side areas are visible. This option is very useful when multiple road beds have been defined.

Profile Database

An OpenFlight file that is used to define a cross-section with any number of points, rather than a simple, flat two-point one (see “Lofting a Road” on page 1265).

Close Profile

When set, all edges of the polygons in the profile database are used to loft the road surface. When cleared, the last edge is not used to loft the road. This is desirable when the road is built on the ground, where you will not see the bottom edge.

Horizontal Break Angle and Vertical Break Angle

Controls the construction of the polygons that form the surfaces of horizontal and vertical curves. The smaller the angle you specify (in degrees), the larger the number of polygons produced. Maximizing rendering performance requires some subjective judgments by the modeler. The **Horizontal Break Angle** and **Vertical Break Angle** values should be set as high as possible to maintain acceptable visual quality when switching between level of detail, while minimizing the number of polygons. A few visual tests on the target realtime software and display system are recommended to optimize these settings for your application.

Spiral Subdivision

Allows a smoother transition through the spiral sections of curves, by dividing road surface polygons into smaller polygons. For example, when a high rate of twist (rapid superelevation) is modeled, large polygons must cover surfaces that ascend or descend in steep angles.The abrupt angle changes between the polygons produce rough spots (bumps) in the visual model of the road. To create smaller polygons within the spiral segment, the **Horizontal Break Angle** and **Vertical Break Angle** values are divided by the subdivide value. For example, a subdivide value of 2 would double the number of polygons used to represent the spiral section for that level of detail. While this results in a smoother driving surface, the extra polygons will impact system performance.

Spiral Type

Specifies how the spiral segments are divided into polygons. This has no effect on the basic geometry of the spiral, but only on how polygons are used to surface that geometry.

- **Angle** – Creates polygons according to the **Horizontal Break Angle** curvature restraint. This generates more polygons for areas of high curvature, and fewer polygons for areas of low curvature.
- **Length** – Uses the same length along the centerline for each polygon in the spiral segment. This length is equal to the length of the polygons in the curved arc segment.

Color

Specifies the color and its palette index number that will be applied to the road surface.

Material

Specifies the material property and its palette index number that will be applied to the road surface. Material properties can only be viewed if Calculate Shading is performed.

Texture

Specifies the texture pattern that will be applied to the road surface. You can specify either a texture pattern name or its index number.

Every texture pattern must be loaded in the database's Texture palette before it can be applied to a surface. When you specify a Road texture file or a **Side Area Texture** file, the texture palette is checked for the texture pattern. If it is not present, the pattern is automatically loaded into the Texture palette, if possible.

- If you type in a texture name without specifying a directory path the Texture palette is searched and the pattern is applied if a match is found. If the specified texture pattern has not been loaded in the Texture palette, the current texture is applied.
- If you type in a texture name without a full directory path, and there are multiple textures in the palette with the same name but different paths, the matching texture with the lowest index is used. To ensure that the correct texture file is used, it is best to specify a full path.
- If the texture pattern cannot be found, an error message is generated. Check the location of the texture pattern file, and enter the correct path.

Texture Length

Defines the length of one copy of the pattern. If this is less than the length of the road section, the pattern repeats, provided the **Wrap** method attribute is set to Repeat. (See the "Texture Attributes" on page 523 for more information on texture attributes.)

Length Offset

Defines the distance before the start of the road that the texture pattern starts. If this is not the first section of the road, **Length Offset** is automatically set to match the texture applied to the previous section.

Side Area Properties: Texture

Specifies the texture file to be applied to the side areas of the road if **Create Side Areas** is enabled in the *Road Bed* dialog box.

Side Area Properties: Color

Specifies the color and the palette index number that will be applied to the road's side area.

Side Area Properties: Material

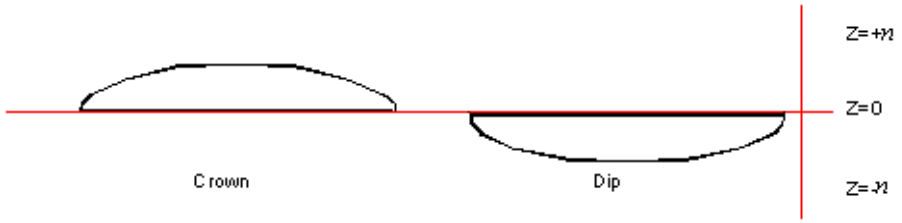
Specifies the material property and the palette index number that will be applied to the road's side area.

Lofting a Road

You can use a profile database to loft the road, rather than generating a flat surface. Be aware, however, that this produces more polygons.

To Loft a Road

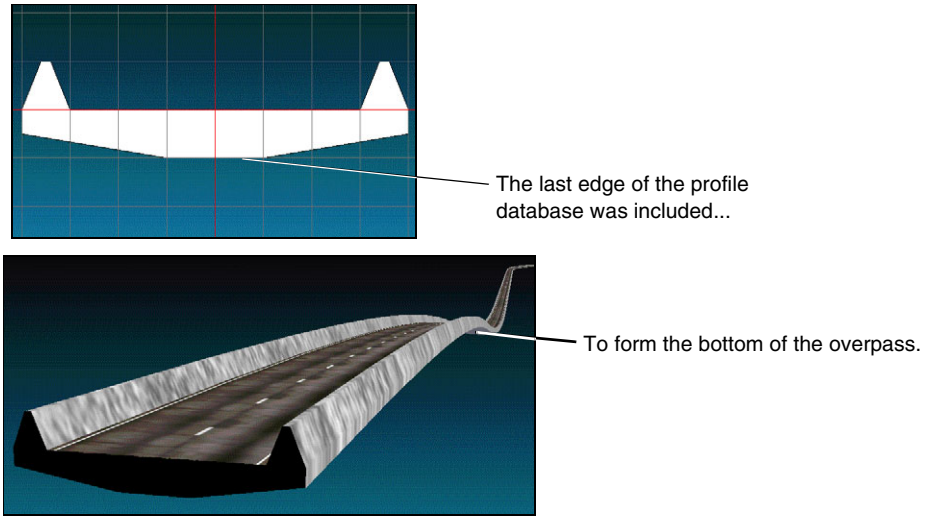
Step 1. Create a separate OpenFlight database that contains the road profile you want. A simple polygon, or a small group of polygons, works best. Convex or concave shapes can be used. All coordinates must lie in the *xz* plane. The vertex order must be left to right, and **Draw Mode** must be set to Wireframe when the file is saved.



Step 2. Open the *Road LOD* dialog box for the first level of detail in which you want to show a lofted road.

Step 3. Enter the name of the profile database in the **Profile Database** field.

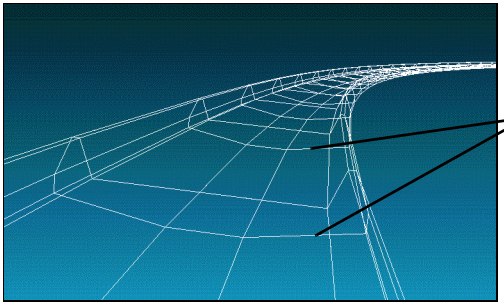
Step 4. Set or clear **Close Profile**. If this feature is set, all edges of the polygons in the profile database are used to loft the road surface. For example, in this figure, **Close Profile** was used to include the bottom of an overpass.



If **Close Profile** is cleared, the last edge is not used to loft the road. This is desirable when the road is built on the ground, where you will not see the bottom edge.

Pay careful attention to the order in which vertices are entered in each face of the profile database, because the "closing" edge between the last and first vertex is ignored when **Close Profile** is cleared.

Close Profile increases the number of polygons that form a road surface. It is best to enable this parameter only if the additional edge or edges are required to produce the roadway model you want.



Adding Features to a Road Definition

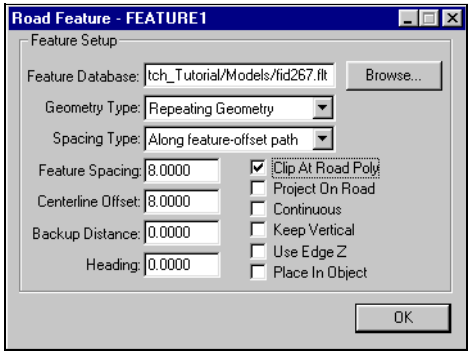
Features are three-dimensional database models, such as street lights and reflectors, that can be added to the tessellation tree at the root or LOD level. You can create up to 10 features at each level. The definitions can be saved in a setup (.rds) file.

Features can be applied as geometry in the current database, or as external references. Parameters in the *Road Feature* dialog box let you specify rotation and spacing.

To Create a Road Feature Definition

Step 1. Select an LOD in the tessellation tree where you want to add the feature. You can also place features at the root level - they are LOD-independent.

Step 2. Select **Feature** in the **Create Definition** menu, and then click **Create**. The *Road Feature* dialog box opens.



To select an existing set of feature parameters, double-click the feature in the **Road Definition Tree**.

Step 3. Enter the appropriate values for each parameter and click **OK** (see "Road Feature Parameters" on page 1268).



Step 4. To apply the feature to the current road, click **Apply**.

To Edit an Existing Road Feature Definition

Step 1. Double-click the feature in the tessellation tree, or select the feature and click **Edit**. The *Road Feature* dialog box opens.



Step 2. Modify the feature attributes as desired, and click **OK**.

To Copy a Road Feature Definition



Step 1. Select the feature in the tessellation tree and click **Copy**.

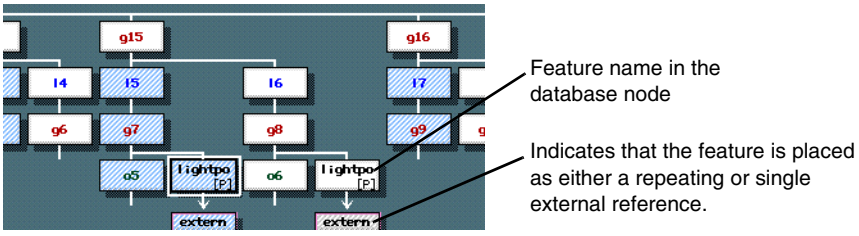
Step 2. Select the LOD where the feature will be assigned, and click **Paste**.

Road Feature Parameters

This section describes the parameters used to define road features in the *Road Feature* dialog box.

Feature Database

Specifies the file name of the feature's database (.flt) file. The database file name becomes the feature's attribute name and appears in the feature's database node.



Geometry Type

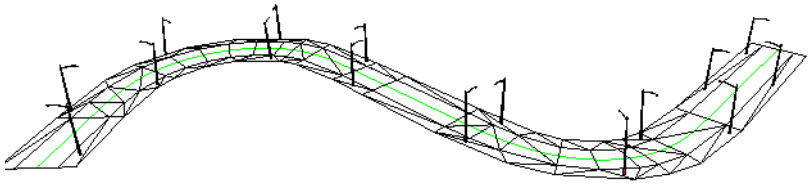
Geometry Type controls whether more than one copy of the feature is placed in the level of detail. These settings also determine whether the feature is inserted as geometry (that is, as a database model) or as a reference to an external database file. Each instance of the feature has its own database node.

- **Repeating XREF** - Inserts the feature as a series of external references.
- **Repeating Geometry** - Inserts the feature as a series of database models.
- **Single XREF -** Inserts the feature once per section, as an external reference.
- **Single Geometry** - Inserts the feature once per section, as a database model.

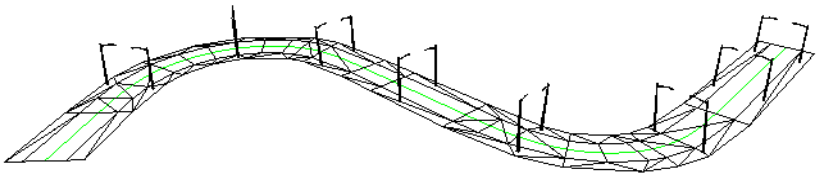
Spacing Type

Determines the manner in which the interval between features will be measured.

- **Along Feature Offset Path** - Measures the distance between features along the side area where the features are placed. When features are placed on a curved road using this spacing method, the features on both sides of the road are not aligned across from each other.



- **Along Centerline** - Uses the centerline to measure the distance between features. The features on both sides of the road line up directly across from each other.



Feature Spacing

Defines the interval at which repeated instances of the feature will be placed. The manner in which the distance is measured is determined by **Spacing Type**.

Centerline Offset

Defines the distance from the origin of the feature to the center of the road. A positive value places the feature to the right of center. A negative value places the feature to the left of center.

Backup Distance

Determines the offset of the road section into the feature's spacing distance. For example, if a feature is spaced every 100 meters, and you set a backup offset of 75, the first feature appears at 25 meters. This value updates automatically as each section of road is added to preserve the feature's spacing along the road.

Heading

Rotates the feature at the specified angle relative to the *z* axis in its original database file before placing the feature on or near the road. As you drive down a road, by default each feature is oriented showing the *xz* plane that contains the origin, viewed down the positive *y* axis of the original database file. This orientation follows the current heading of the road. (When the road curves, light posts follow, for example.)

Clip at Road Poly

Clips the feature at the boundary of each underlying polygon. For example, this might be used to apply a broken stripe to a roadbed. This option is only available when the feature is added to a level of detail, and **Geometry Type** is either Repeating Geometry or Single Geometry.

Project on Road

Adds two-dimensional features, such as road stripes, to the database. If this checkbox is set, the feature is projected onto the road polygons and attached as a subface of the road polygons. If the feature is three-dimensional, the object is flattened into a two-dimensional face as it is attached to the road polygons. This option is only available when the feature is added to a level of detail, and **Geometry Type** is either Repeating Geometry or Single Geometry.

Continuous

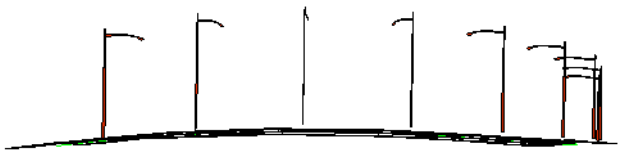
Defines features that flow continuously along a roadbed, such as median strips or stripes. If this checkbox is set, the feature is extended to completely cover each underlying polygon, and then sliced to fit it exactly. **Continuous** is available only when **Clip at Road Poly** is turned on. This option is only available when the feature is added to a level of detail, and **Geometry Type** is either Repeating Geometry or Single Geometry.

Place in Object

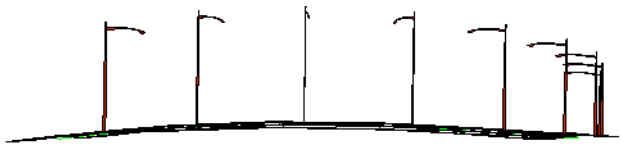
Discards the hierarchy contained in the feature and attaches its faces as siblings of the road faces. This can be used to fix the drawing priority of the feature relative to the road. **Place in Object** cannot be used if the feature is inserted as an external reference. This option is only available when the feature is added to a level of detail, and **Geometry Type** is either Repeating Geometry or Single Geometry.

Use Edge Z

Translates the bottom of the feature to the *z* value at the edge of the road, even if the feature is inserted away from the edge. For example, setting this checkbox prevents light posts from "floating" above a banked turn. If this checkbox is cleared, the bottom *z* of the feature is determined by its position on the plane that is defined by the bank of the road. This option is only available when the feature is added to a level-of-detail.



When Use Edge Z is cleared, features on an elevated curve can appear to be floating above the road



When Use Edge Z is set, features are planted on the road

Keep Vertical

Preserves the orientation of the feature relative to the *z* axis. If this checkbox is not set, the feature is oriented using the current bank and slope of the road.

Applying and Saving a Road Definition

Once you have defined your road components, you can apply them to a selected section of the road, or to the entire road. You can also apply the current road definition to any new road sections you create.

The definitions can be saved in a road definition setup file. Setup files can be called up and applied to new roads, or modified and saved as new setup files. The file name is appended with an .rds suffix and the file is written to the working directory, although you can save it to another location. For Windows systems, the working directory is the Multigen\Creator directory. For IRIX systems, the working directory is the directory where Creator was launched. You can restore the default road definition setup by deleting the file default.rds in the \MultiGen\Creator directory (Windows) or working directory (IRIX).

To Apply a Road Definition to an Existing Road



Step 1. In Road mode, select the road.

Step 2. Choose **Road/Tessellation Tool** to open the *Road Tessellation* dialog box.

Step 3. Click **Apply**.

To Automatically Apply a Road Definition

Choose **Road/Auto Tessellate**. A check mark appears in the menu next to this menu item. **Auto Tessellate** automatically applies the current road definition to any road you create.

To Save a Road Definition Setup

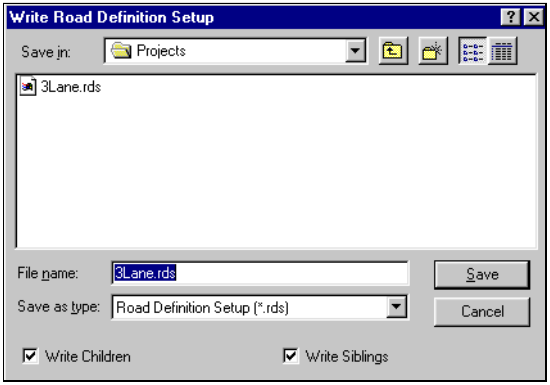
Step 1. Select the level in the **Road Definition Tree** that you want to save. For example, selecting **Default Road** saves the Road Bed and any Road LOD and feature definitions associated with it (if **Write Children** is set), but selecting an **LOD** or **Feature** only saves that setting.

Step 2. Set or clear the **Write Children** and **Write Siblings** checkboxes.

When **Write Children** is set, Creator saves setup definitions for the selected road component and any components associated with it, such as a road containing levels of detail and features.

When **Write Siblings** is set, Creator saves the setup definitions for the selected component and any other components at the same level in the hierarchy (features, for example).

Step 3. Click **Write** and enter a directory path and file name for the setup file in the *Write Road Definition* dialog box, and then click **Save**. The file name is saved with an .rds extension.



To Open a Road Definition Setup File

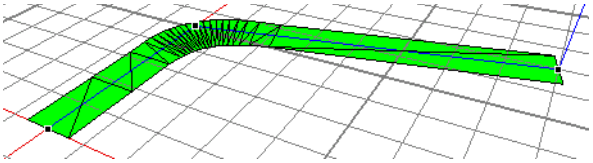
Step 1. Click **Read Setup File** in the *Road Tessellation* dialog box.

Step 2. In the *Open Road Definition Setup* dialog box, choose the setup file (.rds) you want, and click **Open**.

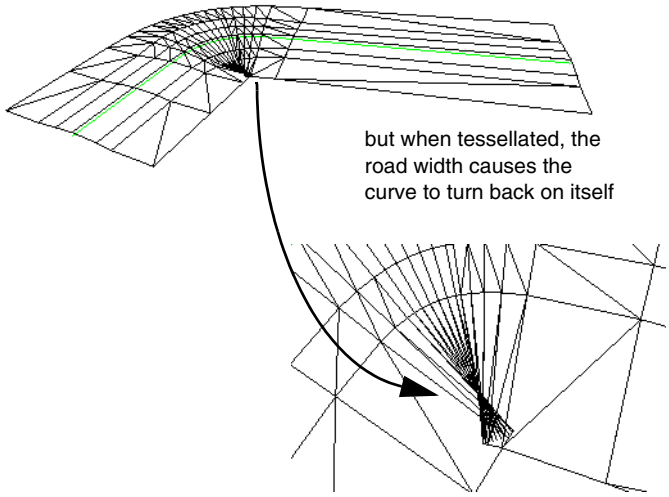
Tips for Tessellating Roads

Keep these tips in mind when defining road components and tessellating roads.

- Select the entire road when tessellating. Although you can select sections of a road for tessellating, textures applied in this manner may not line up correctly, and the distance between features may not be the same.
- Use the **Preview Tessellation** controls in the Construction Tool. Even though the road you build is mathematically logical (i.e., appears green in the Construction Tool), applying a wide width definition to a tightly curved road section can result in the curve turning back on itself, as shown below. This can produce undesirable results when features are added. Use **Preview Tessellation** to view the road at the width you think you will be using. Potential problems such as this one can be caught before the road is built.



Road appears green in the Construction Tool,



- When defining levels-of-detail, be sure the switch-in distance for the next higher level-of-detail does not over lap that of the lower level of detail. This can result in one or more levels-of-detail not switching in.

Driving the Road

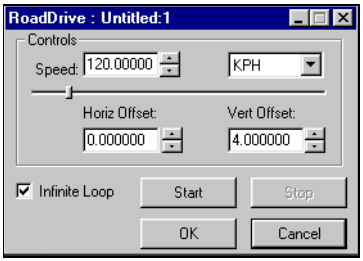
The driving path follows the eyepoint along the center of a lane (or at a specified offset), at a given speed, so you can check the characteristics of the road, and the placement of roadside features. Note that this routine does not include a simulation of vehicle dynamics.

To Drive Along a Roadway



Step 1. In Road mode, select a road section.

Step 2. Choose **Road/Drive Road**. The *Drive Road* dialog box appears.



Step 3. Enter a driving **Speed** and set the units to kilometers or miles per hour.

Step 4. Enter a value for **Horizontal** and **Vertical** offset, if desired. This is the eyepoint's distance from the centerline. When **Horizontal** and **Vertical** offset are 0, the eyepoint will be at the road's centerline.

Step 5. Set or clear the **Infinite Loop Drive** checkbox. When this checkbox is set, the road simulation routine drives to the end of the road and stops. When this checkbox is cleared, the road simulation program drives the road repeatedly, until you click **Stop**, **Done**, or **Cancel**.

Step 6. Click **Drive** to begin the drive simulation. To stop the action without closing the dialog box, click **Stop** at any time. Click **Drive** again to resume the drive simulation.

Step 7. Click **OK** or **Cancel** to stop the driving simulation and close the dialog box.

Generating Path Data

The **Road/Write Paths** menu option generates and writes path data in ASCII text data sets for use by vehicle dynamics and scenario control/traffic simulation subsystems in a driving simulator. These data sets contain raw analytic data. Creator does not include subsystem routines for processing or querying the data during actual realtime simulation—these routines must be created by the developer.

The two data sets are (1) roadway centerline files, and (2) scenario lane files. These data sets, described in the following sections, are correlated spatially with the roadway models in the visual database.

The **Write Path** source files can be modified or customized if you have installed the Creator API. The files are located in the \MultiGen\MgAPI\Sample_Plugins\Road_Export (Windows) directory. For IRIX, these files are automatically installed in /usr/local/multigen/creator.

Defining the Centerline

A roadway centerline is created for every road section, regardless of whether Scenario data has been applied. The centerline is a space curve defined by a sequence of coordinate points (x,*y*,*z*) with heading, pitch, and roll (**h**,**p**,**r**) orientation values. In the Graphics view, roadway centerlines are hidden, unclosed wireframe faces.

Data for an entire roadway, consisting of connected curves, hills, and straight sections, can be generated and written to a centerline file. The centerline data set is a compact curvilinear analytic model describing a roadway's 3D surface geometry and traffic lane definitions. This data can be used to support the realtime data requirements of both vehicle dynamics and scenario control/traffic simulation.

Centerline data can also be used to derive surface geometry (height and normal information) anywhere along the road path for vehicle dynamics/tire models of a driving simulator.

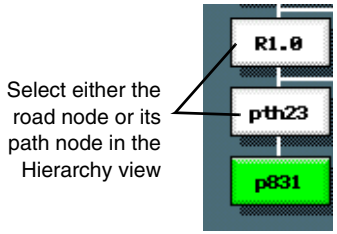
The curvature-based resolution of the sequence of points in the roadway centerline file can be specified in the Tessellation Tool's **Scenario** definition.

To Generate a Roadway Centerline Path File

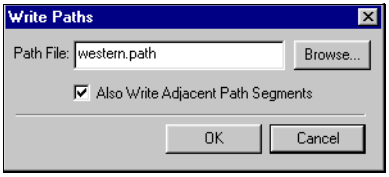
Roadway centerline data is written to an ASCII text file using **Road/Write Path**. The model for this data is based on a roadway centerline, with associated attributes.

Step 1. In Road mode, select the road section that you want to write. Alternatively, you can also select the road node or the road section's first path node in the Hierarchy view.





Step 2. Choose **Road/Write Paths**. The *Write Paths* dialog box opens.



Step 3. Enter a directory path and file name in the **Path File** field, or click **Browse** and select the path and file name of an existing roadway centerline file.

Step 4. Set **Write Adjacent Path Segments** if you wish to write centerline information for the entire road. When this checkbox is cleared, only the selected segment's information will be written to the centerline file.

Step 5. Click **OK** to create the roadway centerline file.

Roadway Centerline File Description

Roadway centerline data consists of the road's construction data– its control points, horizontal and vertical curve controls, and data from the **Scenario** and **Lane** definitions. profile database (the cross-section geometry that is lofted along the centerline to create the 3D surfaces of the roadway). This definition can include roadway crowning, barriers, shoulders, curbs, ditches, etc.

The following is an example of a roadway centerline file.


This is the first section (a curve) in Road #2 of this database. Section numbers start at zero. Road ID also appears on database bead
ROAD_ID: 2.0
ROAD_TYPE: Curve
ARC_RADIUS: 175.000000
SPIRAL_LEN1: 80.000000
SPIRAL_LEN2: 80.000000
SUPERELEVATION: 0.080000
CONTROL_POINT: 0.000000 300.000000 0.000000
VCURVE_LEN: 400.000000
VCURVE_MIN: 20.000000
SLOPE1: 0.000000
SLOPE2: 0.000000
Horizontal and vertical curve control values
WIDTH: 12.000000
CENTER2LEFT: 6.000000
Road width and centerline placement
NUM_LANES: 2
LANE_OFFSET: 1.825000
LANE_OFFSET: -1.825000
Lane data
PROFILE_POINT: 12.000000 0.000000
PROFILE_POINT: 9.823453 0.300000
PROFILE_POINT: 6.000000 0.500000
PROFILE_POINT: 3.200000 0.300000
PROFILE_POINT: 0.000000 0.000000
Lofting information, including the elevation points for the lofted section. Lofting information is only printed for the first section.
SPEED: 70.000000
Drive Road information)
NO_PASSING: TRUE
Heading, Pitch, and Roll data will be stored and reported
STORE_HPR: TRUE
NUM_POINTS: 15
Number of centerline points that follow (for parsing)
POINT: 0.000000 0.000000 0.000000 0.000000 0.000000 0.000000
POINT: 0.000000 83.548590 0.000000 0.000000 0.000000 0.000000
POINT: 2.906056 145.953096 0.000000 8.000000 0.000000 3.574879
POINT: 6.072530 153.131640 0.000000 13.096178 0.000000 4.573921
POINT: 13.249899 186.467582 0.000000 21.096178 0.000000 4.573921
POINT: 23.605154 208.577523 0.000000 29.096178 0.000000 4.573921
POINT: 35.936741 229.031118 0.000000 37.096180 0.000000 4.573921
POINT: 52.985177 247.430262 0.000000 45.096180 0.000000 4.573921
POINT: 71.438096 263.416837 0.000000 53.096180 0.000000 4.573921
POINT: 91.936333 276.679681 0.000000 61.096180 0.000000 4.573921
POINT: 114.080915 285.960649 0.000000 69.096176 0.000000 4.573921
POINT: 136.868360 293.927470 0.000000 76.903824 0.000000 4.573921
POINT: 156.586342 298.521248 0.000000 84.903824 0.000000 2.853243
POINT: 216.451410 300.000000 0.000000 90.000000 0.000000 0.000000
POINT: 300.000000 300.000000 0.000000 90.000000 0.000000 0.000000
Centerline data for each point, in the following order:
x, y, z, h, p, r

```
ROAD_ID: 2.1
ROAD_TYPE: Hill
VCURVE_LEN: 241.761226
VCURVE_MIN: 20.000000
SLOPE1: 0.000000
SLOPE2: 0.248179
WIDTH: 12.000000
CENTER2LEFT: 6.000000
NUM_LANES: 2
LANE_OFFSET: 1.825000
LANE_OFFSET: -1.825000
PROFILE_NAME: /usr/people/mike/road/crown.flt
SPEED: 70.00000
NO_PASSING: FALSE
STORE_HPR: TRUE
NUM_POINTS: 6
POINT: 300.000000 300.000000 0.000000 90.000000 0.000000 0.000000
POINT: 358.238774 300.000000 0.000000 90.000000 0.000000 0.000000
POINT: 426.357384 300.000000 2.381659 90.000000 4.000000 0.000000
POINT: 495.145434 300.000000 9.620488 90.000000 8.000000 0.000000
POINT: 565.298943 300.000000 22.005999 90.000000 12.000000 0.000000
POINT: 600.000000 300.000000 30.000000 90.000000 13.937990 0.000000
```

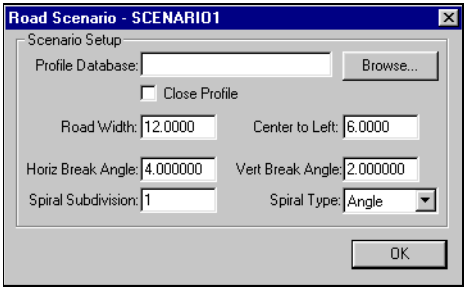
Data for the second section (a hill) in Road #2 of this database

```
ROAD_ID: 2.2
ROAD_TYPE: Straight
SLOPE1: 0.248179
SLOPE2: 0.248179
WIDTH: 12.000000
CENTER2LEFT: 6.000000
NUM_LANES: 2
LANE_OFFSET: 1.825000
LANE_OFFSET: -1.825000
PROFILE_NAME: /usr/people/mike/road/crown.flt
SPEED: 70.000000
NO_PASSING: FALSE
STORE_HPR: TRUE
NUM_POINTS: 2
POINT: 600.000000 300.000000 30.000000 90.000000 13.937990 0.000000
POINT: 697.055698 300.000000 54.087162 90.000000 13.937990 0.000000
```

Data for the third section (straight) in Road #2 of this database

Defining a Road Scenario

Road Scenarios contain information that is written out to the roadway centerline file, such as the profile database and its profile points. Information contained in the *Road Scenario* dialog box has no effect on the road's appearance within Creator.



The **Road Definition Tree** has a **Default Scenario** definition. You can rename and modify the existing scenario, or create a new one. However, until you create lanes, the only drivable path in a road is the centerline.

Scenario information is stored in the path node for the lane. The path node contains data about cross-section and lane definitions for one lane in a road section. The cross-section definition is the same as the profile database specified for the road's Scenario definition. This definition can include roadway crowning, barriers, shoulders, curbs, ditches, etc.

To Create a New Scenario Definition

Step 1. Choose **Road/Tessellation Tool**.

Step 2. Select **Scenario Data** in the **Create Definition** list box, and click **Create**. A new scenario is added to the **Road Definition Tree**, and the *Road Scenario* dialog box opens.

To Modify a Scenario Definition

Step 1. Double-click the **Scenario** in the **Road Definition Tree**. The *Road Scenario* dialog box opens.

Step 2. Enter the directory path and file name for the **Profile Database** you are using to loft the road, if any.

Step 3. Set or clear the **Close Profile** checkbox.

Step 4. Enter a **Road Width** and **Center to Left** distance.

Step 5. If desired, specify the **Horizontal Break Angle** and **Vertical Break Angle** for smoothing curves in the road.

Step 6. Specify the **Spiral Subdivision** and **Spiral Type**, if necessary, and then click **OK**.

Step 7. In the **Tessellation Tool**, click **Apply** to apply the new scenario data to the road.

Road Scenario Parameters

The *Road Scenario* dialog box has the following parameters.

Profile Database

An OpenFlight file used to loft the road rather than generating a flat surface.

Close Profile

When set, all edges of the polygons in the profile database are used to loft the road surface. When cleared, the last edge is not used to loft the road.

Road Width

The distance (in database units) from the left edge of the road to the right edge, on a vector perpendicular to the road's centerline.

Center to Left

The distance (in database units) from the centerline to the left edge of the road.

Horizontal and Vertical Break Angle

Controls the construction of the polygons that form the surfaces of horizontal and vertical curves. The smaller the angle specified (in degrees), the larger the number of polygons produced.

Spiral Subdivision

Allows a smoother transition through the spiral sections of curves by dividing the road surface polygons into smaller polygons. To create smaller polygons within the spiral segment, the **Horizontal Break Angle** and **Vertical Break Angle** values are divided by the subdivide value.

Spiral Type

Specifies how the spiral segments are divided into polygons.

- **Angle** – Creates polygons according to the **Horizontal Break Angle** curvature restraint. This method generates more polygons are areas of high curvature, and fewer polygons for areas of low curvature.
- **Distance** – Uses the same length along the centerline for each polygon in the spiral segment. This length is equal to the length of the polygons in the curved arc segment.

Defining Scenario Lanes

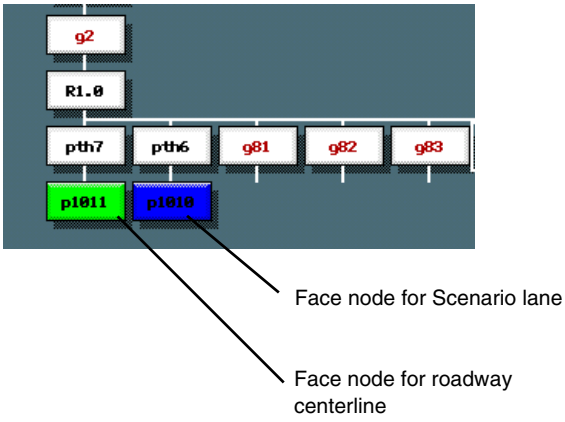
Lanes provide additional scenario paths that can be "driven" by the traffic animation software. Scenario paths are sequences of unclosed wireframe faces. Until you create lanes, the only drivable path in a road is the centerline. You can define any number of lanes in each scenario.

Scenario lanes contain lane information for traffic simulations. They do not contain road design parameters, cross-section data, or other surface information for vehicle dynamics processing, which are contained in **Road** and **Scenario** definitions.

In the Tessellation Tool's **Road Definition Tree**, lanes are children of Scenarios. The default lane is called the **Center Lane**, and its definition is the road's centerline.

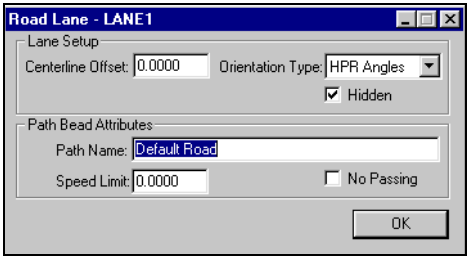
Lanes are defined as lateral offsets from the road centerline. When creating or modifying a **Lane**, Creator creates an unclosed wireframe face to represent the centerline of each lane in the section. The lane centerline is a hidden face.

In the Hierarchy view, Scenario lane information is stored in the face node under the scenario path node. Each lane created generates a new scenario path node and its child face node. Unlike the (default) bright green roadway centerline face nodes, lane face nodes are dark blue, as they appear in the Graphics view when hidden faces are displayed.



To Define a Lane

Step 1. In the **Road Definition Tree**, double-click **Lane**. The *Road Lane* dialog box opens.



Step 2. Set the **Centerline Offset** if desired. Enter a positive value to place the lane to the right of the center, or enter a negative value to place the lane to the left.

Step 3. Select an **Orientation Type**.

Step 4. Click **OK**.

Road Lane Parameters

This section describes the parameters in the *Road Lane* dialog box.

Centerline Offset

The distance from the center of the lane to the center of the road. When the offset is a non-zero value, the centerline appears in the Graphics view as a blue line. The first lane with a zero offset defines the road's centerline.

Orientation Type

Defines the road's surface along the points of the centerline.

- **Up Vector** displays a normalized vector at each point along the centerline.

  Note: When **Up Vector** is specified as the **Orientation Type**, scenario data is not written out with the path data.

- **HPR Angle** specifies Heading, Pitch, and Roll (*h*,*p*,*r*) at each point.

Hidden

When set, creates the lane centerline as hidden faces that can be displayed by choosing **View/Hidden Faces/Show Normal**. When this checkbox is cleared, the centerline is always visible in the Graphics view.

Path Name

The name of the Path.

Speed Limit

The speed at which the road will be driven in the realtime system or simulator.

No Passing

When set it initiates a flag in the realtime system.

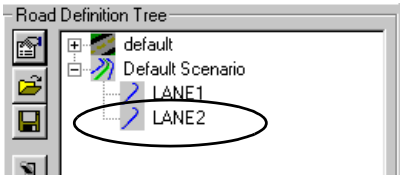
To Add a Scenario Lane and Apply it to a Road

Step 1. In Road mode, click anywhere on the road section you want to select, or select the road node in the Hierarchy view, and then choose **Road/Select Road**.



Step 2. Choose **Road/Tessellation Tool**. The *Road Tessellation* dialog box opens.

Step 3. Highlight **Default Scenario** in the **Road Definition Tree**, select **Lane** from the **Create Definition** list box, and then click **Create**. A Lane appears under Default Scenario and its *Road Lane* dialog box opens.



Step 4. Set the parameters in the *Road Lane* dialog box (see "Road Lane Parameters" on page 1285), and then click **OK**.

Step 5. Click **Apply** to apply the modified definitions to the road.

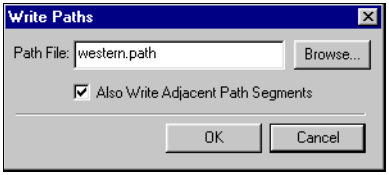
Creating a Scenario Lane File

Lane data provides a subset of the road centerline data described in "Defining the Centerline" on page 1277. Scenario lane data contains the list of points along the lane and a few lane attributes. It specifies Heading, Pitch, and Roll (*h*,*p*,*r*), or the surface normal vector (i,j,k) at each point along the lane centerline. Where the road centerlines have a single sequence of points along a spline and all lanes are expressed as parallel offsets, lane data separates sequences of points for each lane.

To Generate a Scenario Lane File

Step 1. In the Hierarchy view, select the path node of the lane to be written.

Step 2. Choose **Road/Write Paths**. The *Write Paths* dialog box opens.



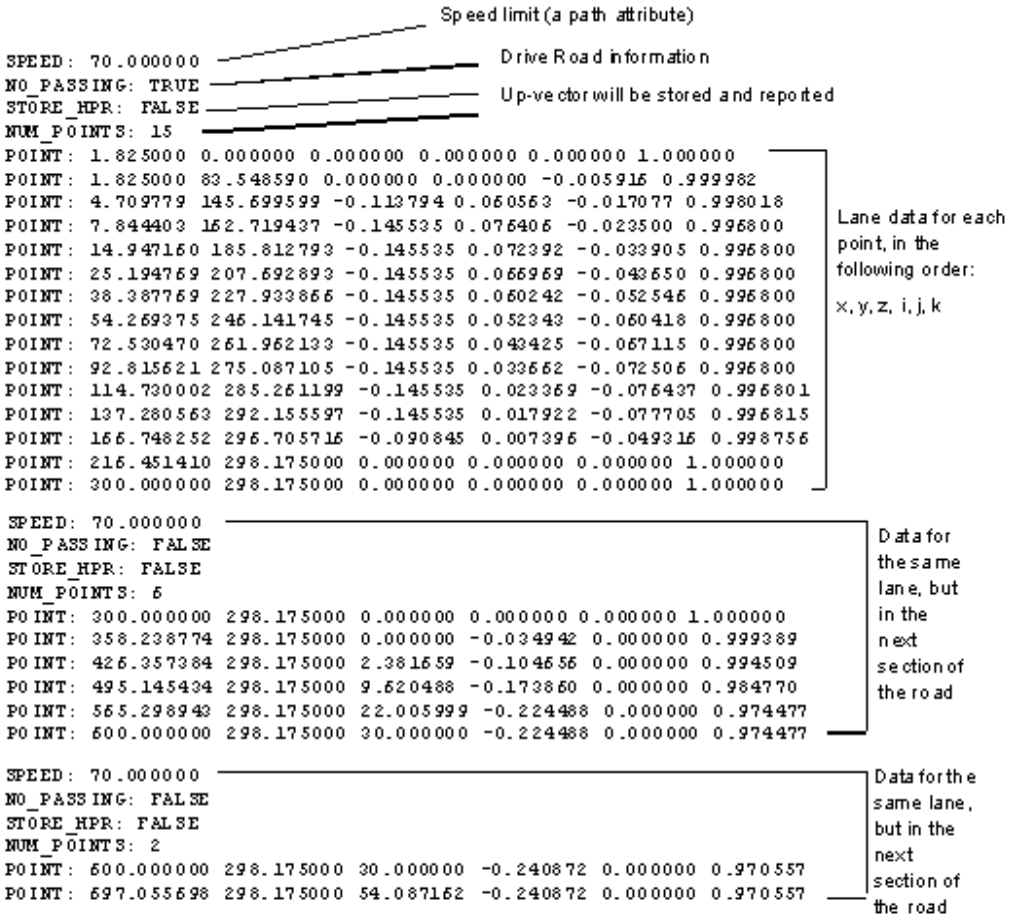
Step 3. Enter a directory path and file name in the **Path File** field, or click **Browse** and select the path and file name of an existing scenario lane file.

Step 4. Set the **Write Adjacent Path Segments** checkbox if you wish to write lane information for the entire road. When this checkbox is cleared, only the selected segment's lane information will be written to the scenario lane file.

Step 5. Click **OK** to create the scenario lane file.

Scenario Lane File Description

Scenario lane files are written out as formatted ASCII files. Each lane is written to a separate file. The following is an example of a scenario lane file.

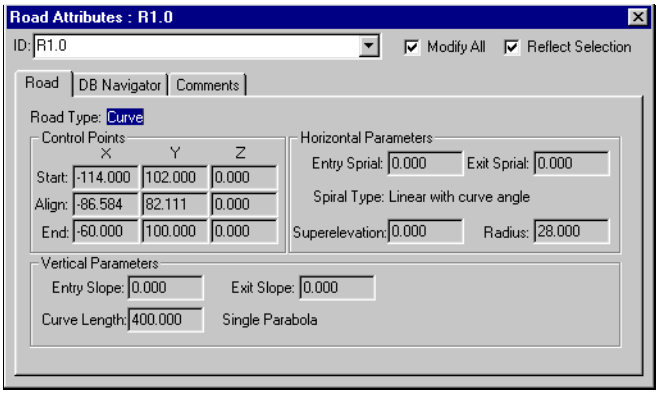

Speed limit (a path attribute)
Drive Road information
Up-vector will be stored and reported
SPEED: 70.000000
NO_PASSING: TRUE
STORE_HPR: FALSE
NUM_POINTS: 15
POINT: 1.825000 0.000000 0.000000 0.000000 0.000000 1.000000
POINT: 1.825000 83.548590 0.000000 0.000000 -0.005915 0.999982
POINT: 4.709779 145.699599 -0.113794 0.060563 -0.017077 0.998018
POINT: 7.844403 162.719437 -0.145535 0.076406 -0.023500 0.996800
POINT: 14.947160 185.812793 -0.145535 0.072392 -0.033905 0.996800
POINT: 25.194769 207.692893 -0.145535 0.066969 -0.043650 0.996800
POINT: 38.387769 227.933866 -0.145535 0.060242 -0.052546 0.996800
POINT: 54.269375 246.141745 -0.145535 0.052343 -0.060418 0.996800
POINT: 72.530470 261.962133 -0.145535 0.043425 -0.067115 0.996800
POINT: 92.815621 275.087105 -0.145535 0.033662 -0.072506 0.996800
POINT: 114.730002 285.261199 -0.145535 0.023369 -0.076437 0.996801
POINT: 137.280563 292.155597 -0.145535 0.017922 -0.077705 0.996815
POINT: 156.748252 296.705715 -0.090845 0.007396 -0.049315 0.998756
POINT: 216.451410 298.175000 0.000000 0.000000 0.000000 1.000000
POINT: 300.000000 298.175000 0.000000 0.000000 0.000000 1.000000
Lane data for each point, in the following order:
x, y, z, i, j, k
SPEED: 70.000000
NO_PASSING: FALSE
STORE_HPR: FALSE
NUM_POINTS: 6
POINT: 300.000000 298.175000 0.000000 0.000000 0.000000 1.000000
POINT: 358.238774 298.175000 0.000000 -0.034942 0.000000 0.999389
POINT: 426.357384 298.175000 2.381659 -0.104656 0.000000 0.994509
POINT: 495.145434 298.175000 9.620488 -0.173860 0.000000 0.984770
POINT: 565.298943 298.175000 22.005999 -0.224488 0.000000 0.974477
POINT: 600.000000 298.175000 30.000000 -0.224488 0.000000 0.974477
Data for the same lane, but in the next section of the road
SPEED: 70.000000
NO_PASSING: FALSE
STORE_HPR: FALSE
NUM_POINTS: 2
POINT: 600.000000 298.175000 30.000000 -0.240872 0.000000 0.970557
POINT: 697.055698 298.175000 54.087162 -0.240872 0.000000 0.970557
Data for the same lane, but in the next section of the road

Viewing Road Attributes

You can view the attributes assigned to each road section by selecting the section of road in the Graphics view (in Road mode), and then choosing **Attributes/Modify Attributes**, or by double-clicking the road node, or any of its children in the Hierarchy view. Although you can modify some of the road's attributes in the respective attributes windows, you should use the Tessellation Tool to change road definitions and regenerate the geometry in order to avoid problems.

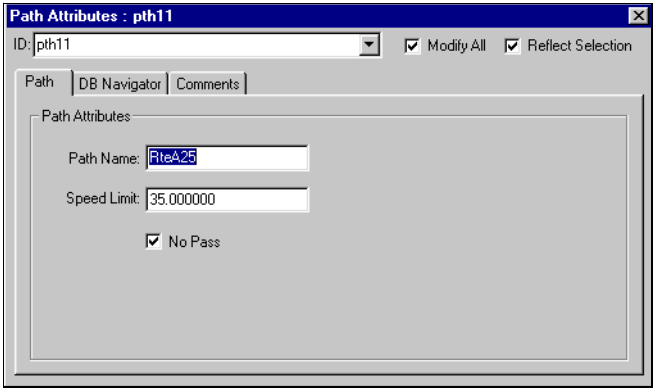
Road Attributes

The *Road Attributes* window displays the Construction Tool settings for the selected road section. It displays the type of road section, its control points, and its **Horizontal** and **Vertical** curve settings. The **Control Points** are the x, y, and z coordinates for each of the **Entry**, **Alignment** and **Exit** points specified in the **Construction Tool**. The information in *Road Attributes* windows can only be viewed. The fields cannot be edited. You must use the **Construction Tool** to modify the road section parameters.

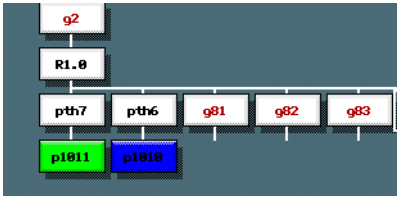


Path Attributes

The *Path Attributes* window contains lane information, such as the path name associated with the lane, the speed limit, and passing information.

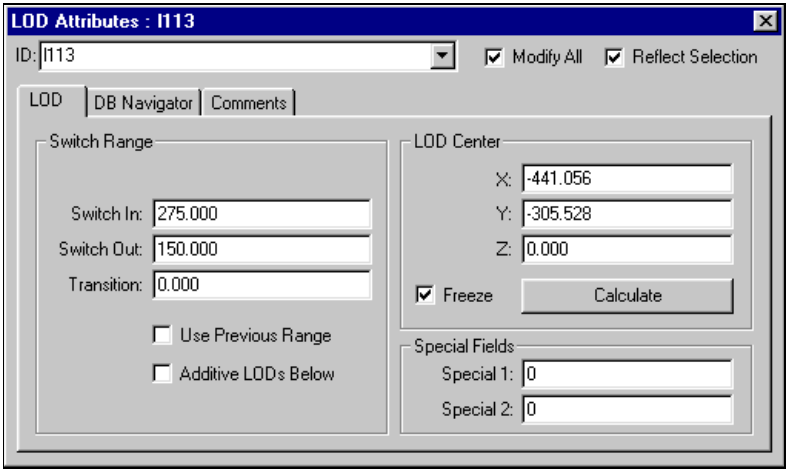


A path node is created for each lane. If no lanes are defined, or if none of the lanes created has a zero offset, a path node is created for the default centerline.



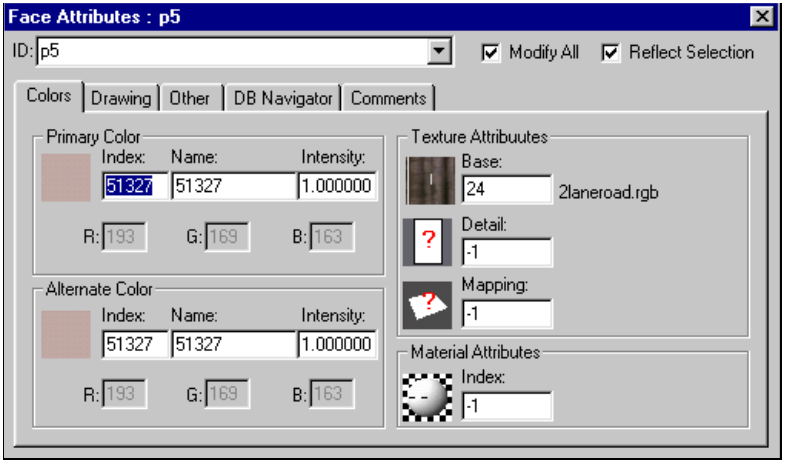
Road LOD Attributes

The Road LOD attributes contain the switch in and switch out range for each LOD, and the coordinates at the center of the bounding volume for that portion of the road section. The switch in and switch out distances may be different depending on the **LOD Method** selected when defining the LOD in the Tessellation Tool. See “Road LOD Parameters” on page 1261 for a description of how the switch in distances are calculated for each method.



Road Face Attributes

A road section's Face attributes contain the **Color, Texture** (for the road) and **Material** properties defined in the Tessellation Tool's *Road LOD* dialog box.



36 Creator Wizards

The **Geometry Tools** palette includes wizards for constructing simple models of trees or buildings. For descriptions of these wizards, see:

- "Billboard Wizard Tutorial" on page 1293
- "Tree Wizard Tutorial" on page 1304
- "Building Wizard Tutorial" on page 1324

Billboard Wizard Tutorial

When constructing an architectural scene, you may want to add trees or other vegetation. Unfortunately, detailed models of trees and plants take an enormous amount of time to construct, and they tax most desktop systems during runtime.

A simple solution is to map a tree texture to a *billboard* – a single polygon which rotates within the simulation, so that the image of the tree always faces the eyepoint. By mapping a texture of the tree to a single billboard polygon, the viewer sees a very detailed tree, which is always facing the eyepoint. When viewed from ordinary perspectives in a scene, such tree models appear realistic enough, while using only a small amount of system resources.

The Creator Billboard Wizard, which is located in the *Geometry* toolbox, constructs tree models quickly.

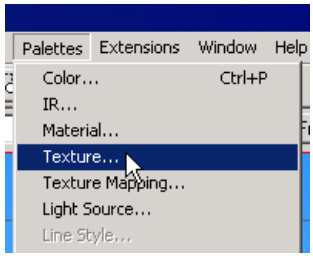
(Another solution is to map a tree texture to cross-hatched polygons. This method is employed in the "Tree Wizard Tutorial" on page 1304.)

Before using the Billboard Wizard, you should open the *Texture Palette* and select a texture for making your tree model. This process is described in the next section, "Preparing to Use the Billboard Wizard."

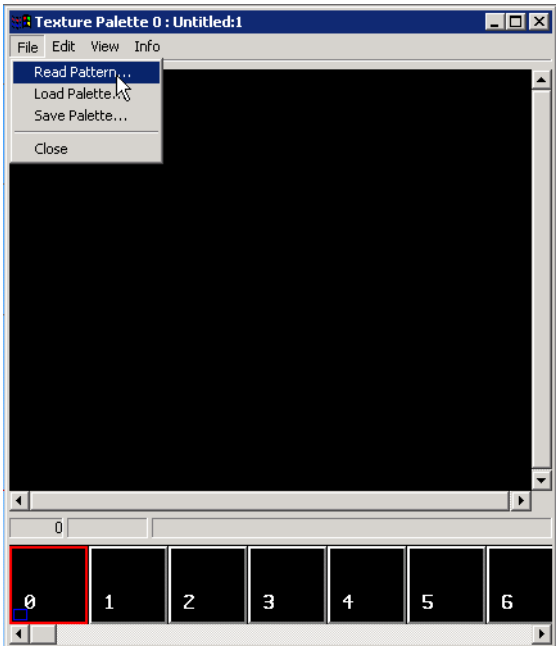
Preparing to Use the Billboard Wizard

Before opening the Billboard Wizard, you should select an image of a tree to use for your model. To do this, use the *Texture Palette.*

Step 1. In the Creator window, choose **Palettes/Texture** to display the *Texture Palette.*

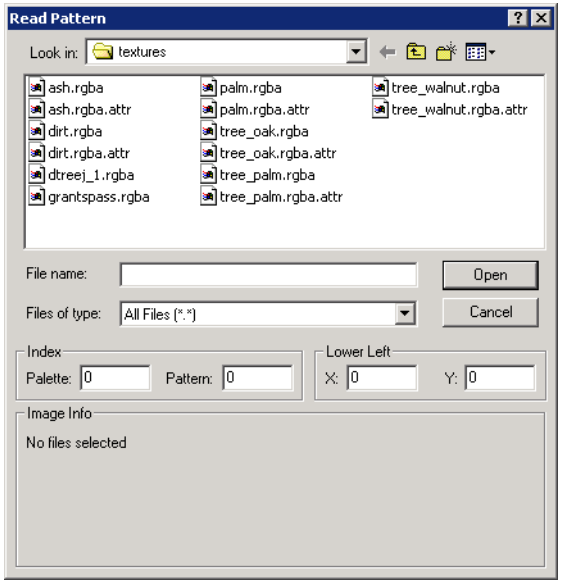


Step 2. In the *Texture Palette* window, choose **File/Read Pattern** to display the *Read Pattern* window.

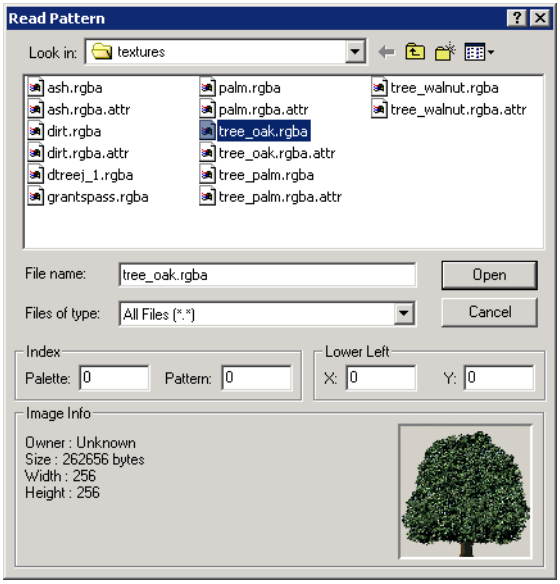


Step 3. In the *Read Pattern* window, navigate to the texture directory `MultiGen-Paradigm\resources\tutorials\creator\wizards\textures` on Windows, or `/usr/local/MultiGen-Paradigm/resources/tutorials/`

`creator/wizards/textures` on IRIX. (In the **Files of type** list, choose **All Files (*)**, if this is not your default setting already.)



Step 4. Choose `tree_oak.rgba`, and click **Open** to load the oak tree texture into the *Texture Palette*.



If you want to experiment with other image(s), you can also load additional textures, by selecting the next numbered box(es) in the *Texture Palette*, and choosing **File/Read Pattern...** for the additional image(s).

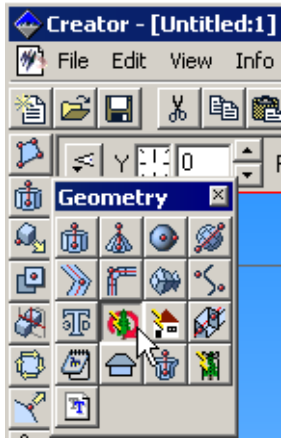
If you prefer to select a different tree image later in the process, you can do so. But we have opened the *Texture Palette* and selected an image now, so that you can complete perform the steps in the Tree Wizard without interruption.

You are now ready to proceed to the next section, “Running the Billboard Wizard.”

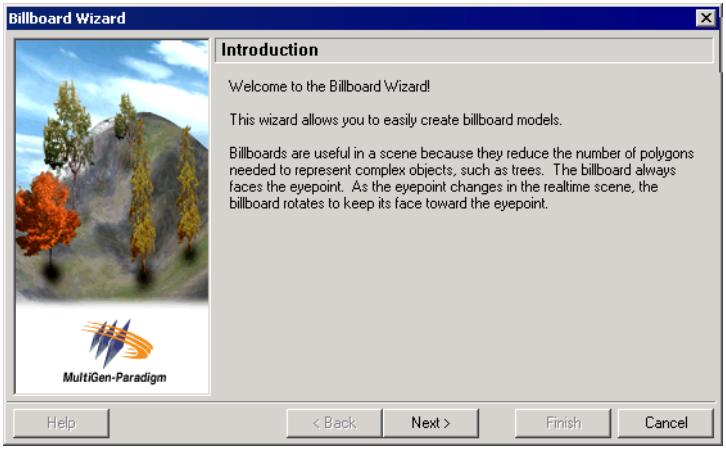
Running the Billboard Wizard

To open the Billboard Wizard:

Step 1. In the *Geometry* toolbox, choose the Billboard Wizard button.



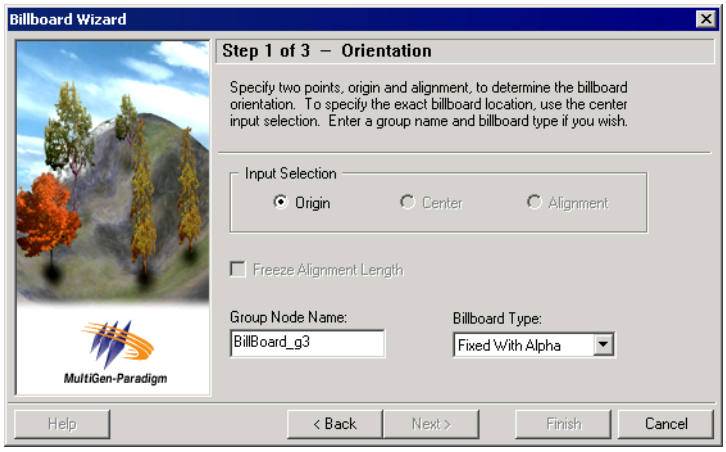
The first panel of the Billboard Wizard is displayed on your screen.



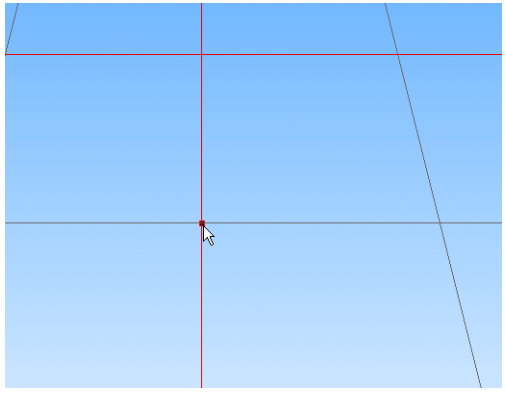
Step 2. Click **Next** to proceed to the first step for creating your tree.

Step 1 – Orientation

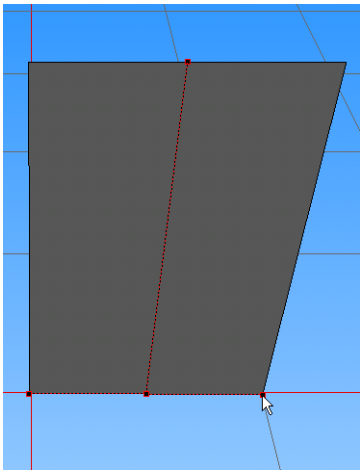
The orientation of your tree is determined by the origin and alignment you choose in this step.



Step 1. In the Creator window, move your cursor on the grid to the desired point of origin, and click once.

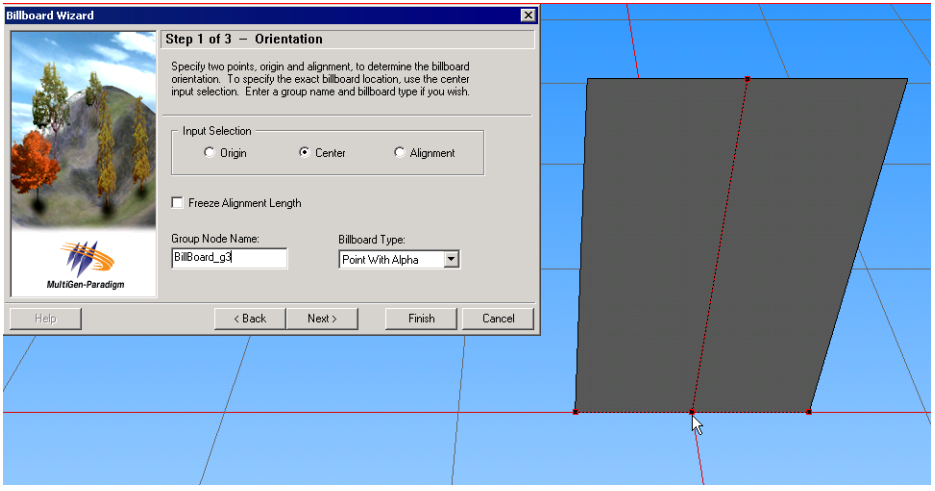


Step 2. Click on a second point to establish the desired alignment point.



Step 3. Click the **Center** option under **Input Selection** on the wizard. By choosing this option, you are centering the base of your billboard polygon onto a point you select.

Step 4. Click the desired center-point on the grid to position your model.



Step 5. In **Group Node Name**, enter a unique file name; for this tutorial, use "tree1". Please keep in mind that the default name, such as "Billboard_g3" is unique; when you choose another name, uniqueness in not guaranteed.

You need to ensure that you do not give the same name to two different groups.

Step 6. Leave the **Freeze Alignment Length** box unchecked for this tutorial.

Note: If you check the option for **Freeze Alignment Length**, your model will be tethered when the Alignment option is chosen; you can change the alignment point to reposition your model, but its length will not change. By leaving this option unchecked, the length of the model can be stretched by pulling on the alignment point.

Step 7. Choose **Axis With Alpha** under **Billboard Type**.

The **Axis** option causes the billboard to rotate about its axis to always face the viewer; you are choosing the **Axis** option with **With Alpha**, because you are using a texture with a transparent background. The other options include:

Fixed With No Alpha – makes the Billboard work like a normal polygon (using a texture *without* a transparent background).

Fixed With Alpha – makes the Billboard work like a normal polygon (using a texture *with* a transparent background).

Point With Alpha – makes the Billboard rotate about a point to always face the viewer (using a texture *with* a transparent background).

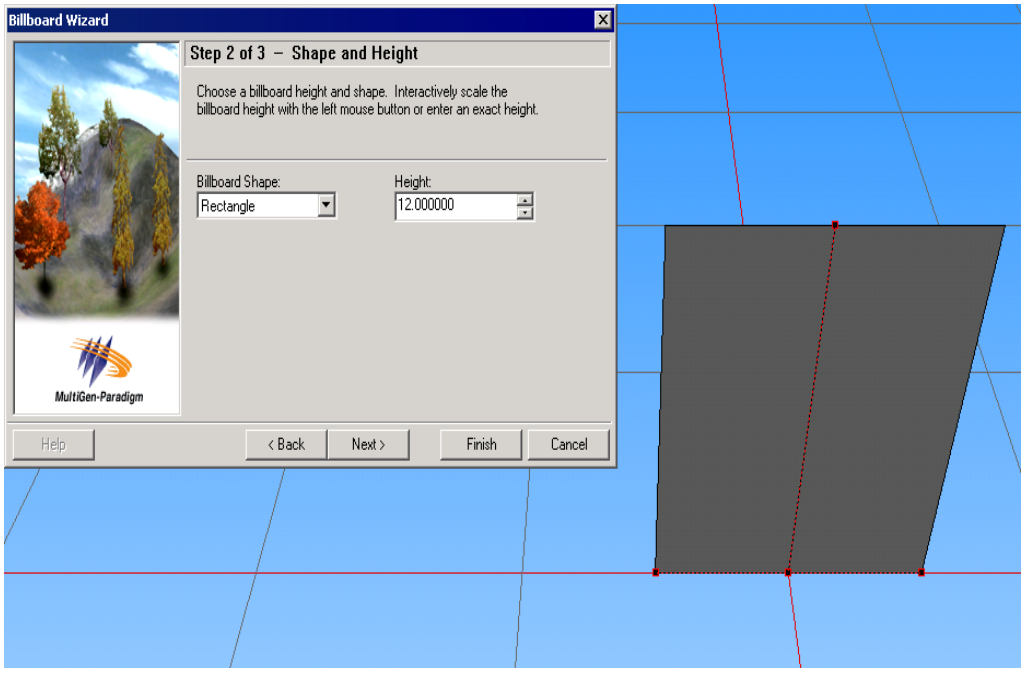
Note: The rotational effects specified by **Billboard Type** are not previewed in Creator. They are implemented by your realtime application.

8. Click **Next** to go to "Step 2 – Shape and Height."

Note: At any time while using the Wizard, you are free to go back or move forward, making additional adjustments, by clicking the **Back** and **Next** buttons at the bottom of the pane.

Step 2 – Shape and Height

Step 1. Choose **Rectangle** for the **Trunk shape type**. (This should be the default value, unless you have previously used the Tree Wizard and changed this selection.)



Step 2. Enter **12** (12.000000) for the **Height** of your tree model, or adjust the height with your mouse by clicking and dragging on the top point of the model. The **Height** is measured in database units; for this tutorial, the units are meters.

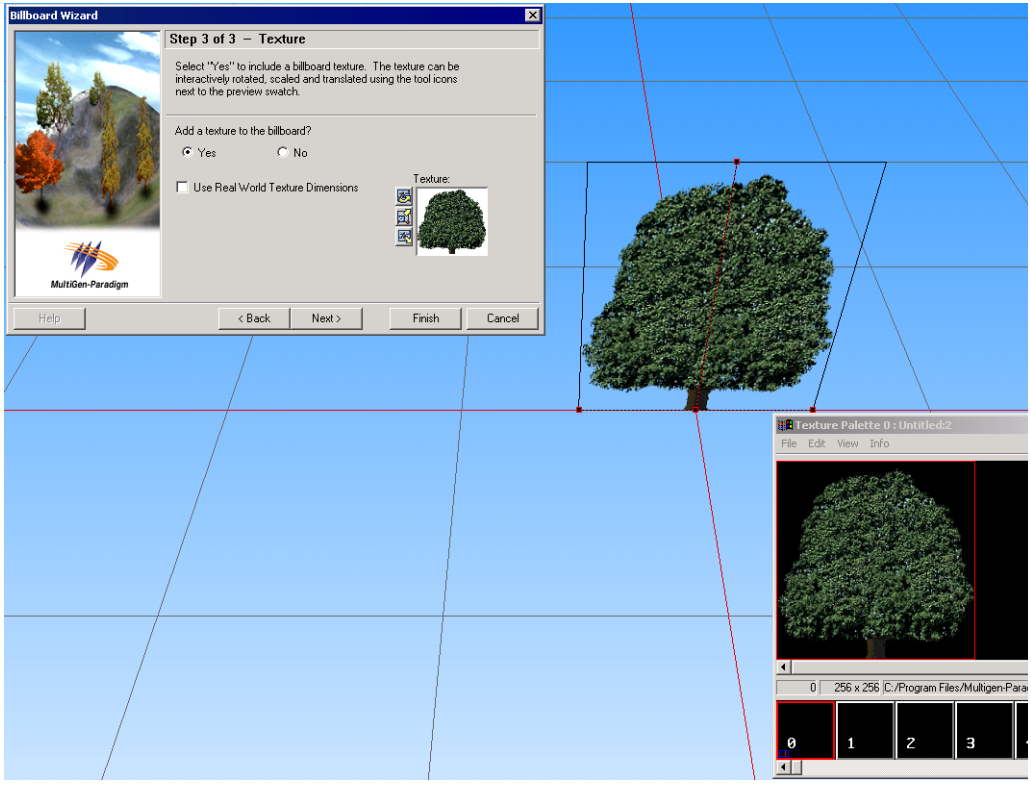
Note: Throughout the wizard, anytime you choose to enter values directly within a box, you can apply the values immediately (and see the results) by using the Tab key on your keyboard. Do not use the Enter key, or you will close the wizard.

Remember, you are free at this point to experiment with different settings, moving back to previous steps, making adjustments, if necessary.

Step 3. Click **Next**, to move to "Step 3 – Texture."

Step 3 – Texture

Step 1. Choose **Yes** to add the selected texture to your model.



Once you have placed the texture onto the model, additional controls appear on the Wizard, allowing you to manipulate the image.

Note: If you have launched the Billboard Wizard without previously opening the *Texture Palette*, you will see the following message: "You must first read in a pattern from the texture palette to proceed to the next step. Would you like to open the texture palette now?" Choose **Yes**, and load your texture image according to the process described in "Preparing to Use the Billboard Wizard" on page 1293. Additionally, if you close the *Texture Palette* (perhaps to provide more workspace on your monitor), you can open it by clicking on the **Texture** square on the right side of the Billboard Wizard.

After you have added a texture to your model, three buttons are active to the left of the texture image on the wizard.



The top button is the **Rotate** button; each mouse-click on this button rotates the image 90° clockwise. If you make a mistake, simply click the image in the *Texture Palette* to return the tree to its original position



The middle button is the **Scale** button. When this button is pressed, you can reduce or enlarge the texture image by dragging the cursor over the image as it appears on the Tree Wizard, while holding down your left mouse button; dragging the cursor down enlarges the image, while dragging it up reduces the image. The changes appear interactively on the model as you drag your cursor within the Wizard. (Try it!) Click the image in the *Texture Palette* to return the tree to its original size (and position).



The bottom button is the **Pan** button. When this button is pressed, you can move the position of the texture on the model interactively, by dragging your cursor over the texture on the wizard, while holding down your left mouse button. If you make a mistake, simply click the image in the *Texture Palette* to return the tree to its original position (and size).

Note: If the **Scale** and **Pan** buttons are unavailable, a red "X" appears on top of the buttons. In this case, you need to check your default settings using the *Texture Palette.* Double-click the image in the *Texture Palette* to display a *Texture Attributes* window. In the **Wrap Method** section of the window, change any settings of **Clamp** to **Repeat**. Close the the window and return to the Tree Wizard; the **Scale** and **Pan** buttons should now be available.

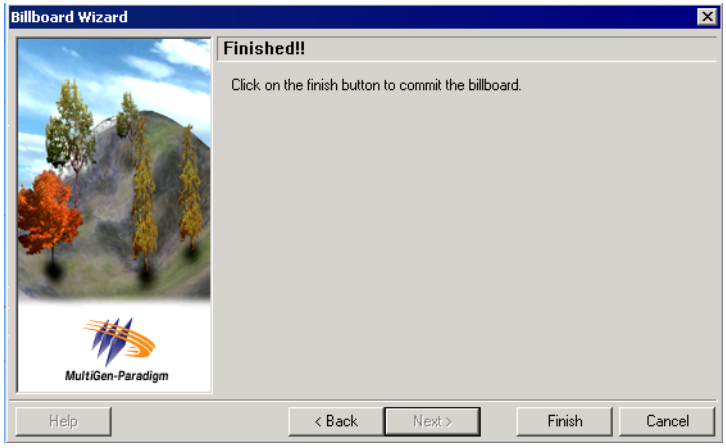
Step 2. Leave **Real World Texture Dimensions** unchecked.

Should your model require exact dimensions for accuracy, you can choose **Real World Texture Dimensions.** For example, if you were using the texture image of a tree based upon a photograph, and you knew the precise height of that tree, you might want to enter that value using **Real World Texture Dimensions**. The **Width** and **Height** here are based upon your default preferences. To change your preferences, go to the top of the Creator window, choose **Info/Preferences**, and choose the **Textures** tab. You can

then change the preferences under **Real World Size**. Again, for this tutorial, leave this option on the Tree Wizard unchecked.

Step 3. Click **Next**, to move to the final panel of the Billboard Wizard.

At this point, you are free to inspect your tree from various angles, and move back to any step necessary to make adjustments. When you are satisfied with your work, choose **Finish** on the final panel. If you choose **Cancel**, the wizard will vanish, along with your partially-constructed tree.



Note: Billboard rotation occurs during runtime, and is not displayed in Creator.

Tree Wizard Tutorial

When constructing an architectural scene, you may want to add trees or other vegetation. Unfortunately, detailed models of trees and plants take an enormous amount of time to construct, and they tax most desktop systems during runtime.

A simpler solution is to place a picture of a tree – that is, a texture of a tree – onto a group of cross-hatched, transparent polygons. When viewed from ordinary perspectives in a scene, such tree models appear realistic enough, while using only a small amount of system resources.

The Creator Tree Wizard, which is located in the *Geometry* toolbox, enables you to construct tree models quickly, using cross-hatched polygons.

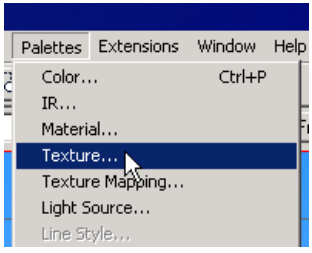
(Another solution is to map a tree texture to a *billboard* – a single polygon which rotates within the simulation, so that the image of the tree always faces the eyepoint. This method is employed in the "Billboard Wizard Tutorial" on page 1293.)

Before using the Tree Wizard, you should open the *Texture Palette* and select textures for making your tree model. This process is described in the next section, "Preparing to Use the Tree Wizard."

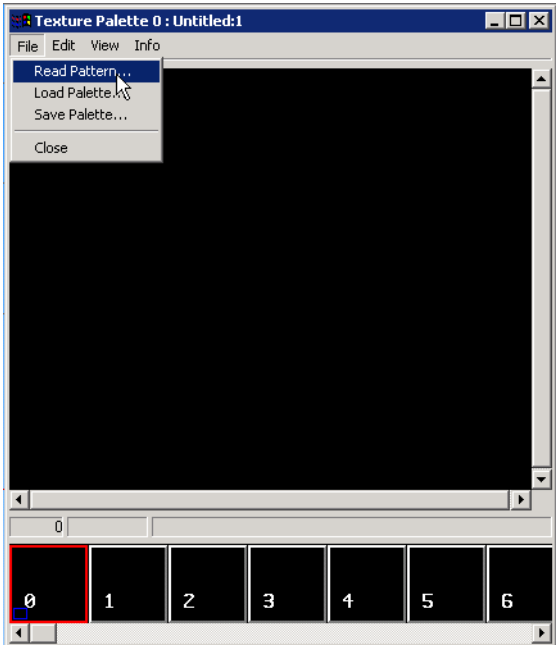
Preparing to Use the Tree Wizard

Before opening the Tree Wizard, you should select an image of a tree to use for your model. To do this, use the *Texture Palette.*

Step 1. In the Creator window, choose **Palettes/Texture** to display the *Texture Palette.*

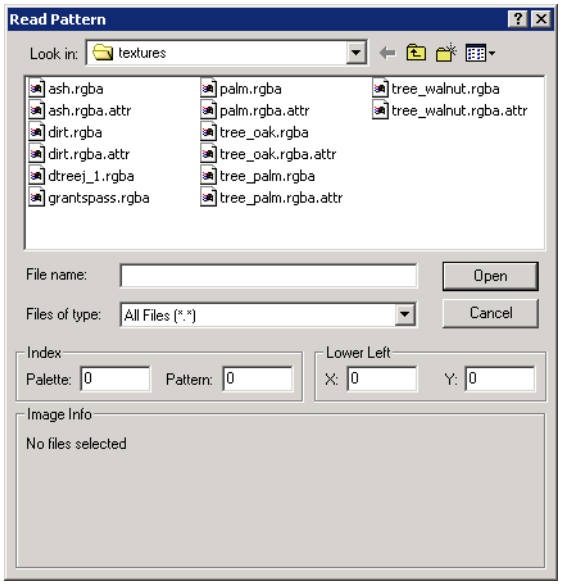


Step 2. In the *Texture Palette* window, choose **File/Read Pattern** to display the *Read Pattern* window.

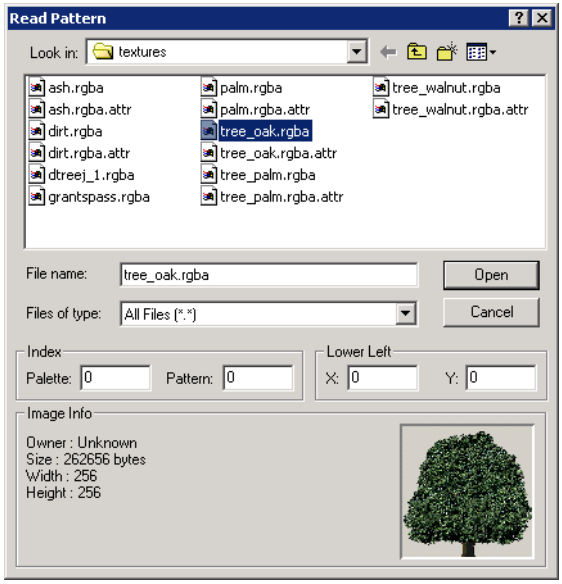


Step 3. In the *Read Pattern* window, navigate to the texture directory `MultiGen-Paradigm\resources\tutorials\creator\wizards\textures` on Windows, or `/usr/local/MultiGen-Paradigm/resources/tutorials/`

`creator/wizards/textures` on IRIX. (In the **Files of type** list, choose **All Files (*)**, if this is not your default setting already.)



Step 4. Choose `tree_oak.rgba`, and click **Open** to load the oak tree texture into the *Texture Palette.*

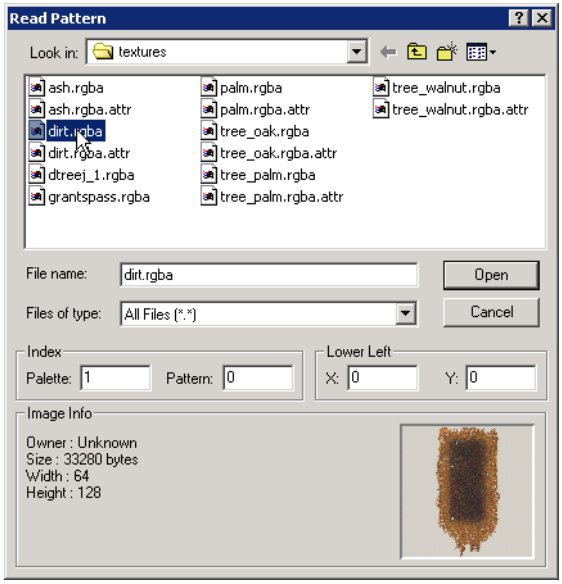


Step 5. Return to the *Texture Palette,* and select the next bay (number **1** at the bottom of the palette).



Step 6. Choose **File/Read Pattern** again to display the *Read Pattern* window.

Step 7. Select `dirt.rgba`, and click **Open** to load a shadowed dirt texture into the *Texture Palette.*



If you want to experiment with other image(s), you can also load additional textures, by selecting the next numbered box(es) in the *Texture Palette*, and choosing **File/Read Pattern...** for the additional image(s).

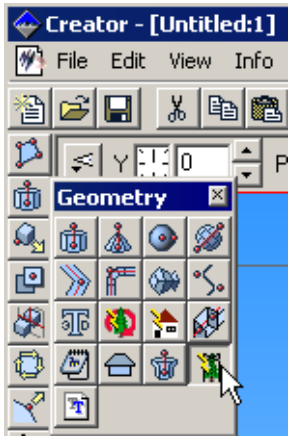
If you prefer to select different images later in the process, you can do so. But we have opened the *Texture Palette* and selected images now, so that you can complete perform the steps in the Tree Wizard without interruption.

You are now ready to proceed to the next section, “Using the Tree Wizard.”

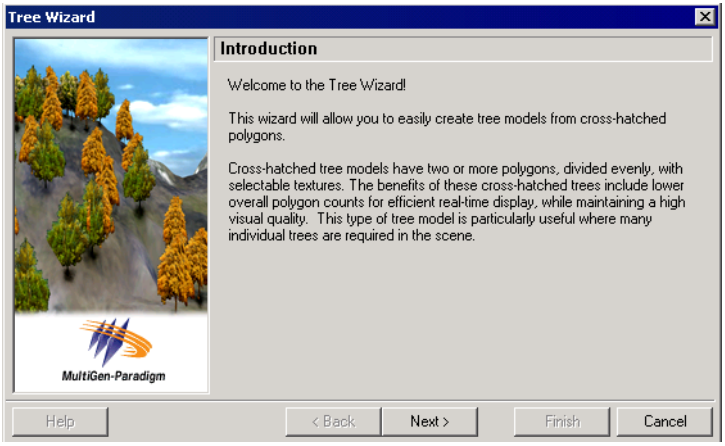
Using the Tree Wizard

To open the Tree Wizard:

Step 1. In the *Geometry* toolbox, choose the Tree Wizard button.



The first panel of the Tree Wizard is displayed on your screen.



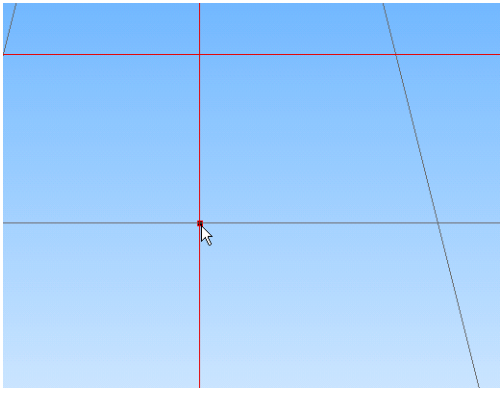
Step 2. Click **Next** to proceed to the first step for creating your tree.

Step 1 – Orientation

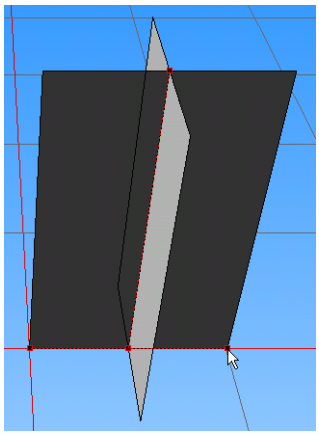
The orientation of your tree is determined by the origin and alignment you choose in this step.



Step 1. In the Creator window, move your cursor on the grid to the desired point of origin, and click once.

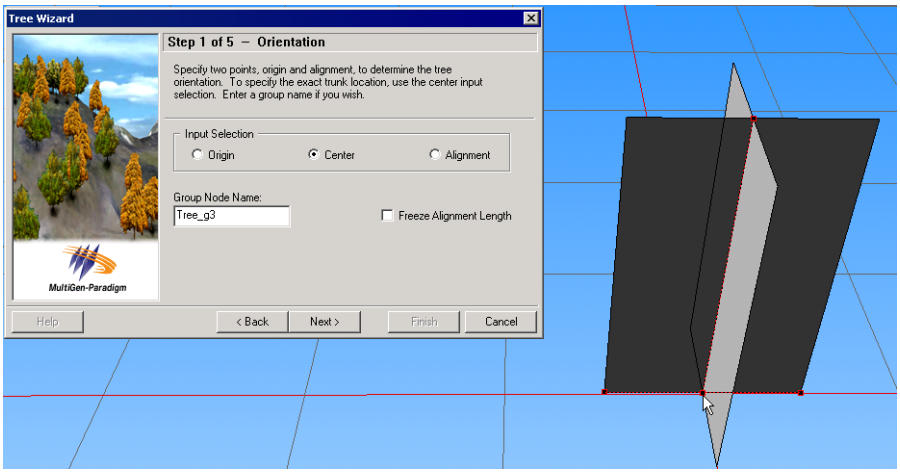


Step 2. Click on a second point to establish the desired alignment point.



Step 3. Click the **Center** option under **Input Selection** on the wizard. By choosing this option, you are centering the base of your polygons onto a point you select.

Step 4. Click the desired center-point on the grid to position your model.



Step 5. In **Group Node Name**, enter a unique file name; for this tutorial, use "oaktree1". Please keep in mind that the default name, such as "Tree_g3" is unique; when you choose another name, uniqueness in not guaranteed. You need to ensure that you do not give the same name to two different groups.

Step 6. Leave the **Freeze Alignment Length** box unchecked for this tutorial

Note: If you check the option for **Freeze Alignment Length**, your model will be tethered when the Alignment option is chosen; you can change the alignment point to reposition your model, but its length will not change. By leaving this option unchecked, the length of the model can be stretched by pulling on the alignment point.

Step 7. Click **Next** to go to "Step 2 – Shape, Height, and Divisions."

Note: At any time while using the Wizard, you are free to go back or move forward, making additional adjustments, by clicking the **Back** and **Next** buttons at the bottom of the pane.

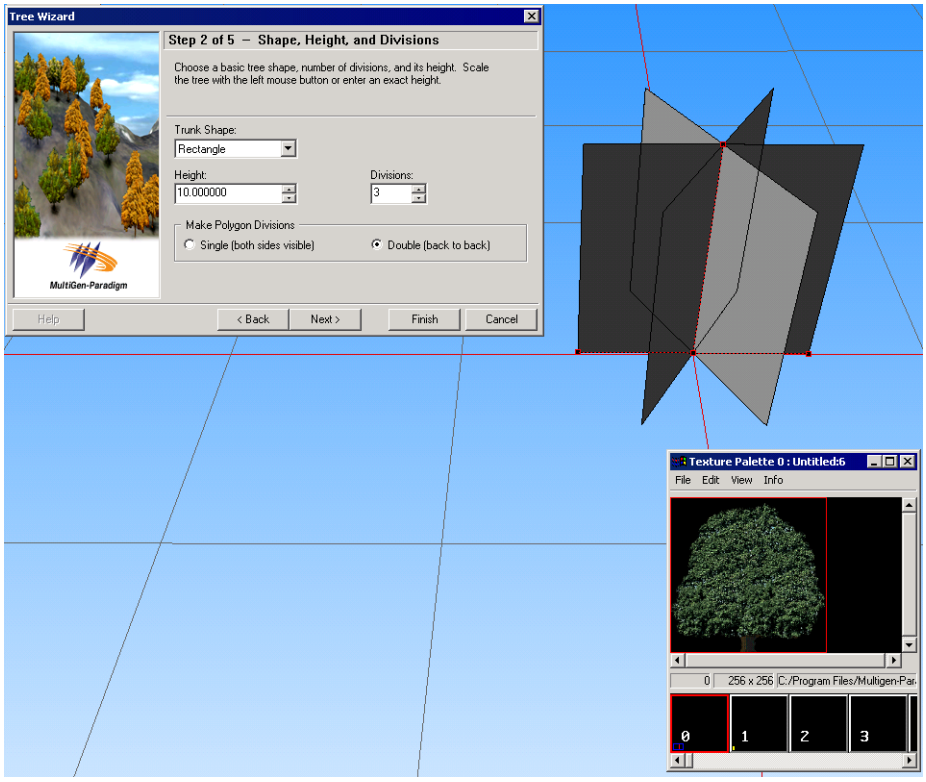
Step 2 – Shape, Height, and Divisions

Step 1. Choose **Rectangle** for the **Trunk Shape**. (This should be the default value, unless you have previously used the Tree Wizard and changed this selection.)

Step 2. Enter **10** (10.000000) for the **Height** for your tree model, or adjust the height with your mouse by clicking and dragging on the top point of the model. The **Height** is measured in database units; for this tutorial, the units are meters.

Note: Throughout the wizard, anytime you choose to enter values directly within a box, you can apply the values immediately (and see the results) by using the Tab key on your keyboard. Do not use the Enter key, or you will close the wizard.

Step 3. Choose **3** divisions for your model. The more divisions you use on a cross-hatch model, the more realistic the tree will appear; but additional divisions also require more system resources to run.



Remember, you are free at this point to experiment with different settings, moving back to previous steps, making adjustments, if necessary.

Step 4. Under **Make Polygon Divisions**, the default setting should be **Double (back to back)**. If you are using SiteBuilder or Vega with your model, make certain that **Double (back to back)** is selected here, or your model will not work properly.

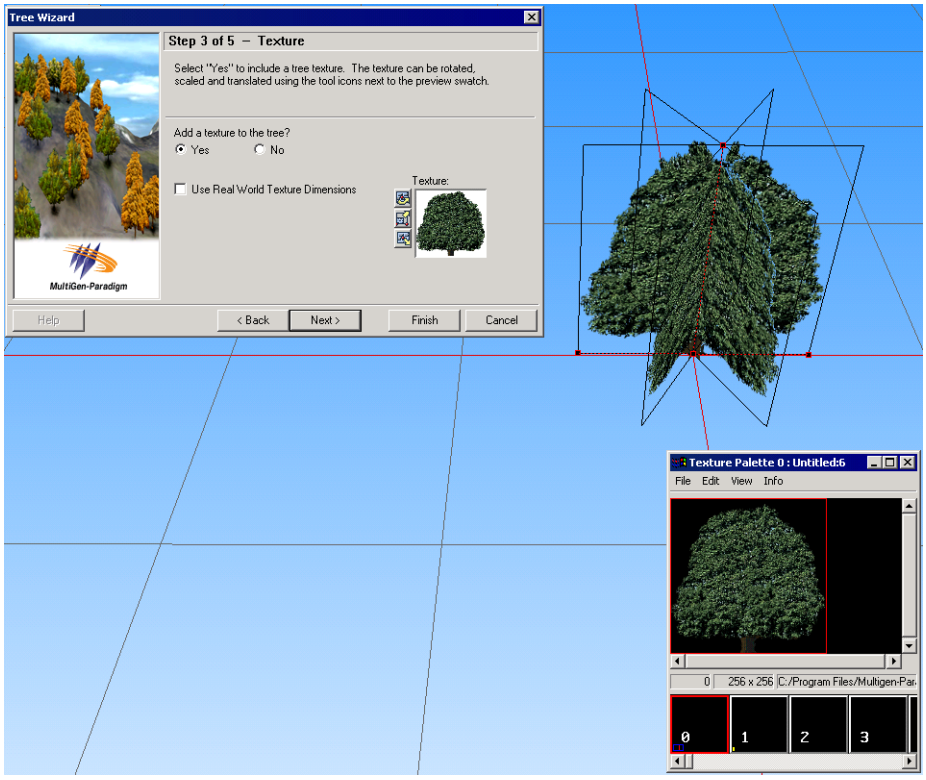
Step 5. Click **Next**, to move to "Step 3 – Texture."

Step 3 – Texture

You are now ready to add texture to your model.

Step 1. Click once on the tree image in the *Texture Palette*.

Step 2. Choose **Yes** to add the selected texture to your model.



Once you have placed the texture onto the model, additional controls appear on the Wizard, allowing you to manipulate the image.

If you have launched the Tree Wizard without previously opening the *Texture Palette*, you will see the following message: "You must first read in a pattern from the texture palette to proceed to the next step. Would you like to open the texture palette now?" Choose **Yes**, and load your texture images according to the process described in "Preparing to Use the Tree Wizard" on page 1305. Additionally, if you close the Texture Palette (perhaps

to provide more workspace on your monitor), you can open it by clicking on the **Texture** square on the right side of the Tree Wizard.

After you have added a texture to your model, three buttons are active to the left of the texture image on the wizard.



The top button is the **Rotate** button; each mouse-click on this button rotates the image 90° clockwise. If you make a mistake, simply click the image in the *Texture Palette* to return the tree to its original position



The middle button is the **Scale** button. When this button is pressed, you can reduce or enlarge the texture image by dragging the cursor over the image as it appears on the Tree Wizard, while holding down your left mouse button; dragging the cursor down enlarges the image, while dragging it up reduces the image. The changes appear interactively on the model as you drag your cursor within the Wizard. (Try it!) Click the image in the *Texture Palette* to return the tree to its original size (and position).



The bottom button is the **Pan** button. When this button is pressed, you can move the position of the texture on the model interactively, by dragging your cursor over the texture on the wizard, while holding down your left mouse button. If you make a mistake, simply click the image in the *Texture Palette* to return the tree to its original position (and size).

Note: If the **Scale** and **Pan** buttons are unavailable, a red "X" appears on top of the buttons. In this case, you need to check your default settings using the *Texture Palette.* Double-click the image in the *Texture Palette* to display a *Texture Attributes* window. In the **Wrap Method** section of the window, change any settings of **Clamp** to **Repeat**. Close the the window and return to the Tree Wizard; the **Scale** and **Pan** buttons should now be available.

Step 3. Leave **Real World Texture Dimensions** unchecked.

Should your model require exact dimensions for accuracy, you can choose **Real World Texture Dimensions.** For example, if you were using the texture image of a tree based upon a photograph, and you knew the precise height of that tree, you might want to enter that value using **Real World**

Texture Dimensions. The **Width** and **Height** here are based upon your default preferences. To change your preferences, go to the top of the Creator window, choose **Info/Preferences**, and choose the **Textures** tab. You can then change the preferences under **Real World Size**. Again, for this tutorial, leave this option on the Tree Wizard unchecked.

Note: You can use your right mouse button to manuever around your tree, and view its appearance from different angles.

Click **Next**, to move to "Step 4 – Tree Shadow."

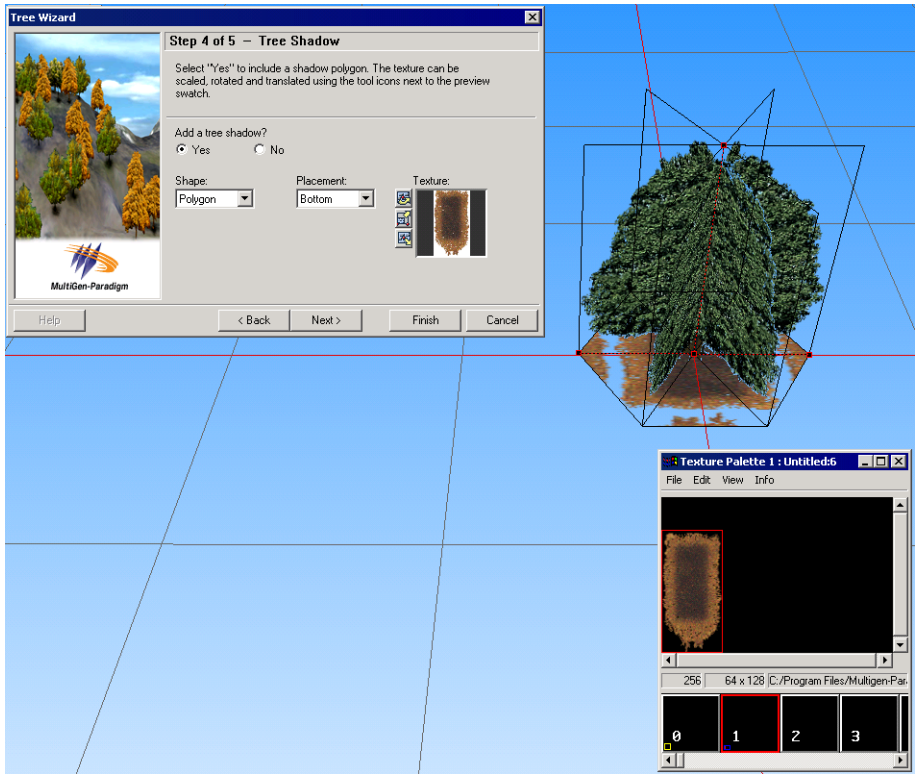
Step 4 – Tree Shadow

For added realism, place a shadow beneath the tree.

Step 1. On the *Texture Palette*, click the second texture, `dirt`.

Step 2. Click **Yes** to add this texture as a shadow pologon. This choice displays an additional option for **Shape**, as well as the **Rotate**, **Scale**, and **Pan** buttons described in the previous step.

Step 3. Choose **Polygon** for the **Shape** of your shadow.

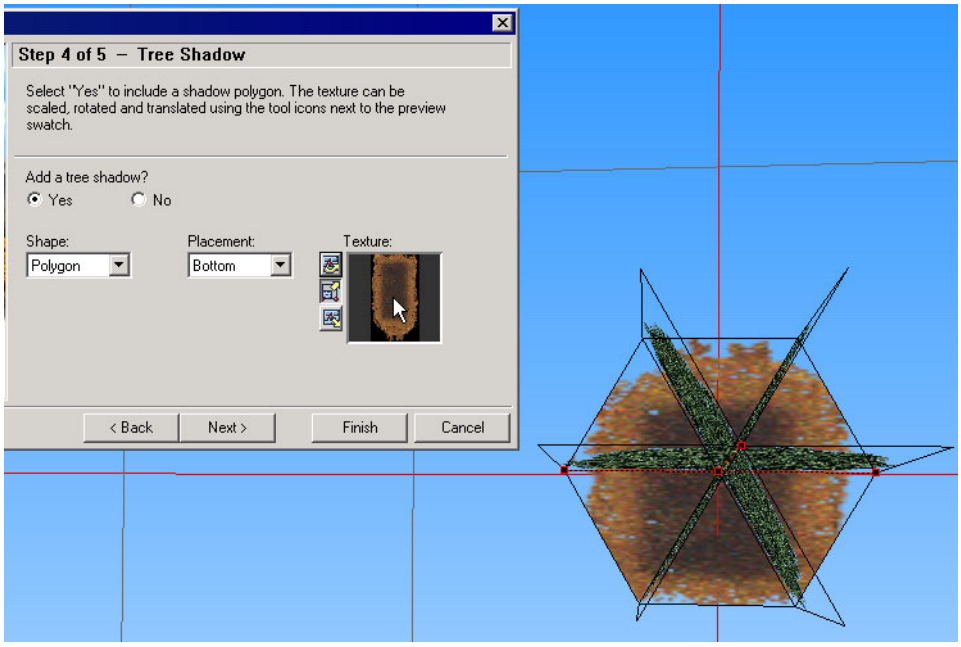


We have chosen the polygon option to make the shadow more realistic, but the image needs to be modified.

Step 4. Using your right mouse button, position your view of the model, so that you can see the shadow polygon more clearly from the top.

Step 5. Click the **Scale** button (the middle button) next to the texture image on the wizard.

Step 6. Holding down your left mouse button, drag your cursor within the texture image on the wizard to enlarge the image, pushing it out toward the edges of the polygon.



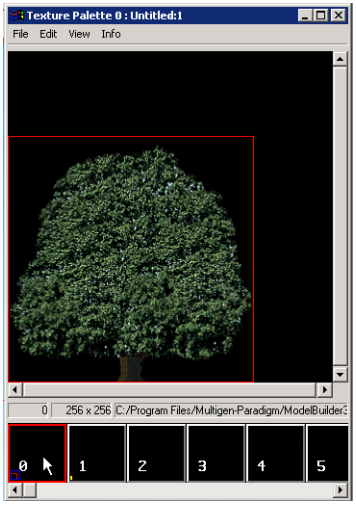
When you have finished manipulating the shadow polygon, proceed to the next step.

Step 7. Click **Next**, to move to "Step 5 – Tree Crown/Top-down View".

Step 5 – Tree Crown/Top-down View

While setting up the shadow polygon, you might have noticed that your tree looked good, except when viewed from the top. If you anticipate that your model will be seen from the top down (for example, in a flight simulation), you might want to add a crown polygon. Using switches, the crown can be made visible during runtime when the tree is viewed from above; the cross-hatches will be visible when the tree is viewed from the sides.

Step 1. Click on the oak tree image on the *Texture Palette* (the first bay, **0**). You are using the same texture to make the crown that you used for the body of the tree

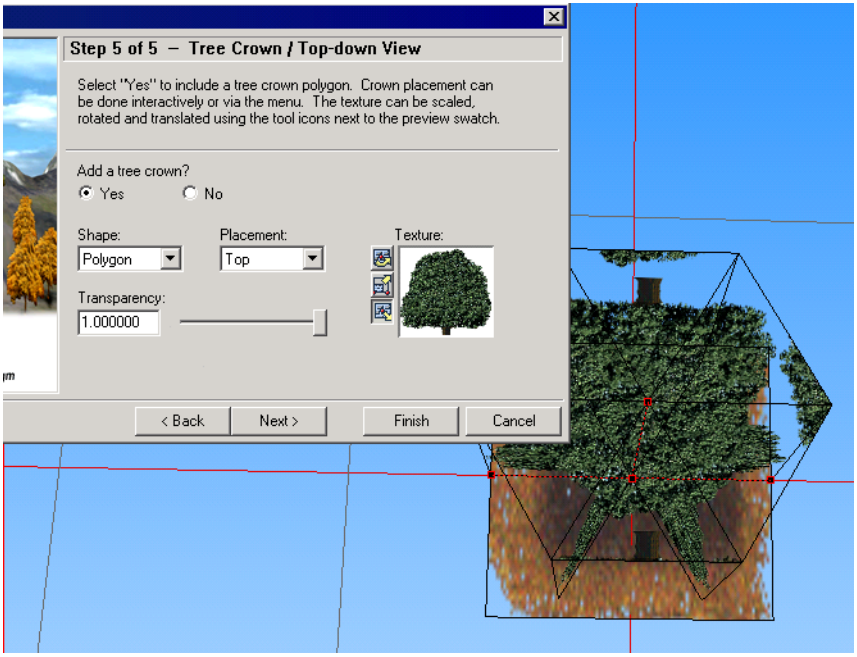


Step 2. On the Tree Wizard, choose **Yes** under **Add a tree crown?**.

Step 3. Choose **Polygon** for **Shape**. (Because you are using three cross-hatches in this tutorial, a polygon provides a more realistic crown.)

Step 4. Choose **Top** for **Placement**. (This should be the default, unless you have previously used the Wizard.)

Step 5. Leave **Transparency** at **1 (1.000000)**. With a value of **1**, the crown is completely opaque; with a value of **0**, it becomes completely transparent; you can adjust the value, if you desire partial transparency.

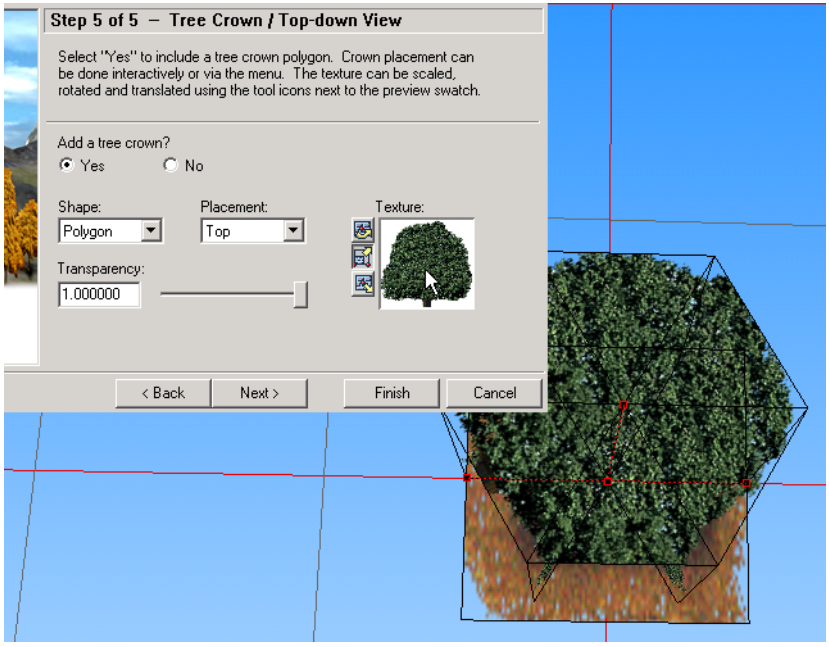


At this point you need to manipulate the image, much as you did for the tree shadow in the previous step.

Step 6. Click the **Scale** button (the middle button) next to the texture image on the wizard.

Step 7. Holding down your left mouse button, drag your cursor within the texture image on the wizard to enlarge the image, pushing it out toward the edges

of the polygon. The idea is to fill the crown with tree leaves, making the trunk no longer visible.

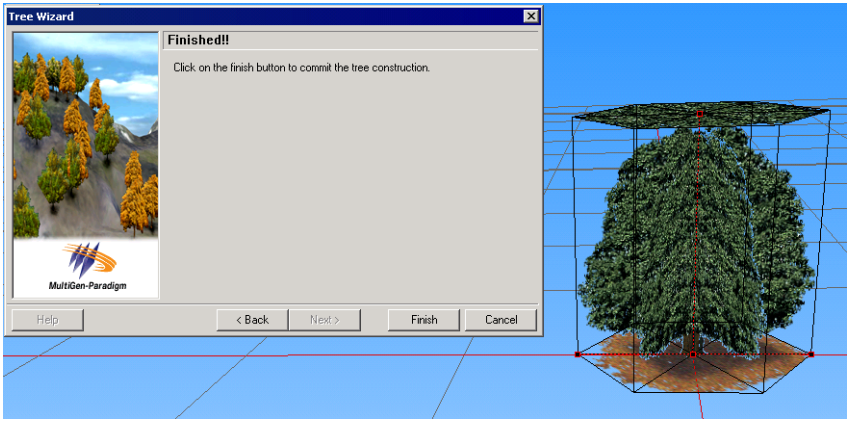


When you have finished manipulating the crown polygon, proceed to the next step.

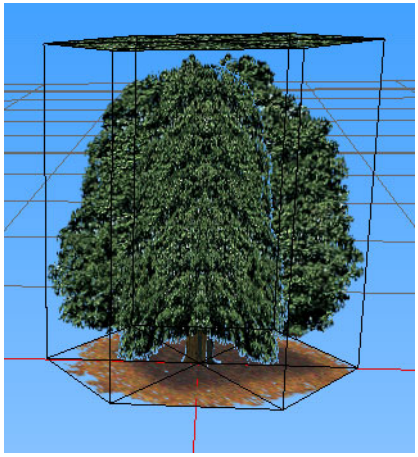
Step 8. Click **Next**, to move to the final panel.

At this point, you are free to inspect your tree from various angles, and move back to any step necessary to make adjustments. When you are satis-

fied with your work, choose **Finish** on the final panel. If you choose **Cancel**, the wizard will close, and your partially-constructed tree will vanish.



After you have chosen **Finish** to commit the tree construction, it is a good idea to save your Creator file. Then you can sit back, admire your tree, and consider how much time you have saved by using the wizard.



Building Wizard Tutorial

With the Building Wizard, you can create models of houses or light industrial buildings with varying types of roofs. The models created with this wizard can be used individually, or as a point substitution in your realtime 3D environment.

This tutorial illustrates some of the basic features of the Building Wizard. Nevertheless, because there are so many possible variations within the within the wizard, take some time to explore additional options on your own.

Before using the Building Wizard, you should select textures for making your model. Also, you will need a footprint for your building. These preliminary matters are covered in the next section, "Preparing to Use the Building Wizard."

Preparing to Use the Building Wizard

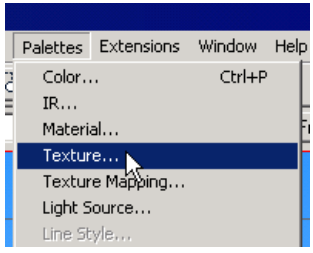
Before opening the Building Wizard, there are two tasks to perform to make your use of the wizard run more smoothly:

- Load textures into the *Texture Palette.*
- Create a footprint for your builing model.

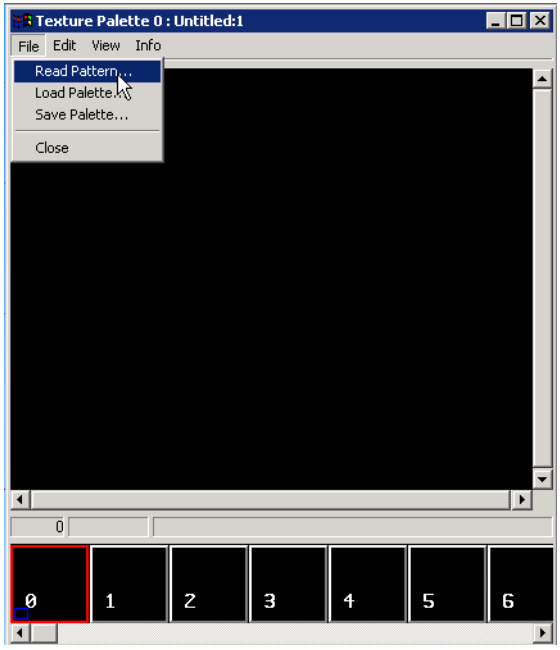
Loading Textures into the Texure Palette

Before opening the Building Wizard, select some textures to use with your model. To do this, use the *Texture Palette.*

Step 1. In the Creator window, choose **Palettes/Texture** to display the *Texture Palette.*

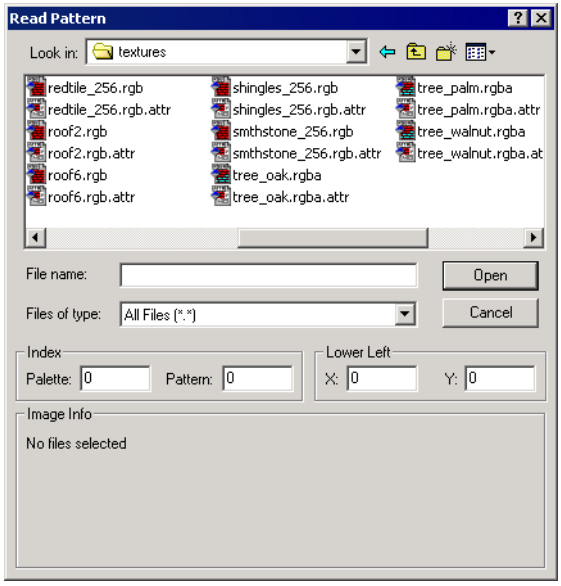


Step 2. In the *Texture Palette* window, choose **File/Read Pattern** to display the *Read Pattern* window.



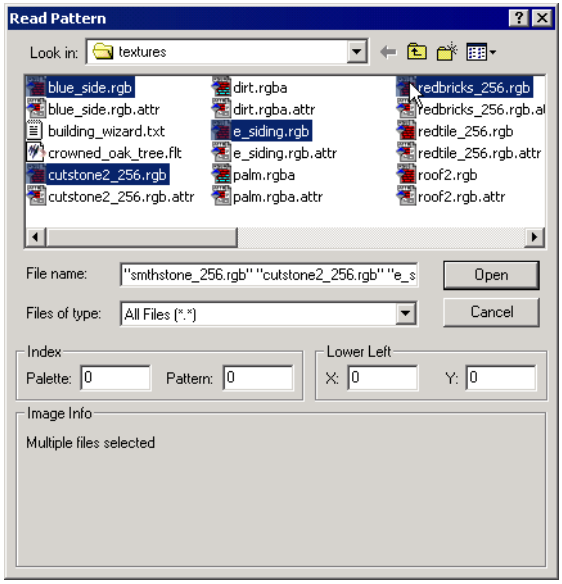
Step 3. In the *Read Pattern* window, navigate to the texture directory `MultiGen-Paradigm\resources\tutorials\creator\wizards\textures` on Windows, or `/usr/local/MultiGen-Paradigm/resources/tutorials/`

`creator/wizards/textures` on IRIX. (In the **Files of type** list, choose **All files (*)**, if this is not your default setting already.)

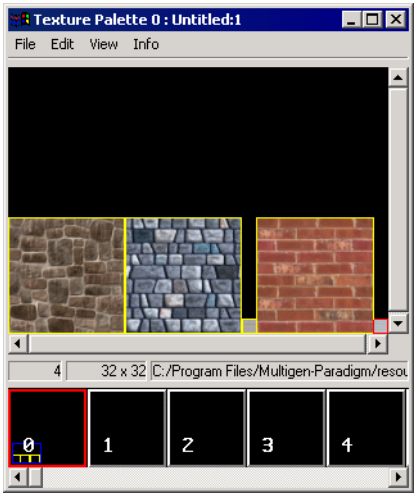


Step 4. Holding down the Ctrl key on your keyboard, use your cursor to choose the following files: `blue_side.rgb`, `cutstone2_256.rgb`, `e_siding.rgb`,

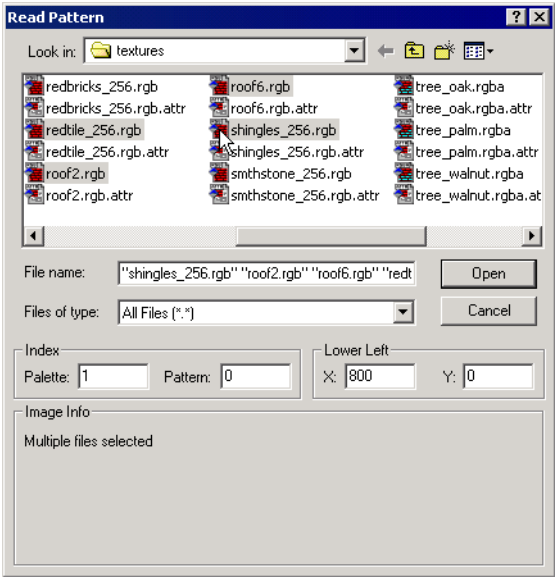
`redbricks.rgb` and `smthstone_256.rgb`; click **Open** to load theses textures into the *Texture Palette.*



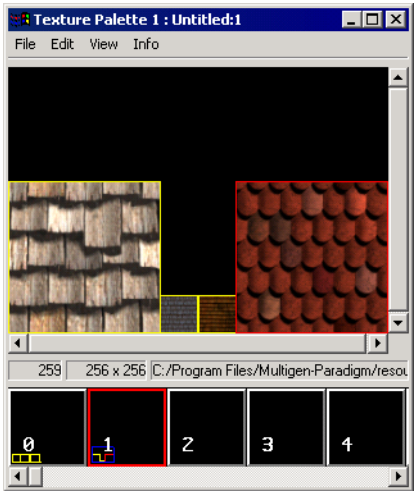
Your texture palette now has an assortment of textures from which to choose for building materials.



Click on bay **1** in the *Texture Palette* window. Repeat the loading process and read the following texture patterns into this bay: `redtile_256.rgb`, `roof2.rgb`, `roof6.rgb`, and `shingles_256.rgb`.



Your texture palette now has an assortment of textures from which to choose for roofing materials.



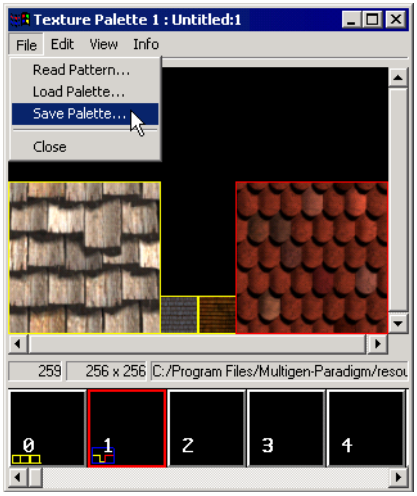
You won't necessarily use all of these textures in your current model, yet they provide options, so that you can experiment with the wizard. Typical-

ly, if you are working on a building model, you will find it useful to group various types of building materials into several bays, such as the wall textures in one bay and the roof textures in another bay.

If you prefer to select different textures later in the process, you can do so. But we have opened the *Texture Palette* and selected images now, so that you can complete the steps in the Building Wizard without interruption.

Additionally, since you have gone to the trouble to load numerous textures, it is a good idea to save the palette, in case you wish to use these same groups of textures later. Should you need to quit the application in the midst of a project, you can reload the palette and resume your work, without having to load all of the textures individually.

Step 5. To save the *Texture Palette* now, choose **File/Save Palette** from the *Texture Palette* menu.



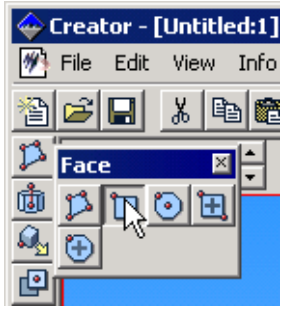
Since you are using this palette with the Building Wizard, name the file "building_wizard" when saving it. After you have named the file, you can resize or minimize the *Texture Palette*, in order to have plenty of workspace in the main Creator window.

You are now ready to proceed to the next section, "Make a Footprint."

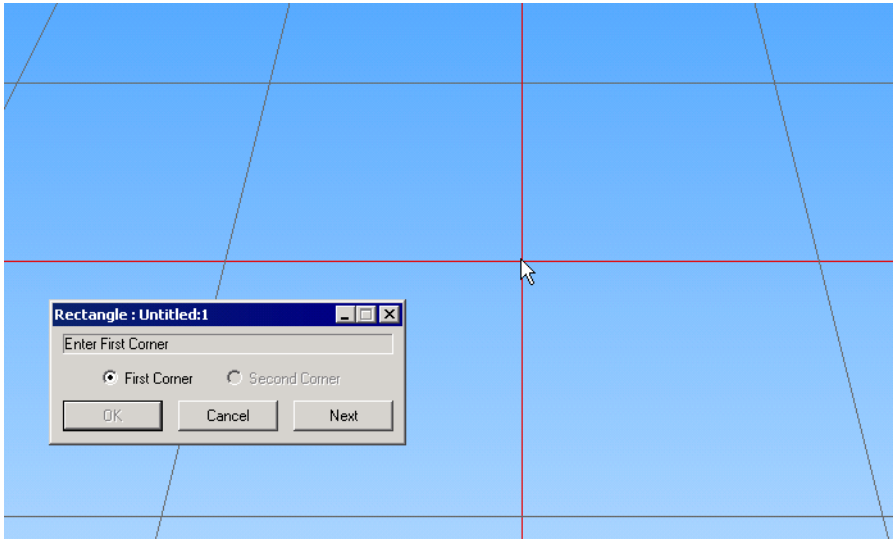
Make a Footprint

Before opening the Building Wizard, you must first select a footprint in the Face mode. You may already have a footprint which you wish to use with the wizard. Yet, for this tutorial, draw a rectangle to serve as the footprint of the building.

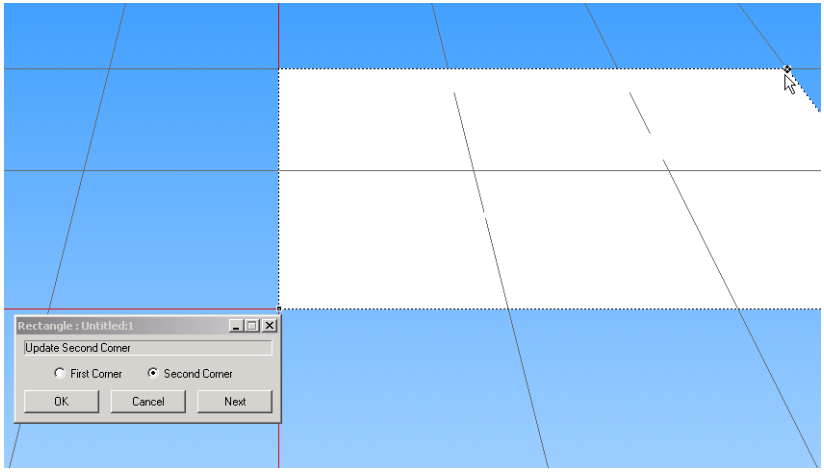
Step 1. In the Face toolbox, choose the rectangle.



Step 2. In the Creator window, move your cursor to the desired point for the first corner and click once.



Step 3. Next, move your cursor to the desired point for the second corner and click again.



Step 4. Click **OK** to close the **Rectangle** dialogue box.

Step 5. Make sure the **Face** mode is selected in the Mode list (Shift+F is the keyboard shortcut to select the face mode).

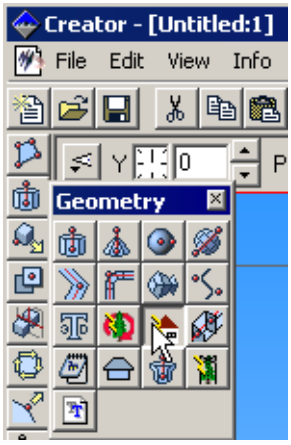


With your rectangular face (footprint) selected, you are now ready to proceed to the next section, “Running the Building Wizard.”

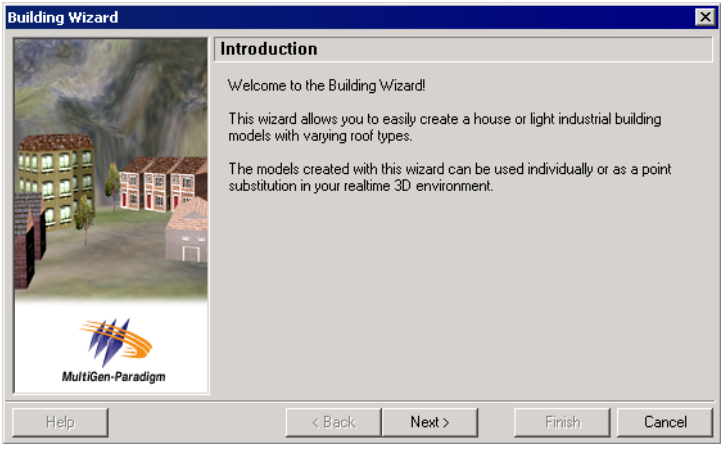
Running the Building Wizard

To open the Building Wizard:

Step 1. In the *Geometry* toolbox, choose the Building Wizard button.



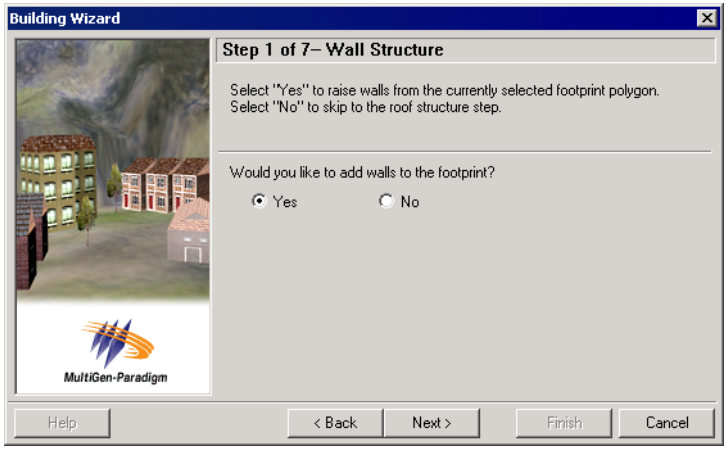
The first panel of the Billboard Wizard is displayed on your screen.



Step 2. Click **Next** to proceed to the first step for creating your building.

Step 1 – Wall Structure

This step is used to raise the walls for your building.

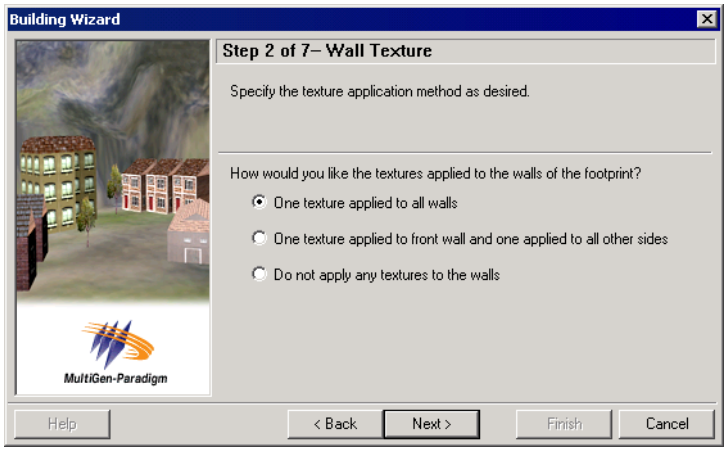


Step 1. Since you are beginning a new model of a building, choose **Yes** (the default setting) on this panel. (If you already had the walls for your building, and wished merely to add a roof, you would could choose **No**, and then skip the steps pertaining to walls.)

Step 2. Click **Next**, to move to "Step 2 – Wall Texture."

Step 2 – Wall Texture

The choices you make here determine which textures are applied to various parts of your building.



Step 1. The building for this tutorial is simple, so choose **One texture applied to all walls**. (This should be the default setting.)

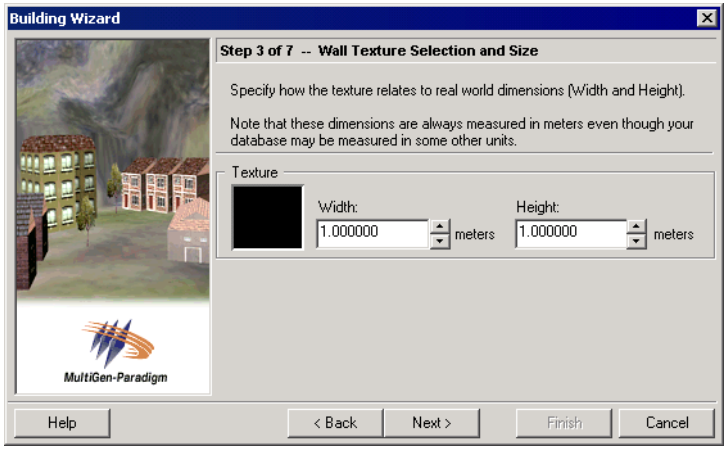
If you had a unique texture for the front of your building, such as a texture with doors and/or windows, you would select the second option, **One texture applied to front wall and one applied to all other sides**.

If you were building a model without using textures on the walls, you would, of course, select the third option, **Do not apply any textures to the walls**.

Step 2. Click **Next**, to move to "Step 3 – Wall Texture Selection and Size."

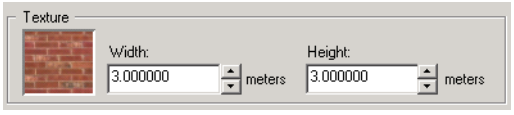
Step 3 – Wall Texture Selection and Size

You must now choose the texture to be applied to the walls of your building.



Step 1. In the *Texture Palette*, click on the `redbricks_256.rgb` texture in the first bay. This is the basic texture for your walls.

Step 2. Change **Width** to "3" (3.000000) and **Height** to "3" (3.000000). This change is necessary to adjust the real world dimensions of the texture.



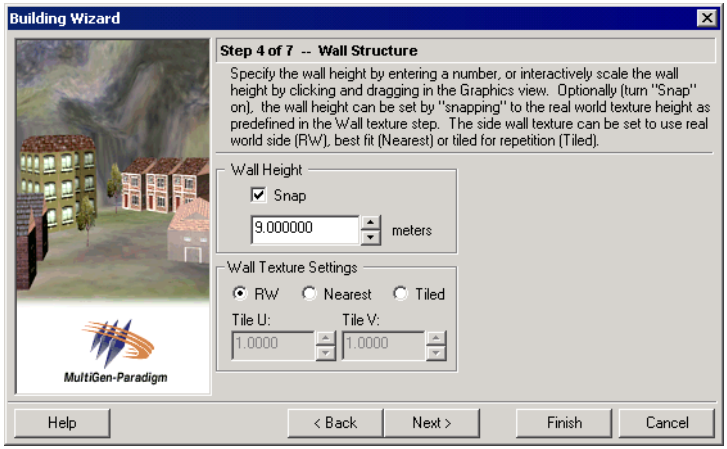
Note: The real world dimensions for **Texture Width** and **Height** are always measured in meters. If your database is in other units, Creator will convert the metrical values entered here to your default units of measurement, when you move to the next step in the wizard.

Note: Throughout the wizard, anytime you choose to enter values directly within a box, you can apply the values immediately (and see the results) by using the Tab key on your keyboard. Do not use the Enter key, or you will close the wizard.

Step 3. Click **Next**, to move to "Step 4 – Wall Structure."

Step 4 – Wall Structure

You can now set the height of the walls for your model.



Step 1. Make sure that **Snap** is set. With **Snap** set, your choices for the height of the building are in multiples of the texture height chosen in the previous step.

Step 2. Enter a value of 9 for the **Wall Height.** You can enter this value directly within the box, or by clicking on the control arrows to the right of the box.

Note: The value for **Wall Height** in this tutorial assumes database units of meters. If you are using other default units (such as feet), those units are used here for specifying **Wall Height**.

Step 3. For **Side Wall Texture Settings**, choose **RW** (real world). This selection insures that the texture will not be resized from the values chosen in the previous step.

In some cases you might not want the **RW** setting, because your texture would be mapped with fractions (partial amounts showing), or your model might possess an undesirable wrapping of the texture at the corners of the building. In such circumstances, you can choose the **Nearest** option; with this setting, the dimensions of the texture are rounded if necessary, to insure that no partial amounts of the texture are showing.

The final option, **Tiled**, specifies the way the texture is mapped to the wall. If you choose this option, the values you assign for **Tile U** and **Tile V** determine the details of the mapping.

Note: If the footprint of your building is not parallel to the tracking plane (for example, a house built on a slope), the textures still are walled in the Z direction, regardless of which texture setting you choose.

If you are making a model of an industrial building or skyscraper, you might not need a stylized roof. In such cases, you can choose **Finish** here, to skip the remaining steps in the Building Wizard.

If you are modeling a multi-story building, you might choose **Finish** at this point, in order to proceed with the next story of your building. After finishing the first story of your building here (and skipping the remaining steps of the wizard pertaining to roof construction), you then select the top face of your building, and start the Building Wizard again. The top face of your first story then serves as the footprint for the next story of the building. After completing the different stories of a multi-story building, you can then continue with the wizard to add a roof upon the top story. Regardless of whether you are modeling a single-story building, or a multi-story building, the remaining steps in the wizard enable you to put a stylized roof on the structure.

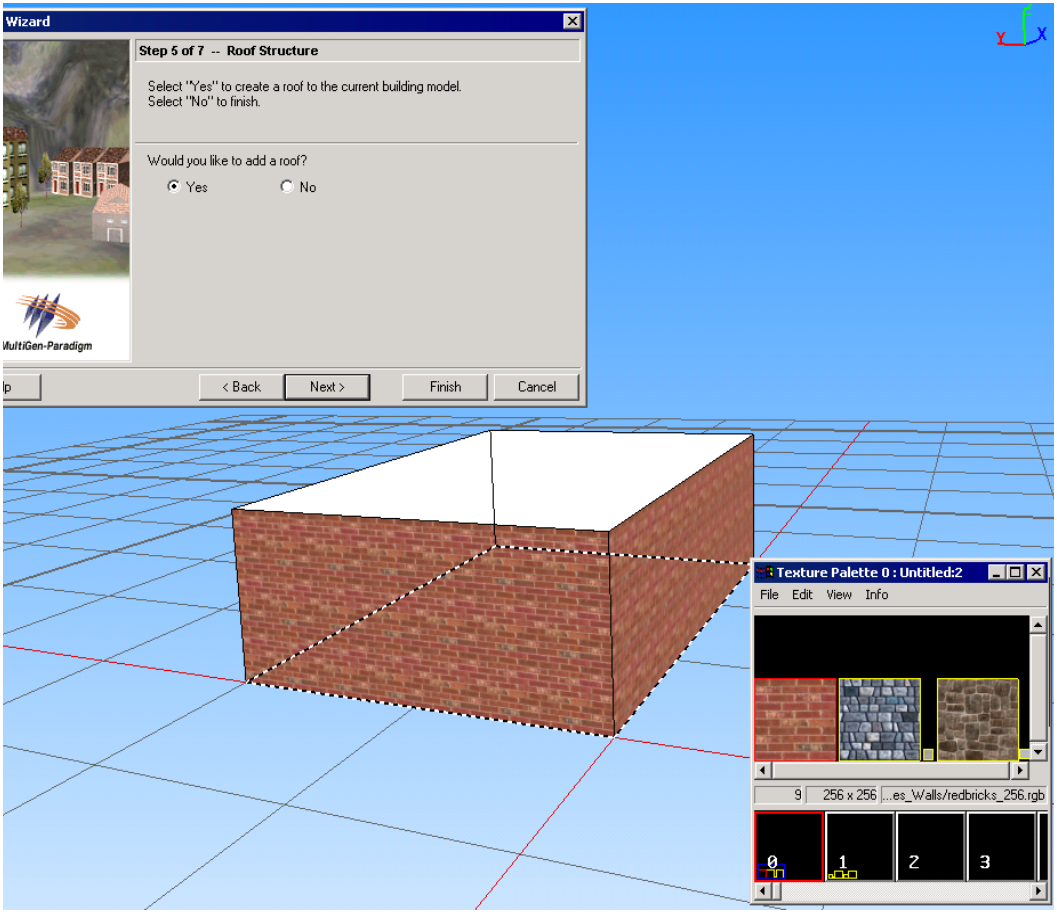
In this tutorial, we are still working with the single-story building you began in the first four steps.

Step 4. Click **Next**, to move to "Step 5 – Roof Structure."

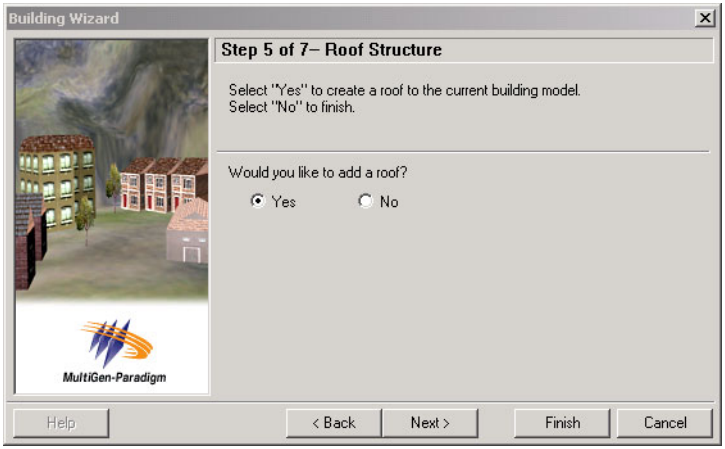
Step 5 – Roof Structure

Before adding any roof elements, take this opportunity to adjust your view of the model. Using your right mouse button, swing the model around, so that you can obtain a better

perspective. (By holding down the **Ctrl** key, while using the right mouse button, you can move the model and the tracking plane within the Creator window).



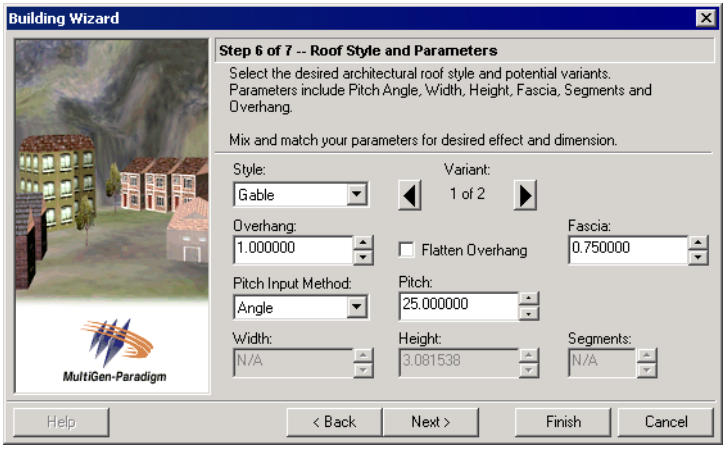
Step 1. You are now ready to begin adding roof elements. For Step 5, choose **Yes**.



Step 2. Click **Next**, to move to "Step 6 – Roof Style and Parameters."

Step 6 – Roof Style and Parameters

In this step, select a style for the roof of your building. Depending upon which style you choose, you have choices for various parameters related to that style (not all parameters apply to all styles).



Step 1. Under **Style**, choose **Gable**. The **Gable** style has two choices. Clicking the arrows to the right of **Gable** demonstrates the different **Gable** styles in the Graphics view. After examining the differences, set the choice to **1 of 2**.

Note: **Overhang**, **Fascia**, **Width** and **Height** are measured in database units; for this tutorial, the units are meters. As you enter or change the values for options on this panel, notice how they are applied to your building.

Step 2. Enter a value of **1** for **Overhang**. If you set the option for **Flatten Overhang**, the roof is raised so that the bottom of the roof is even with the top of the walls; check the box to examine the difference, then leave this box unchecked.

Step 3. For **Fascia**, enter a value of **.75**.

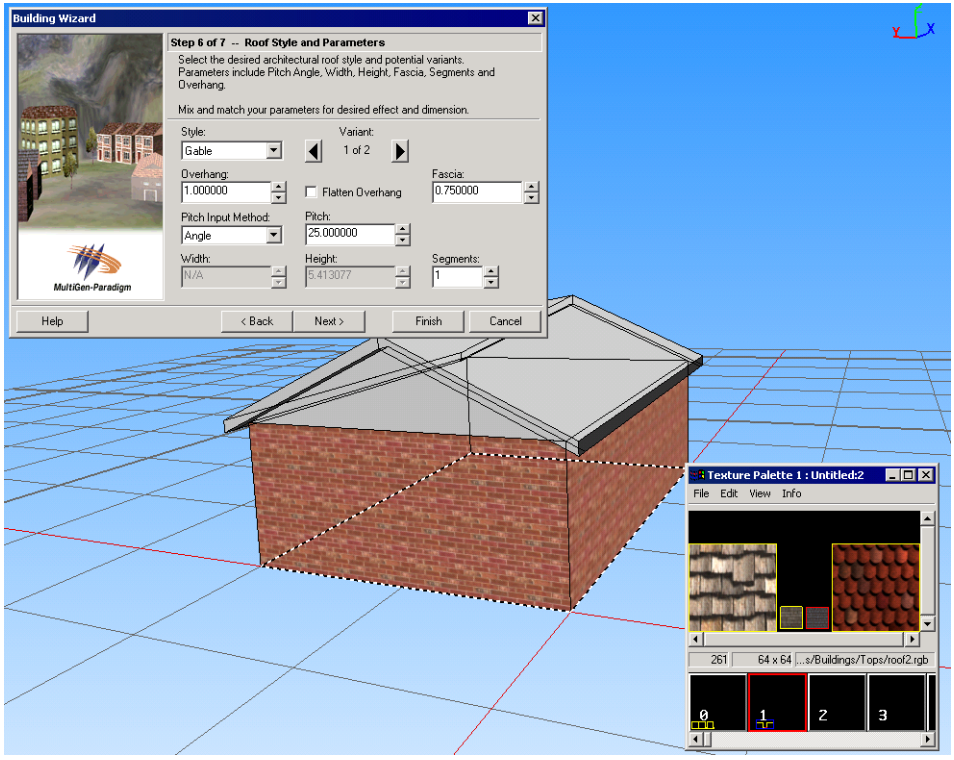
Step 4. For **Pitch Input Method**, choose **Angle**. The other choice of **1/12ths** is a convenience for users who prefer that method.

Step 5. Under **Pitch** Value, enter **25**.

For the **Gable** style, the parameters for **Width** and **Height** are not active. For other roof styles, you must enter **Width** and **Height** values, but the options for **Input Method** and **Value** will be inactive. Take a few moments to explore other options under **Style** and notice how they change the appearance of your building.

When you have finished experimenting with various roof styles, return to the **Gable** style, and make sure that the options are reset to the parameters specified above.

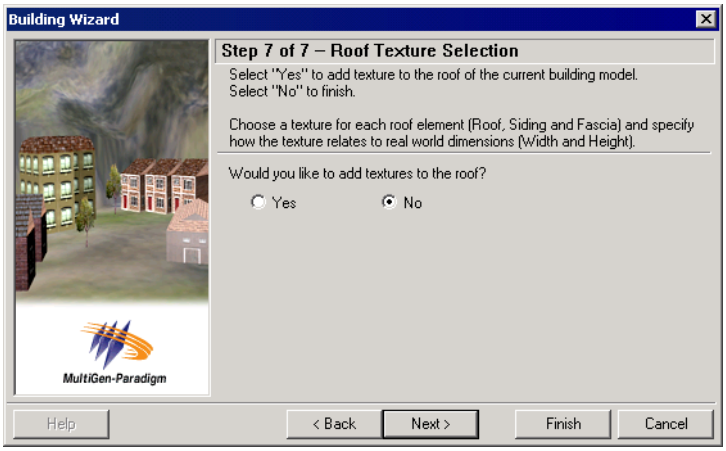
Step 6. In preparation for the next step, select the bay in the *Texture Palette* which displays the textures for the roof (this should be bay **1**).



Step 7. Click **Next**, to move to "Step 7 – Roof Texture Selection."

Step 7 – Roof Texture Selection

This step enables you to select a texture for the various elements of your roof (roof, siding, and fascia), and to specify how the texture relates to real world dimensions.



Step 1. Choose **Yes** to add textures to the roof; you are now given additional options related to each roof element.



Step 2. Select **Roof**; choose the `roof2.rgb` texture from bay **1** in the *Texture Palette* (it is the small, lighter-colored texture of a composition roof). Make sure the **Apply** box is checked; set the **Width** to 2 (2.000000) and the **Height** to **2** (2.000000).

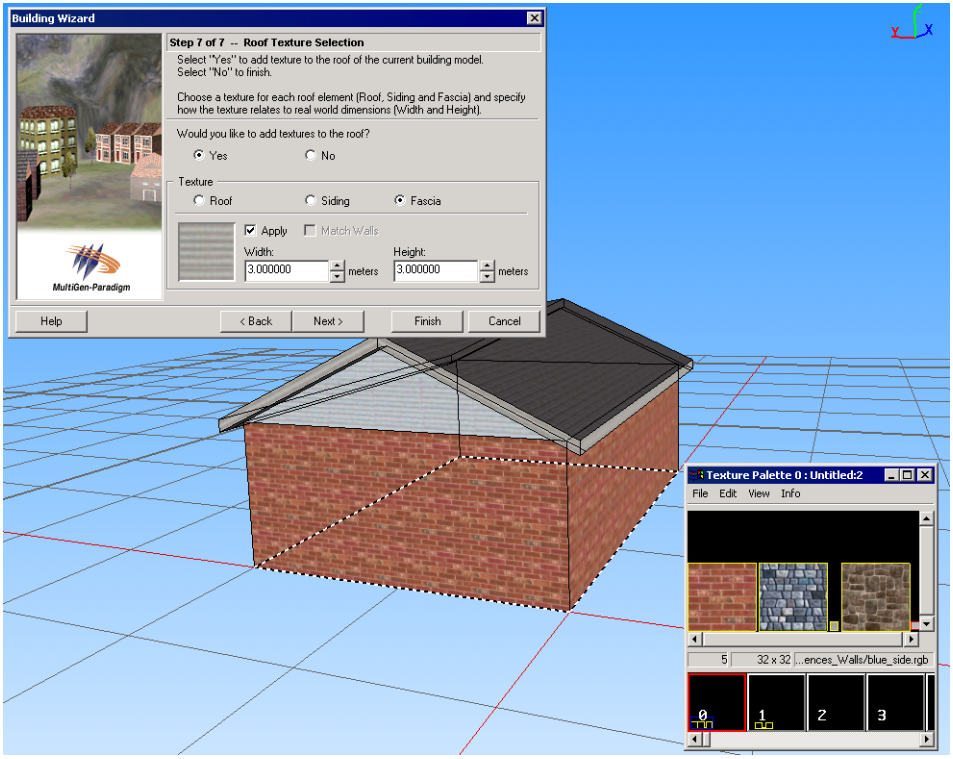
Note: Remember, the real world dimensions for **Texture Width** and **Height** are always measured in meters, so you must enter these values in meters, regardless of the database units employed for your database.

Step 3. Select **Siding**; choose the `blue_side.rgb` texture from bay **0** in the *Texture Palette* (it is the small, lighter-colored texture of siding, on the far right side of bay **0**). Make sure the **Apply** box is checked; set the **Width** to 3 (3.000000) and the **Height** to 3 (3.000000). (If you want the texture for the siding to match the walls, select **Match Walls**; when **Match Walls** is chosen, you cannot select a different texture for the siding.)

Step 4. Select **Fascia**; choose the `e_siding.rgb` texture from bay **0** the *Texture Palette* (it is the small, darker-colored texture of siding, to the left of the red brick texture). Make sure the **Apply** box is checked; set the **Width** to 3 (3.000000) and the **Height** to 3 (3.000000).

Take a moment to look at the house and the roof. Explore other options by applying different textures to different parts of the roof. When you have

finished experimenting with various roof styles, make sure that the options are reset to the parameters covered in the preceding section.



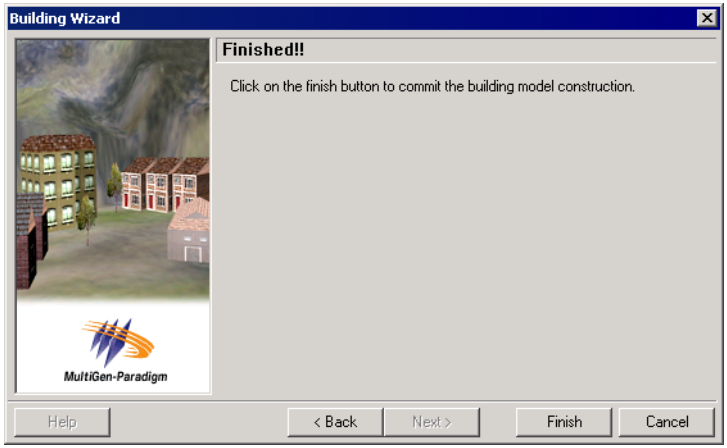
If you wish to modify any particular elements from previous steps in the wizard, you can do so now, using the **Back** button to reconsider your previous choices.

Step 5. Click **Next**, to move to the Finished Button.

Finished Button

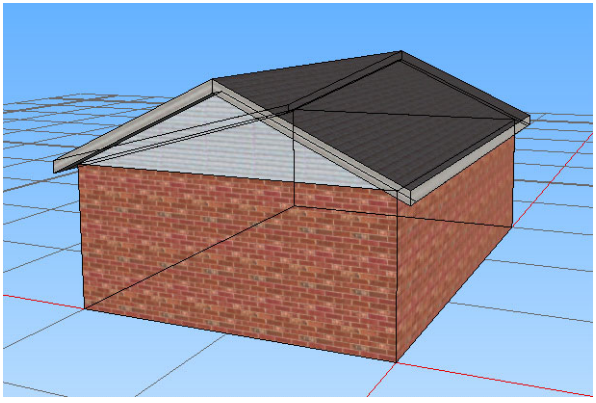
Should you wish to review any of the choices you have made for your model, this panel provides you with one last opportunity to double-check your work. As you look at the model in the main Creator window, you might wish to modify the parameters of your

building; use the **Back** button to move to previous steps and make adjustments. When you are satisfied with your work, click **Finish**.



If you choose **Cancel**, the wizard will close, and your partially-constructed building will vanish, so use this option judiciously.

After you have chosen **Finish** to commit the building model construction, it is a good idea to save your Creator file. Then you can sit back, admire your building, and consider how much time you have saved by using the wizard.



37 Glossary

Click a letter to view the definitions that begin with that letter.

"A" on page 1348

"B" on page 1349

"C" on page 1351

"D" on page 1354

"E" on page 1356

"F" on page 1357

"G" on page 1359

"H" on page 1361

"I" on page 1361

"L" on page 1362

"M" on page 1363

"N" on page 1365

"O" on page 1365

"P" on page 1366

"R" on page 1368

"S" on page 1368

"T" on page 1370

"U" on page 1371

"V" on page 1372

"W" on page 1373

"X" on page 1373

"Y" on page 1373

"Z" on page 1373

A

alpha

A *texture* component (channel) that specifies an amount of transparency for each *texel* in the pattern.

ambient light source

A light source that illuminates all surfaces equally. In Creator these are called Infinite light sources.

ambient reflection

The amount and color of light a *material* reflects from other objects in the scene; brightness.

ancestor node

A node reached by traveling up the hierarchy through a series of parent-child relationships.

anti-alias

A technique used to reduce the jaggedness of lines displayed on a computer screen.

area block

A geographical partition of the DED file (terrain post file). You select area blocks to generate a batch terrain in Creator. Each area block that you select is processed into a separate terrain file, which a master file references. Each terrain file contains its own geometry, LODs, and properties that you define in preferences files.

areal feature *or* areal

A cultural *feature* that is defined by a closed set of ordered, connected vertices (that is, a polygon). The area enclosed by a polygon represents the feature. For example, a lake is represented by the perimeter of its shoreline. An attribute defines the areal's height. Lakes, forests, and cities are typical areal features.

attenuation

The degree to which a spot light's intensity decreases with increasing distance from the position of the light.

attribute window

A window that contains the attribute controls for each type of node in a database.

attributes

Graphic, physical and theoretical characteristics defined for each node type in a database. Attributes provide meta-data for the display and realtime processing of the node.

axis

A reference line in space along which coordinates are measured. A 3D *coordinate system* uses three orthogonal axes (x, y, and z) to determine the location of any point in space.

azimuth

In Creator, the rotation of a light source along a circle through the vertical axis.

B

backface

The side (face) of a polygon that is opposite the polygon normal. In OpenFlight, this is the side on which vertices are numbered clockwise. By default, the backface is not visible. Backfacing polygons are not drawn in most realtime systems.

bead

See "node" on page 1365.

batch process

A process for generating terrain from area blocks that you choose in Creator. Each area block that represents a geographical portion of the DED file is processed into a separate terrain file. With the batch process, you can process multiple terrain databases from a DED file and adjacent databases switch in and out smoothly in the realtime simulation.

billboard

A textured polygon that imitates a 3D model, because it continuously rotates to face the eyepoint.

binary space partition

See “BSP” on page 1350.

binary separating plane

See “BSP” on page 1350.

bounding volume

A simple volume (such as a box, sphere, cylinder, or convex hull) that encloses the geometry attached to a group node. Bounding volumes are used by some realtime systems to provide an estimate of the shape of the group.

BSP

Binary separating planes. A method of resolving drawing conflict by modeling planes that separate portions of a database from one another. Known in some applications as “binary space partition.”

A plane used to bisect two volumes, one on the visible (true) side of the plane, the other on the non-visible (false) side of the plane. A BSP node contains references to the true and false volumes. Graphics systems can use this to determine hidden surfaces by traversing the hierarchy and comparing the eyepoint with the direction of the plane. In theory, this is a much less time-consuming process than z-buffering or ray tracing.

C

CAD

Computer-aided design.

CAT (Continuous Adaptive Terrain)

A Creator terrain feature that provides seamless level of detail (LOD) management.

center of rotation

In the Graphics view, the point around which the eyepoint rotates. Often, this is the *viewing vector's* endpoint.

channel

In *texture*, a component that is assigned a value for each texel in the pattern. For example, intensity textures have only one channel (shade of gray), while intensity plus alpha textures have two channels (shade of gray and amount of transparency).

In realtime, a separately-generated component of a scene.

child node

A node directly below (and linked to) another node in the database hierarchy.

clipping bead

A node that defines a four-sided polygon where each edge, and the polygon itself, defines a clipping plane (a total of five clipping planes). Each clipping plane can be independently enabled or disabled. Clipping beads are used to model instruments. See the Instrumentation chapter for more information on clipping beads.

clipping planes

Two planes that define the near (inner) and far (outer) limits of the *viewing volume* in the Graphics view. Creator does not display geometry that is positioned in front of the near clipping plane or in back of the far clipping plane. Clipping plane adjustment is important in *z-buffer* rendering.

clip texture

A small, square image created from a larger, geospecific base texture, whose height and width are a power of two. SGI's Infinite Reality systems page clip textures off disk and load them into memory as the view moves over the area they represent.

The ability to page texture in as needed, for example, with large textures and mipmaps.

clock

A Creator GUI control used to set a parameter having a circular range of values.

collision detection

The realtime testing of two or more groups for points of intersection. This is often done using the bounding volume associated with a group.

computer simulation

In the context of Creator, an interactive computer-generated environment that mimics aspects of the real world in order to perform research or provide training or entertainment. A common application of computer simulation is to allow pilots to practice take-off and landing.

concave polygons

A polygon with at least one vertex that bulges inward, rather than outward. In general, convex polygons render more efficiently (and are thus more desirable) than concave polygons.

consecutive LODs

Two levels of detail representing the same model, in which the **Switch Out** attribute of the lower LOD matches the **Switch In** attribute of the higher LOD.

constraint

A control that limits the input or transformation of geometry in one or more axes.

construction edges and vertices

Temporary vertices and edges that are created to use as reference points during modeling. Construction edges and vertices are discarded when you close a database.

convex hull

The smallest convex region that completely encloses a geometric shape. Building a convex hull is the polygonal equivalent of "shrink wrapping" the geometry—none of the surfaces of a convex hull are allowed to bulge inwards.

coordinate

A point in space, as defined by its numerical (x, y, and z) components.

coordinate axis

See "axis" on page 1349.

coordinate system (3D)

A positional system, defined by three orthogonal axes (x, y, and z), in which three-dimensional geometry can be described. See "MultiGen coordinate system" on page 1364.

coplanar polygons

Polygons that both lie in the same plane and overlap.

cull

To ignore unnecessary faces from a visual scene. For example, to ignore polygons that are outside of the field of view, closer than the near clip or farther than the far clip. Realtime systems use culling to reduce the number of polygons that must be processed before rendering a visual scene.

culture

Data that describes what sits on top of the land; for example, lakes, forests, fields, buildings, roads and rivers. Individual culture objects are called features. In Creator, feature data must be converted to DFD format before it can be edited or projected.

This term can also refer to all models in a database that are not part of the terrain.

D

database

A file containing data that describes a visual scene.

database header

The root (top) node in the database hierarchy.

database origin

See “origin” on page 1365.

database traversal

The order in which the database hierarchy is processed for rendering. The Creator database traversal is “depth-first,” which means nodes are processed from top to bottom and then from left to right.

database tree

The entire hierarchy of a database.

database units

The units of linear measurement used in a database.

DED *or* .ded

Creator’s internal file format for elevation post data derived from DTED.

degree of freedom (DOF)

A possible direction of motion. In a 3D coordinate system, six degrees of freedom define a full range of motion from any position: up, down, left, right, forward, and back.

Delaunay triangulation

A method for forming a polygonal terrain surface from elevation post data. Delaunay triangulation creates a terrain surface that is a series of triangles. The vertices are not regularly spaced; Creator’s terrain generation algorithms choose vertices from a DED file based on the priorities you establish by setting triangulation parameters. The Delaunay algorithm is used because it avoids long, skinny triangles.

DEM (digital elevation model)

A form of elevation post data published by the U.S. Geological Survey. DEM is the primary elevation data available to civilians. Creator imports this format through a utility, `readusgs.exe`, which converts DEM data to DED.

descendant node

A node reached by traveling down the hierarchy through a series of parent-child nodes.

DFAD

Digital Feature Analysis Data. Vector data supplied by the National Imagery and Mapping Agency (NIMA) to describe natural and man-made features such as lakes, roads, and industrial buildings. Each DFAD file represents a specific geographical region of the Earth. DFAD data is laid on top of DMA DTED data (in the `.ded` format), which is data representing the shape of the Earth.

Information about the DFAD specification, including *SMC* and descriptions, is posted on the Internet by NIMA at `http://164.214.2.59/publications/specs/draft/DFAD/dfad.html`.

DFAD manuscript

A set of cultural data representing a geographical area less than or equal to one degree by one degree of the Earth. A raw DFAD file may contain several manuscripts. However, when Creator converts raw DFAD to DFD format, each manuscript is saved in a separate DFD file.

DFD *or* .dfd

DFD (Digital Feature Data) is a form of DFAD developed by MultiGen as an internal culture file format. It is 32-bit, for more efficient processing, and supports a higher resolution that does standard DFAD.

diffuse

A material property representing the amount of light and color reflected from a surface in all directions, regardless of eyepoint position.

When applied to light sources, diffuse is the light reflected from the angle formed by the light direction and the vertex normal.

DIS

Distributed Interactive Simulation. The process of sharing a simulation across a network, so that multiple users can participate. These simulations are also referred to as MUD (multiple user distributed simulations).

DLG

Digital Line Graph. A form of culture data published by the USGS.

DMA

Defense Mapping Agency. See "National Imagery and Mapping Agency (NIMA)" on page 1365.

drawing order

The order in which the faces of a database are drawn to produce the appearance of depth in a visual scene.

DTED

Digital Terrain Elevation Data. Gridded elevation data that describes terrain contours, supplied by the *NIMA*. Creator imports this format through a utility, `readdma.exe`, which converts DTED data to DED.

E

edge

A line segment between two consecutive vertices in the same face.

elevation

In lighting, the rotation of the light source relative to the vertical axis.

elevation post data

Data that describes the shape of the land, usually in terms of its height above sea-level. Elevation data is usually organized into a regularly-spaced grid of height fields. An individual height field is called a "post." DTED, DEM, and DED files are all organized in this way.

ellipsoid

A mathematical model of the shape of the Earth, which bulges in the middle and is flattened at the poles. An ellipsoid is the formula which defines that shape. There are many different ellipsoids used in Cartography, but the primary ones used in US data are WGS 1972 and WGS 1984. Ellipsoids are used in conjunction with projections to convert real world locations to places on a map.

emissive

The amount and color of light that a material produces. The brightness and color of the highlight or glare.

environment variable

A variable that defines default settings for directory paths and other controls, such as the default location of textures. Environment variables are set in the operating system, and take effect each time Creator is started.

eyepoint

The point in 3D space from which a database is viewed. See “viewing volume” on page 1372.

eyepoint scale

The magnification of a visual scene relative to the eyepoint. This affects the area of the database that is visible within the *viewing volume*.

F

feature

An instance of culture; a cultural object. Features can be *points*, *linears*, or *areals*. Each feature contains vector data (a set of ordered vertices) plus attribute information that defines the characteristics of the feature.

feature analysis code (FAC)

The unique DFAD identifier for each feature in cultural data. This information is discarded when the culture is projected into the database.

FAC1

The feature analysis code (identifier) for the first feature in every DFAD manuscript. FAC1 is the background feature; it covers the entire *DFAD manuscript*, usually representing soil or water. It is always an *areal*.

FACC

A standard scheme of culture attribution published by the DMA. FACC is a living standard; it is updated each year.

face

A collection of ordered, coplanar vertices describing a surface. Also called a polygon.

far clipping plane

See “clipping planes” on page 1351.

feature

An instance of culture; a cultural object. Features can be *points*, *linears*, or *areals*. Each feature contains vector data (a set of ordered vertices) plus attribute information that defines the characteristics of the feature.

feature analysis code (FAC)

The unique DFAD identifier for each feature in cultural data. This information is discarded when the culture is projected into the database.

feature identification (FID)

A number and description that identifies the nature of the feature. For example, **174: Strip mine**. Every FID can occur in *point*, *linear*, or *areal* form.

field of view

An angle relative to the *viewing vector*, that defines a conical area of visibility in the database. Polygons outside of this cone are not displayed. See “viewing volume” on page 1372.

fixed list

A rendering method that uses the position of each *node* in the hierarchy to determine the *drawing order* of a scene.

flat shading

No shading. Faces are displayed in their assigned colors.

FLT or .flt

Creator OpenFlight file format.

frame rate

The rate at which a visual scene is dynamically updated and rendered. More generally, refers to the rate at which the entire realtime system is processed and updated.

frustum

The volume of a cone between the base and a plane parallel to the base.

G

geocell

An area of data which measures one degree of latitude by one degree of longitude. Elevation and culture data are often organized into geocell-sized files. Also see "DFAD manuscript" on page 1355.

geocoordinate

A location on the Earth specified in terms of latitude and longitude.

geodetic format

The projection format of a geospecific image.

geographic information system (GIS)

A CAD-like application designed to edit terrain data, primarily culture. GIS systems are used by a wide variety of professions, including cartographers, civil engineers, geologists, and surveyors. Commercial GIS applications are not geared for realtime visual simulation. Like architectural CAD programs, they are designed primarily for accuracy, not for speed. ARC/INFO is a leading commercial GIS. Another is GRASS, a public domain PC-based GIS initially created for the Army.

geometry

A collection of 3D polygons that represent one or more physical objects.

geospecific texture pattern

Aerial or satellite imagery of a region of the earth, or a bitmapped image, applied as a texture to a terrain model of the same region. The image contains geographical attribute information – the image's latitude and longitude. The terrain area it is mapped to must encompass the same coordinates. Geospecific textures typically are used to apply satellite photographs to large terrain areas. Because Creator automatically correlates geospecific attributes with database coordinates, a geospecific texture can be applied automatically during terrain conversion. It can also be applied with the **Geoput Texture** tool.

geotypical texture pattern

Texture applied to a terrain model from a different location in the real world, e.g., generic forest texture.

geographic information system (GIS)

A CAD-like application designed to edit terrain data, primarily culture. GIS systems are used by a wide variety of professions, including cartographers, civil engineers, geologists, and surveyors. Commercial GIS applications are not geared for realtime visual simulation. Like architectural CAD programs, they are designed primarily for accuracy, not for speed. ARC/INFO is a leading commercial GIS. Another is GRASS, a public domain PC-based GIS initially created for the Army.

Gouraud shading

Shading where faces are displayed using vertex colors when **Calculate Shading** is performed.

Graphics view

A 3D representation of a *database*.

grid

A two-dimensional coordinate grid, located on the tracking plane, which can be defined in Cartesian or Radial coordinates. The grid can be visible or invisible. It provides a measure for coordinate entry, and allows you to see where the tracking plane is positioned.

group

A collection of objects or other logically related nodes in the database hierarchy.

H

hierarchy

The logical, efficient grouping and organization of a database's structure that determines the drawing order of its individual components and defines the relationship between items in the database. This structure is set up as a "parent-child" relationship.

Hierarchy view

A 2D structural representation of a database.

HLS

Hue, Lightness, and Saturation components of a color.

Hz

Abbreviation for hertz, a unit of frequency equal to one cycle per second.

I

image generator (IG)

A generic term for a high performance computer used to render 3D images in real time. For example, a RealityEngine is a Silicon Graphics IG.

indirect texture

An indirect texture contains a bitmapped image file that defines specific colors, textures, and materials as well as an ASCII file that keys each color in the image to a specific group of properties. Indirect textures are applied to terrain.

L

levels of detail (LODs)

Sets of models that represent the same object with varying amounts of complexity. The realtime system selects one of the LODs to display, depending on the distance from the eyepoint to the LOD and on the number of polygons the realtime system can process.

light points, strings, grids, and polygons

Light points are individual vertices that represent distantly-viewed points of light such as stars, runway lights, or city lights. Like all vertices, light points can be adjusted and moved independently after you create them. Unlike light sources, light points do not illuminate faces, or any other part of the visual scene, or affect shading.

Light strings are light points that are arranged along a predefined path. *Light polygons*, and *light grids* are arrangements of light points that produce special effects such as lighted cities. You can choose the type of light points you want to create and define their arrangement, color, and other properties at the time you create them using the **Create Light Point** tool.

light source

A modeled light that illuminates all or part of a database. There are three types of light sources: *infinite*, *local*, and *spot*.

linear

A cultural *feature* defined by a set of vertices, connected in a line. A linear feature has attributes that define its width, and height. Roads, rivers, and tree lines are typical linear features.

lit Gouraud

Display mode in which faces are rendered using both lit shading and Gouraud shading.

lit shading

Shading that uses vertex normals generated by the **Calculate Shading** function, as well as applied material attributes and the effects of any light sources.

local coordinate system (LCS)

A coordinate system defined by the user, usually when modeling far from the database origin.

local light

A light source that is located at a point in space and shines in all directions. Local light sources have ambient and diffuse properties.

LOD

Acronym for level of detail, pronounced "L-O-D." See "levels of detail (LODs)" on page 1362.

M

mask

In switch nodes, an array of zeros and ones. Each digit in the array controls whether the corresponding child node is displayed or hidden.

In image processing, a selected area of the image that is temporarily protected from editing.

material

A set of properties that, when applied to a face, simulate the light-reflecting characteristics of real world objects such as wood, plastic, and metal. Material properties include ambient, diffuse, emissive, shininess, and specular.

MIP mapping

The process of automatically scaling a texture to fit a polygon regardless of the polygon's screen resolution.

MIP map set

A set of textures whose texel widths and heights must be a power of two. The textures are related because they are different resolutions (to the power of two) of the same image.

model

A 3-dimensional structure comprised of geometry, hierarchy, and attributes.

modeling mode

The type of hierarchy *node* that is selected for processing in Creator.

morphing

A smooth visual merging of one *level of detail* into another.

Mosaic texture pattern

A *geospecific texture pattern* that is a composite of a number of separate texture files, each of which can be swapped in and out of memory as needed during a realtime simulation.

MultiGen coordinate system

MultiGen uses a three-dimensional, right handed, Cartesian coordinate system where *x* points to the right, *y* points straight ahead, and *z* points up.

MultiTexture

MultiTexture is a feature that allows you to assign up to 8 texture layers to each face. The texture assignments are stored in the face attributes, but the layers are applied using the View panel. Your hardware must be able to display multiple textures in order to see the blending effect. See “MultiTexture” on page 470

MVF

MVF (MultiGen Vector Format) is similar to DFD, but includes a generic attribution structure that allows for storage of additional data for each feature. MVF files are created during the conversion of CTDB files to OpenFlight format, or by exporting to MVF.

N

National Imagery and Mapping Agency (NIMA)

The government agency that replaced the Defense Mapping Agency (DMA). NIMA supplies defense-related projects with accurate information about the elevation, as well as specific features of any given area of the globe.

near clipping plane

See “clipping planes” on page 1351.

node

Fundamental element or building block to construct a database hierarchy. Nodes represent both graphical elements like geometries and procedural elements that maintain scene hierarchy, like *group* or *switch* nodes.

normal

A *vertex* normal (used in shading) is a vector that extends out from the vertex of a polygon and is perpendicular to the face.

A *face* normal (not used in Creator) is a vector that is perpendicular to the frontfacing (visible) side of a polygon.

O

object

A collection of *polygons*.

OpenFlight

The scene-description format used in all Creator databases. Creator database files are saved using the OpenFlight format.

origin

The center of a 3D *coordinate system*, defined by the point x=0, y=0, and z=0.

orphan node

A *node* that has no point of attachment in the hierarchy, although it may have descendants. Orphan nodes are not saved when you close a Creator database.

ortho image

A square or rectangular 2D image.

orthographic view

Flat (2D) projection of a 3D scene, such that object sizes are not dependent on their position (perspective is not represented). Also see “perspective view” on page 1366.

orthorectification

The process of converting an aerial or satellite image from a perspective view (as photographed) to an orthographic view. Imagery must be orthorectified before it an be used as texture.

P

pan

To move the eyepoint up, down, left, or right, always parallel to the view.

parent node

A *node* directly above (and linked to) another node in the database *hierarchy*.

perspective view

Depth (3D) projection of a visual scene, such that object sizes are dependent on their position. Also see “orthographic view” on page 1366.

phototexture

Texture that is derived from a photograph.

pitch

Rotation around the x-axis.

pixel

The smallest display unit of a video image. Originally, a combination of the terms "picture element."

pixel depth

The distance from the *eyepoint* to each pixel in a visual scene. Used in the computation of *z-buffer* drawing order.

pixel unit format

The projection type of a geospecific image.

point

A cultural *feature* defined by a single vertex representing its geographical position. A point feature has attributes that define its length, width, and height. Buildings are typical point features.

polygon

A flat geometric shape having at least three *edges* (sides). Also called a *face*.

polygon normal

See "normal" on page 1365.

Polymesh triangulation

A terrain triangulation method where the vertices are regularly spaced along the *x* and *y* axis.

popping

An abrupt change in the appearance of a visual scene that may occur when a *level of detail* switches in or out of the display.

projection

A mathematical formula for mapping the round Earth onto a flat surface, such as a piece of paper (a map). Projections (in conjunction with *ellipsoids*) are used to convert between latitude/longitude coordinates and Cartesian coordinates. All projections distort the data. Different projections distort the data at different places on the Earth.
See "ellipsoid" on page 1357.

R

raster

Information organized into a regularly spaced two-dimensional grid, e.g., images and elevation post files are both raster data.

realtime system

A combination of computers, software, input devices (such as joysticks) and output devices (such as computer monitors) that process data and provide feedback to a user in a predictable and realistic fashion. See “computer simulation” on page 1352.

relative priority

A method of fixing the *database traversal* order among *sibling nodes* by assigning each node a numerical attribute. Nodes with higher relative priority are sorted to the right in the database hierarchy.

render

To draw a visual scene, given as input a 3D database describing the scene, and a *viewing volume* describing the amount of the database that is visible. The term render also refers to non-visual effects, such as the playing of sounds in the database.

roll

Rotation around the *y*-axis.

root node

The node at the top of the database hierarchy. Root nodes are created automatically for each new file and cannot be deleted or explicitly selected.

S

select

To choose an item for processing in Creator. Sometimes called “picking.”

shading

The creation of natural lighting effects in a database.

shininess

A value that controls the amount of specular reflection, or highlight, of a *material*. The value range is 0.0 to 128.0. A high shininess value gives a metallic appearance.

sibling nodes

Two or more children of the same parent *node*.

simulation

See “computer simulation” on page 1352.

slant range

In Creator, the distance from the eyepoint to the center of a *level of detail (LOD)*. In a *realtime system*, this distance is dynamically recalculated for each LOD as the *eyepoint* moves through the database.

specular

In *lighting*, the color of the highlight that a light source produces on objects in the database. Specular components can be set for infinite, local and spot lights but are not used in Creator rendering.

In *material* properties, the color of the highlight that is reflected from a shiny object. The specular property is directly affected by *shininess*.

spheroid

See “ellipsoid” on page 1357.

spot light

A local light source having a coordinate position, direction vector, and a cone of effect.

subface

A *polygon* that has another polygon for its parent. A subface is modeled on top of, and in the same plane as, the parent polygon, and is intended to give the appearance of layering.

Subtexture

Subtexture lets you define a smaller area of a texture to apply to a face. See "Subtexture" on page 478 for more information.

subtree

Any *node* in the hierarchy, together with all its descendants.

superface

A polygon that is the parent of another polygon (its *subface*).

Surface Material Code (SMC)

An attribute for each cultural *feature*, which represents its surface type, e.g., cement, soil, metal, glass, etc. Surface material code is used to set up automatic texturing of the geometry created when DFAD features are projected into a database.

switching

The process of selecting a *level of detail* (LOD) for display based on the attributes of the LOD node and the distance from the center of the LOD to the *eyepoint*.

T

Terrain Culture Triangulation (TCT)

A constrained Delaunay algorithm that extracts feature data from DFD files as it processes elevation data. Feature data is pre-projected and the terrain is stitched around features so that features correctly align with terrain. TCT is useful for ground-based simulations where more detail is needed on terrain skin.

tiling

The practice of breaking a terrain database into smaller separate files which fit together like floor tiles. This frees the *image generator* from having the entire database in memory at one time, and instead allows it to have only what is needed at the moment.

tracking plane

A plane defined within the three-dimensional coordinate space of each Creator Graphics view. Used to position vertices that are entered with the left mouse button.

The tracking plane includes a two-dimensional *grid*, which can be defined in Cartesian or radial coordinates.

transformation

A change in the position, size, or orientation of an object in 3D space.

transformation matrix

In Creator, a matrix, applied to a *node*, that changes the apparent position, size, or orientation of the node and all its descendants without changing the coordinate values of the geometry.

transparency

A value that determines the relative transparency of a face. In a *material*, a face that is partially transparent is blended with the face that is drawn behind it in the visual scene. Texture patterns can also contain areas of transparency, through the use of an *alpha* component. This produces ghost images.

traversal order

See “database traversal” on page 1354.

triangular irregular network (TIN)

A series of triangles that form a surface, usually a terrain surface. The vertices in a TIN are not regularly spaced, as in a grid. TINs are the preferred method to model terrain, rather than a *polymesh*, because a TIN can represent the same terrain to a higher level of accuracy using the same number of polygons.

U

u and v coordinates

u is the relative width of a texture. ***v*** is the relative height of a texture. ***u*** and ***v*** always equal 1.

United States Geological Survey (USGS)

A branch of the U.S. Department of the Interior, a civilian agency that, among other things, compiles and distributes cartographic data to civilians.

universal transverse mercator (UTM) format

A set of standard projections.

V

vertex

A coordinate point in a 3D space, defined by its *x*, *y*, and *z* components.

vertex normal

A direction vector associated with shading. Calculate Shading calculates the vertex normals and allows them to be averaged. See “normal” on page 1365.

viewing frustum

See “viewing volume” on page 1372.

viewing vector

A vector with the eyepoint as its origin and ending in the direction the observer is looking. Used in the determination of *viewing volume*.

viewing volume

The portion of the database that is visible in the Graphics view. Geometry outside of the viewing volume is not rendered. The viewing volume is a cone, defined by the *field of view*, that extends from the *eyepoint* in the direction of the *viewing vector*. The amount of geometry visible within this cone is further restricted by the near and far *clipping planes*. Also called the *viewing frustum*.

viewpoint

See “eyepoint” on page 1357.

W

WGS72

An ellipsoid. This was the standard ellipsoid of the *DMA* prior to the establishment of WGS84

WGS84

The standard ellipsoid of the *DMA*.

wireframe

A rendering method in which only the edges (outlines) of polygons are drawn.

X

x, y, z

Variables that define the position and orientation of an object in 3D space. See "ellipsoid" on page 1357.

Y

yaw

Rotation around the z-axis.

Z

z-buffer

A software or hardware buffer that stores information about the distance from the eyepoint to each pixel on the surface of a polygon in a visual scene. Also, a *rendering* method that uses a z-buffer to determine the proper *drawing order* of a scene.

zoom

To move the eyepoint forward or backwards along the current *viewing vector*.

Index

Numerics

180 degree curves, in road generation 1245
2D text, placing 812
3D Studio file import 1226
3D Text tool 119
4-Point Put Texture tool 450

A

AASHTO, policies on road design 1251
Accuracy tolerance in CAT conversion 1040
Action Name, GeoFeature Action Definition preference 1149
Actual Size 650, 653
Adaptive attributes
    Adaptive Scope 1046
    Adaptive Type 1046
    Far Switch Distance 1046
    Near Switch Distance 1046
    Offset Distance 1046
    Origin 1045
    Vector 1046
Adaptive Mode tool
    creating an attribute 1044
    creating attributes on vertices 1044
    description 237
    editing attributes 1045
    overview 1043
Adaptive Type, Adaptive attribute 1046
Add Vertex tool 186
Adding new road segments 1234
Additive LODs 711
Additive LODs Below 711, 714
address
    MultiGen-Paradigm, Inc. 3
Algorithm
    advantages and disadvantages 983
Alignment control point 1235
All Off, switch attribute 864
All On, switch attribute 864
Allow Default Parent 63, 226, 291
Alpha 391, 392
Alternate Color
    Index 842, 854
    Intensity 842, 854
    Name 842, 854
    RGB value 842, 854
Ambient
    Modify Light Source parameters 609
    Modify Material parameters 392
Ambient Intensity 647, 648
Ancestor 280
Angle Limit 630
Angular Tolerance 623
Animation
    creating with Flip Book tool 128
    overview of group animation 759
    parameters in Create Light Point tool 654, 656
Antialias Lines 316
API 37
appending road sections 1236
Application window, description of 69
Applications processing, effect on terrain 957
Apply GeoFeature Texture 1139, 1140
Applying a line style from a face 206
Applying a line style to a face 206
Applying material properties 394
Applying road attributes 1256
Arc Radius
    for special curves 1246
    Horizontal Curve controls 1252
Arcs
    in dial gauges 779
Area Block batch processing
    enabling 1050
Area Block File Naming, Batch preference 1050
Area Block Size in X (and Y), Batch preference 1050
Areal feature
    creating 1102
Arrange Icons 78
Arrow keys, selecting nodes with 331
ASCII files
    converting to DED 1067
ASCII text reports, for driving simulation 1276, 1278
Assigning DFAD rules to LODs
    procedure 1156
Attach tool 230
Attenuation controls 610
Attribute Resolver 880
Attribute search
    Attributes menu 872
    procedure 869
    reopening an attribute search 872
Attributes 292

BSP 864
clipping node 806
curve 851
DOF 379, 865
face 1291
general information 825
Comments 830
Display box 829
Modify All 827
navigation 828
opening 826
Reflect Current Selection 827
generic 869
group 835
header 832
in defining models 241
light points 661
light source index 602
LOD 712, 1290
path 1289
road 1289
Sound 868
text 818
texture 429
texture - see “Texture attributes”
Use Attribute Index Matching preference 64
vertex 848
viewing and changing 826
Attributes menu
Animate 761
Attribute Resolver 880
Attribute Search 872
Calculate Shading (Ctrl+G) 622
Modify Attributes (Ctrl+=) 826
New Attribute Search 870
Place Light Source 614
Put Bounding Volume 698
Rename (Ctrl+J) 241, 289
Attributes, header
Draw New Textured Polygons White 530
Auto Apply Current Material 394
Auto Apply Texture and Mapping 545
Auto LOD 715
Auto Save 61, 247
Auto Save Checkpoints 247
Auto Save Confirmation Prompt 61, 247
Auto Save Every k Minutes 61, 246
Auto Slice if Not Separable 914, 928
AutoCAD
exporting files to 1229
importing files from 1228
Automatic trackplane alignment 1235
Average Vertices tool 204
procedure 204
Azimuth
Modify Light Source parameters 608
Modify Vertex Normals parameters 630

B

Background image 361
aligning with a model 363
Build Polys From Image 1106
displaying 365
drawing mode and 362
loading 361
modeling with 363
View control 318, 362
Backing up files automatically 245
Backup Distance, Feature parameter (Road) 1270
Balanced spirals, in road generation 1252
Baseline Controls, Sky Color 271
Batch
saving batch preferences 1049
Batch Culture conversion 1177
Batch GeoPut
overview 1198
using 1198
Batch GeoPut texture coverage
Full 1201
Partial 1199
Batch GeoPut, Terrain menu 1197
Batch preference file (terrain) 891
Batch preferences
Area Block File Naming 1050
Area Block Size in X (and Y) 1050
Calculate Shading 1050
Deselect All 1052
Enable Area Block Processing 1050
Read Select 1052
Select All 1052
Show Grid 1050
Units 1050
Batch processing
adding culture to existing terrain 1169
master file 1029

procedure 1178
Bead - see “Node”
Bend Deform tool 178
parameters 179
procedure 178, 179
Best Fit, Polymesh preference 1016
Beveled geometry 98
Bidirectional 1106, 1113
Billboard wizard
tutorial 1293
Billboards 380
Billboard Wizard tutorial 1293
creating 380
positioning 381
positioning an instance 382
Binary files
converting to DED 1067
Binary separating planes - see “BSP separation”
Black Hot/White Hot 417
Blend texture algorithm
MultiTexture 470
Block processing
Batch preferences 1049
Bottom Left/Upper Right, Polymesh preference 1017
Bottom Right/Upper Left, Polymesh preference 1017
Bounding volumes 694
changing 696
displaying 695
translating 698
BSP attributes 864
BSP drawing order
about 686
setting in State Preferences panel 912
BSP menu
Check Separability 913, 915
Collapse BSP 926
Create BSP Plane 924
Diagnosis 927
Display Planes 926
Recursive Separation 913, 920
Remove Plane/Hulls 926
Select Planes 926
Separate Children 913, 919
Separate Selected 913, 916
BSP Mode tool 236
BSP preferences 913
Auto Slice if Not Separable 914, 928
Generate Only Vertical Planes 914
Interpenetration Tolerance 914
Separate Hierarchy at Lowest Level 914
Skip Disabled Nodes During Separation 914
BSP separation 907
combining with Fixed List 908
concave object problems 929
correcting failures 926
interpenetrating hull problems 928
manual BSP insertion 923
preferences 913
tips and techniques 909
tree structure 908
Build Polys From Image 1106
Building roads 1232
Building wizard
tutorial 1324
Bulldoze tool 182
Bump map
MultiTextures 471

C

CAD view (Shift+F10) 79
resetting 79
Calc. RMS Error 942
Calculate Plateaus From 838
Calculate Shading (Ctrl+G) 622
Angular Tolerance 623
Flat shading mode 623
Lit Gouraud shading model 460, 623
Lit shading model 623
Sampling Tolerance 623
Triangle Area Weighting 623
Update Vertex Colors 625
Update Vertex Normals 623
Use Face Color Intensities 625
using vertex colors and normals 620
Calculate Shading, Batch preference 1050
Calculate, group attribute 837
Cascade (Shift+F5) 77
CAT
advantages and disadvantages 1011
CAT attributes
Adaptive 1043
Alt Color 1042
Alt Color Name 1043
Alt Intensity 1043
Color 1042

Color Name 1042
Detail Texture 1043
Infrared 1042
Intensity 1042
IR Material 1043
Material 1043
SMC 1043
Texture 1043
Texture Mapping 1043
CAT preferences
Accuracy Tolerance 1040
Cutout Feature List 1041
Maximum Poly Count 1040
Z Scale Multiplier 1041
CAT terrain conversion
displaying in realtime 1038
level of detail management 1035
level of detail overview 1037
level of detail switching, description 1037
levels of detail 1037
node attributes 1041
overview 1035
preferences 1039, 1040
Catdma
DED concatenation utility 1212
Center model at origin 352
Center of rotation 256
repositioning 257
setting eyepoint to 257
Center Select (C) 261, 285
Center to Left
Road Bed parameter 1257
Scenario parameter 1282
Center to Left, Preview Tessellation control 1244
Center, group attribute 837
Centerline
as a drivable path 1281, 1283
data files 1277
distance to left edge 1244
distance to left edge of road 1257
hidden faces 1285
lane definition 1277
length of curve 1241
lofting 1278
spacing features on roads 1270
Centerline Offset
Feature parameter (Road) 1270
Road Lane parameter 1285
Centerline tool 216
Change mode (Ctrl+Shift+C) 233, 339
Change Vector Delta 630
Changing a material name 391
Changing textures in the Texture palette 427
Changing the current material 396
Check Relative Priority When Shifting Nodes Left or Right 62, 230, 231, 288, 688
Check Separability 913, 915
Checkpoints
Auto Save 247
Child 279
Child edges, in terrain generation 1038
Child faces, in terrain generation 1039
Chop Texture 464
Circle Plus tool 92
Circle tool 91
Circle, creating 91, 342
Clamped texture pattern 445
Clear Construction 220
Clear Each Time a Database is Closed 542
Clear Heights 1034
Clear Points (Modify Delaunay) 971
Clearing selected area blocks 1052
Clip at Road Poly, Feature parameter (Road) 1271
Clip Mode tool 236
Clip Nodes
turning display off and on 319
Clipping nodes
definition 802
nesting 807
Clipping plane, procedure 689, 691, 692
Clipping polygons 807
Clock controls, defined 80
Close (Ctrl+W) 248
Close All 78
Close Profile
Road LOD parameter 1263, 1266
Scenario parameter 1282
Closest Point on a Line tool 222
Closest Point on a Plane tool 224
Coastline Posts, Delaunay preference 1024
Collapse BSP 926
Collect tool 232
Collision detection 694
effect on terrain 957

Color
color intensities for shading 621
color matching 415
Color Name, face attribute 402
default Color palette 413
effect on texture 530
Elevation Scale 951
Face color with geospecific textures 1202
Fog Controls 268
in road side area 1265
Road LOD parameter 1264
Use Attribute Index Matching preference 64
Color and feature projection 1135
Color matching 64, 414
Color palette (Ctrl+P)
creating default palette 414
index, definition of 405
loading 414
opening 399
saving 413
Color Palette, Project panel preference 1001
Color parameters, Light Points tool 637
Color scale for lights - see "Light Ruler"
Combine Faces tool 189
using 189
Combine Similar Faces Only editing preference 190
Comments in attribute windows 830
Compound radial curves
in road generation 1246
Concave objects, inseparability issues 929
Constrain DOF 740, 744
Constrain, DOF attribute 866, 867, 868
Construct Average Vertex tool 221
Constructing circular roads 1247
Construction Curve 218
clearing 220
creating 218
tool 218
Construction Tool parameters
Control Points 1235
Delete Last 1236
Horizontal Curve controls 1236
Next Section 1235
Preview Tessellation controls 1243
Previous Section 1235
Section Creation 1236
Trackplane Control 1235
Twist Type 1238
Vertical Curve controls 1239
Construction tools
Construct Average Vertex 221
Vertex From Mouse 220
Constructions
turning off 319
contacts
MultiGen-Paradigm, Inc. 3
Continuous, Feature parameter (Road) 1271
Contour colors
assigning to terrain elevations 947
used by Bulldoze tool 183
Contour line 947, 993
Contour properties
assigning 947, 949, 995, 997
Face Colors From 996
Get Face Color From 948
Indirect Texture 948
Material 951, 997
Texture 951, 997
Contours
adding 994
applying texture 1183
creating uniform contours 995
deleting 994
description 993
Fixed Elevation 995
properties 995
resizing 993
selecting 993
Contours, terrain 947
adding 950
creating uniform contours 950
deleting 950
Fixed Elevation 951
properties 947
resizing 949
selecting 949
Contract Hierarchy tool 232, 281
Control bar 70
Control Points, Construction Tool parameter 1235
Conversion utilities 940
Convert Database Projection
description 1059
procedure 1061
Convert Database to DED 1214

Adding post data to an existing DED file 1216
Create New DED File option 1215
database boundaries 1216
Face attributes during conversion 1215
Insert Posts Into Existing DED File option 1215
Converting elevation data to DED format 1089, 1210
Converting external references to instances 374
Convex hull, definition 907
Cookie Cutter tool 176
example 176
Coordinate panel 341
Current 342
Delta 342
displaying 74
Distance 342
Reference 341
using to create rectangles 342
Coplanar faces
correcting non-coplanar faces 357
Select Nonplanar Faces calculation 334
with z-buffer drawing 687
Copy
light source definition 618
Copy (Ctrl+C) 72, 346
Copy Culture Attributes tool 208
Copy GeoFeature Attributes 207, 208, 1115
Copy texture layers 474
Correlated data
roadway centerline files 1277
scenario lane files 1283
Create
blank texture 426
Create Areal Feature 1102
Create BSP Plane 924
Create Clip tool 228
Create External Reference tool 373
Create GeoFeature action
procedure 1146
Create Instance tool 228, 369
Create Light Points tool 227, 660
Animation parameters 654, 656
Color parameters 637
Direction parameters 639, 646
Create Light Points tool, Performer parameters 648
Create Light Source tool 599
Create Linear Feature 1104
Create LOD tool 227
Create Object tool 227
Create Point Feature 1101
Create Quad Tree LODs, Delaunay preference 1021
forcing straight boundaries 1022
Create Side Areas, Road Bed parameter 1257
Create Sound tool 227, 673
Create Switch tool 228
Create tool (Shift+C) 226, 290
Create tools
Create (Shift+C) 226
Create BSP 228, 923
Create DOF 227, 736, 737
Create External Reference 228
Create Group 226
Create Instance 228, 369
Create Light Points 227, 660
Create Light Source 228
Create LOD 227
Create Object 227
Create Sound 227, 673
Create Switch 228
Create, GeoFeature Action List preference 1147
Creating
light points 631
line styles 822
Creator preferences
Draw Sound 673
selection line color and type 328
Creator Terrain Studio (CTS)
overview 1204
creator.ksc file 889
creator1.prefs file 889
Crests, in road generation 1249
CTR - see “Trackplane Center”
Culling
balancing with drawing 957, 1057
defined 300
effect on terrain 957
Cultural Footprint, face attribute 848
Cultural Footprint,mesh attribute 859
Culture
adding to existing batch terrain files 1169
Culture conversion in batch mode 1177
Current Alternate Color 75
Current directory, defined 541
Current IR Color 75, 416
Current Light Source 75

Current Mask Index 863
Current Material 75, 387
Current modeling mode 327
Current Primary Color 74
  definition of 401
Current Texture 75
Current Texture display 425
Current Texture Map 75
Current Texture pattern, definition of 423
Current, DOF attribute 866, 867, 868
Curve attributes 851
Curve Length, Vertical Curve control 1241, 1242
Curve Mode 118, 236
Curve road sections 1236
Curve tool 116
  modifying existing curves 118
  procedure 116
Curve Type 852
Curved String 636
Curves
  Arc Radius 1252
  changing curve types 118
  compound radial 1246
  cutting and pasting 119
  definition 1236, 1239, 1244
  deleting 119
  modifying 118
  moving 119
  procedure 116
  slope 1239
  special examples 1245, 1247
  superelevation 1251
Custom Toolbox 238
Custom Toolbox tool 238
custom.ksc file 889
Customizing keyboard shortcuts 903
Cut (Ctrl+X) 346
Cut Out Geospecific Texture, Batch GeoPut preference 1200
Cut Subfaces parameter 302
Cut Subfaces tool 190
  procedure 190, 196
Cutout Feature List, CAT preference 1041
Cutout files
  errors 1179
  geospecific coordinates 1163
Cylindrical gauge, creating 790

D

Database
  designing efficient 299
  designing separable 907
  displaying information about 81
  IDs 289
  saving part of 296
  shading 620
Database attributes 292
Database file
  data contained in 70
  definition of 70
Database identifiers - see “Database IDs”
Database Origin, Map panel preference 1004
Database parent 290
  clearing 291
  default 291
  invalid 291
  setting 291
Database reorganization
  parameters 301
Database traversal 299, 688
Database tree 280
Database Units 832
Database Units, Map panel preference 1004
Database window
  coordinate axes 70
  description of 70
  drawing options status 70
  managing multiple 77
  multi-pane to single-pane view 78
  splitting 83
  switch to non-orthographic view 80
DB Navigator 828
DB Origin 834
Decal texture algorithm
  MultiTexture 470
DED Builder
  DED file cells 1220
  editing DED headers 1224, 1225
  importing and converting elevation files 1221
  overview 1219
DED file
  Adding post data to an existing DED file 1216
  converting from ASCII or binary files 1067
  converting from image files 1068
  creating from OpenFlight 1215

Overwriting existing DED information 1215
DED file, saving 1006
DED files
from OpenFlight databases 1214
DED headers 1225
ded_file_name.prefs file (Terrain window settings) 891
ded_file_name1.bat file (terrain Batch settings) 891
ded_file_name1.bpf file 891
Default Color palette 413, 414
Default Database Units 62
Default Hierarchy Display Level 283
Default Material palette 393
creating 393
Default Settings, Fog Controls 268
Default Texture palette
creating 443
search rules 541
Default Texture palette file 441
default.rds file (default road setup definition) 893
default.txt file 890
Degrees of freedom - see “DOF”
Delaunay
advantages and disadvantages 1011
Delaunay Error Log, Delaunay preference 1022
Delaunay preferences
Calculate RMS Error 1022
Coastline Posts 1024
Create Quad Tree LODs 1021
Delaunay Error Log 1022
Group Boundary Posts 1024
Group Boundary Weighting 1025
Group Size in Latitude Posts (X and Y) 1020
High LOD Polygon Count 1020
Land and Water Edge Weighting 1025
Land Posts Color Threshold 1020
LOD Number 1025
Maximum Faces/Group 1025
Preserve Coastline 1025
Quad Tree LOD Boundary Posts 1024
Rect. High Res 1022
Ridge and Valley Detection 1023
Ridge and Valley Tolerance 1023
Terrain Accuracy Tolerance 1026
Use Post Weighting Multipliers 1024
Use Post Weighting Parameters 1018
Z Scale Multiplier 1020
Delaunay terrain conversion
overview 1018
preferences 1019
quad tree LODs 1026
Delete 346
Delete Contour 950, 994
Delete Last, Construction Tool parameter 1236
Delete Level tool 229
Delete Light Source 603
Delete LOD 956
Delete LOD Rules Definition preference 1155
Delete texture layers 477
Delete, GeoFeature Action List preference 1148
Delete, switch attribute 864
Deleting road segments 1234, 1236
Delta, Replicate tool parameters 162
Descendant 280
Deselect All, Batch preference 1052
Deselect All, Select menu 333
Design tips for terrain 956
Detach tool 230
Detail texture 530
Detail texture settings
Assignment for related detail textures 537
Control values for sharpen/detail texture 535
Mipmap 8 x 8 filter kernal 534
Sub-tiling of texture image 539
DFAD
converting for import 1091
converting to DFD 1091
supported levels for import 1091
DFAD Feature ID, face attribute 847
DFAD Feature ID,mesh attribute 859
dfad.prefs file (terrain feature preferences) 891
dfadbat1.prefs file (Rules and Actions) 892
DFD
definition of 1091
DFD construction layer - see Feature layer
Diagnosis, BSP 927
Dial gauge, creating 770
Diffuse
Modify Light Source parameters 609
Modify Material parameters 392
Dimension control 312
Dimmer tool 204
Direction parameters, Light Points tool 639, 646
Directivity 1105

Disable Edge Matching 942
Disable LOD Boundary Wall, Polymesh preference 1014
Display box, in attribute windows 829
Display Planes 926
Display Update Rate 319
Display Users 36
Displaying light sources in the Graphics view 600
Displaying sound nodes in the Graphics view 673
Displaying texture in the Graphics view 521, 523
Distance between features 1270
Divide Quads tool 194
Dividing a texture 464
DMA DTED file conversion 1207
Document External Resources 832
Document Externals 376
DOF 735
  assigning transformations 739
  clearing limits 745
  constraining 744
  creating 736, 737
  displaying in the Graphics view 737
  freezing limits 745
  positioning DOF axes 737
  previewing multiple nodes 745
  selecting in the Graphics view 737
  setting limits 739
DOF attributes 379, 865
  Constrain 866, 867, 868
  Current 866, 867, 868
  Maximum 866, 867, 868
  Membrane Mode 867
  Minimum 866, 867, 868
  Repeat Texture 867
  Step 866, 867, 868
DOF Mode tool 235
DOF Viewer 745
Down, LOD Rules Definition preference 1155
Drag-and-drop editing 287
  avoiding 285
Draped, Adaptive scope 1046
Draw (Modify Delaunay) 970
Draw Bounding Box Nintendo preference 65
Draw Bounding Volumes 319, 695
Draw Clip Nodes 319
Draw Clip Nodes, View control 807
Draw Constructions 319
Draw Control Points 318
Draw DOF Axes 316, 737
Draw Edges Nintendo preference 65
Draw Indirect Image 971
Draw Light Sources 600, 619
Draw Lights Large 317, 661
Draw Mode
  State preference 321
Draw Mode (D) 321
Draw New Textured Polygons White 833
Draw NewTextured Polygons White 530
Draw Normal 272
Draw Order (Z) 682, 684
  See also “Drawing order”
  State preference 683
  Z-buffer 686
Draw Orders Used 683
Draw Polygons Nintendo preference 65
Draw Select Outline 319, 329
Draw Sound control 673
Draw Sound preference 673
Draw Texture (T) 521
Draw tools, Texture Editor
  Bucket Fill 562
  Draw Closed Polygon 560
  Draw Closed Shape 561
  Draw Ellipse 560
  Draw Filled Ellipse 560
  Draw Filled Polygon 560
  Draw Filled Rectangle 560
  Draw Filled Shape 561
  Draw Line 559
  Draw Polygon 560
  Draw Rectangle 560
  Draw Shape 561
  Pen 559
  Rubber Stamp 561
Draw Vertex Large 317
Draw Vertex Normals 316, 626, 661
Draw Vertex Numbers 318
Draw Wire Backface 317
Draw Wireframe 273
  Modify Delaunay window 970
Drawing and culling terrain 957, 1057
Drawing order 682
  BSP 912
  combining BSP and Fixed List 908

Z-buffer 686
Drawing Order (Z)
BSP 686
Fixed List 685
Drive Road parameters
Horizontal/Vertical Offset 1276
Infinite Loop Drive 1276
Speed 1276
Driving a road
driving speed 1276
Infinite Loop Drive 1276
overview 1275
starting the drive simulation 1276
stopping 1276
Duplicate tool (Shift+D) 160
Duplicate tools
Duplicate
copying an instance 370
Replicate 162
Dynamic text 814
creating 815
Dynamic Texture control 523
Dynamic Texture Default 543
Dynamic Update, Fog Controls 268
Dynamic Wireframe 255, 318
Dynamically updating materials 391

E

Earth Ellipsoid Model, Map panel preference 1005
Edge
defined 295
referencing 356
selecting 328
Edge From Mouse tool 209
Edge Mode tool 234
Edge tools
Centerline 216
Construction Curve 218
Edge From Mouse 209
Intersection of Two Planes 217
Parallel to Edge 210
Perpendicular to Edge 212
Perpendicular to Trackplane 214
Edge Turn tool 191
Edge Z Value Basis, Road Bed parameter 1258
Edit Light Source 604
Edit menu
Clear Construction 220
Combine Faces 189
Copy (Ctrl+C) 346
Cut (Ctrl+X) 346
Cut Subfaces 190, 196
Delete 346
Insert Color (Ctrl+R) 409
Insert Material 395
Insert Texture 461
Match Vertices 202
Paste (Ctrl+V) 347
Remove Material 395
Remove Texture 462
Replace Graphics 347
Texture From Face 462
Triangulate 188
Undo (Ctrl+Z) 244
Undo History 244
View Graphics Buffer 345
Edit menu, Color palette
Edit Color Name 403
Make HLS Map 413
Edit menu, Light Source palette
Delete Light Source 603
New Light Source 602
Edit menu, Material palette
Edit Material 391
Edit menu, Sound palette
Delete All 671
Edit Sound 671
Edit menu, Texture Editor
Modify Alpha 555
Redo 555
Undo 555
Edit menu, Texture Mapping palette
Modify Mapping 521
Edit menu, Texture palette
Blank Pattern 426
Chop Texture 464
Clear All Palettes 430
Clear Palette 430
Delete Pattern
removing a pattern 429
Edit Pattern 426, 545
Modify Attributes 429
Modify Mapping 488
Move Pattern 428
Reload 427

Remove Unused 430
Replace Pattern 427
Edit Sound 671
Edit, GeoFeature Action List preference 1148
Editing preferences
Auto Apply Current Material 394
Loft Faces in Order Selected 115
Match Vertices Tolerance 202
Max Undos 244
Shade All if Nothing Selected 619, 621
Slice Displayed Nodes Only 175
Editing road segments 1234
Eight Point Warp Texture tool 488
Elev (Modify Delaunay) 970
Elevation
Modify Light Source parameters 608
Modify Vertex Normals parameters 630
Elevation data
conversion utilities 940
converting float data to DED 1089
converting NIMA DTED to DED 1087
converting USGS to DED 1088
description of 1063
post values 1063
reading DTED from CD-ROM 1085
sources 1085
Elevation Scales 951
Ellipsoid Model 834
Ellipsoid projection 1008, 1061
e-mail
MultiGen-Paradigm, Inc. 3
Emissive
Modify Material parameters 393
Enable Area Block Processing, Batch preference 1050
Enable Feature Processing 1054
Enable Switch Nodes 63
Enabled 616
Enforce Polygon Limits, GeoFeature Action Definition preference 1153
Enforce Polygon Limits, LOD Rules Definition preference 1155
Entry control point 1235
Entry slope, in special curves 1247
Entry Slope, Vertical Curve control 1240
Entry Spiral and Exit Spiral 1251
Environment Map Texture tool 459
Environment variables
about 43
GVIEWDFADPREFS 45
MPI_CREATOR_FLTEXTERNPATH 44, 372
MPI_CREATOR_FLTPATH 44
MPI_CREATOR_PLUGINDIR 43
MPI_CREATOR_RESOURCEPATH 44
MPI_CREATOR_TERPATH 44
MPI_CREATOR_TXTPATH 44, 541
setting in Windows NT 45, 46, 47
Error messages 82
Every k Minutes 246
Exit (Ctrl+Q) 35
Exit control point 1235
Exit slope, in special curves 1247
Exit Slope, Vertical Curve control 1240
Expand Hierarchy tool 231, 281
Export DFD, GeoFeature menu 1125
Export filters
AutoCAD 1229
STL 1229
VRML 1229
Export Select to DFD, GeoFeature menu 1126
Export to Realtime 616
Exporting files 1228
External Reference Mode tool 235
External References 370
and MPI_CREATOR_FLTEXTERNPATH 44
converting to instances 374
creating 372
documenting externals 376
in master files 1029
listing external references 375
manual display procedure 373
replacing external references 376
searching 375
selecting nodes 373
Extruding faces 96, 98, 103, 121
Extrusion tool 121
Eyepoint
as center of rotation 257
changing position 252
coordinates 252
current position 251
default 261, 262
defined 251
panning 255

rotating 255, 257
zooming 254
Eyepoint controls
Eyepoint clocks 76, 257
Eyepoint coordinates 77, 252
Eyepoint menu 76, 252
Pan View 72, 255
Rotate View 255
Zoom View 73, 254
Eyepoint coordinates 77, 252
Eyepoint LOD 715
Eyepoint menu 76, 252
numeric keypad, using 254
opening 252
positioning choices 252
recalling positions 254
Save 253

F

FAC - see “Trackplane From Face”
Face attributes
Alternate Color Index 842
Alternate Color Intensity 842
Alternate Color Name 842
Alternate Color RGB 842
Color Name 402
Cultural Footprint 848
DFAD Feature ID 847
Hidden Polygon 846
Intensity 407
IR Color 847
IR Material 416
Line Style 845
Line style index 823
LOD Control - Preserve Face 847
Material Attributes - Index 843, 855
Material ID 847
Primary Color Index 841
Primary Color Intensity 842
Primary Color Name 842
Primary Color RGB 842
Relative Priority 846
Render Both Sides Visible 381, 846
Render Textured Polygons White 846
Roofline 848
Shade 625, 845
SmartScene Palette 848
Surface Material Code 848
Terrain Flag 848
Texture Attributes - Base 843
Texture Attributes - Detail 843
Texture Attributes - Mapping 843
Transparency 845
Type 845
Face attributes, in roads 1291
Face Colors From 996
Face Creation tools
Circle 91
Circle Plus 92
Polygon (Shift+I) 85
Rectangle 86, 92
Rectangle Plus 87
Face Mode tool (Shift+F) 234
Face node 294
Far Switch Distance, Adaptive attribute 1046
Feature Action, GeoFeature menu 1146
Feature construction
Build Polys From Image 1106
Scatter Points in Areals 1118
Feature data
converting data for import 1091
creating a linear feature 1104
creating a new layer 1099
creating a point feature 1100
creating an areal feature 1102
editing 1115
importing 1094
projecting 1118, 1127
thinning and reducing 1109
Feature Data Reduction Parameters 1095
Reduce Areals 1098
Reduce Linears 1098
Reduction Tolerance 1098
Feature Data Retrieval Parameters 1094, 1096
Use Feature Data Extents 1097
Feature Database, Feature parameter (Road) 1269
Feature layer 1096
Feature Panel Preferences 1053
Feature parameters (Road)
Backup Distance 1270
Centerline Offset 1270
Clip at Road Poly 1271
Continuous 1271
Feature Database 1269
Feature Spacing 1270

Geometry Type 1269
Heading 1271
Keep Vertical 1272
Place in Object 1271
Project on Road 1271
Spacing Type 1269
Use Edge Z 1272
Feature Preferences
Apply GeoFeature Texture 1139, 1140
dfad.prefs file 891
Library Substitution 1140
Light Strings 1137
mgprop.dat file 892
Projection Preferences 1134, 1136, 1138
setting up 1131
Transferring between Feature Types 1142
Valid Surface Material Codes 1132, 1133
Feature Preferences file, Project panel preferences 1001
Feature projection
conflict checking in TCT conversion 1034
conflicts with bridge features 1179
global projection preferences 1144
projecting features without terrain 1179
Feature Search, GeoFeature menu 1123
Feature Selection Parameters
Select All Features 1097
Select By Attributes 1097
Show Feature Summary 1097
Feature Spacing 1270
Along Centerline 1270
Along Feature Offset Path 1269
Feature parameters (Road) 1270
Feature Summary 1098
Feature Type 1107
creating a new Feature Type 1108
selecting for preference definition 1132
Feature types, in road generation 1269
Repeating Geometry 1269
Repeating XREF 1269
Single geometry 1269
Single XREF 1269
Features
sequence of tasks for applying 1127
Features (Road)
creating 1267
definition 1267
Fence Touch 260
Fence Touch On 73
Fence Touching 328, 340
Fence Within 73
Fence, selecting with 327
FID 645
use in MultiGen 1107
Field of view (FOV) 263
File menu
Close (Ctrl+W) 248
Convert Externals 374
Exit (Ctrl+Q) 35
Export 1228
External Reference 372
Import 1226
New (Ctrl+N) 244
Open (Ctrl+O) 242
Recent Files 243
Reread Externals 374
Save (Ctrl+S) 244
Save as 247
Write Select 296, 298
File menu, Light Source palette
loading 617
saving 617
saving a default palette 618
File menu, Sound palette
Read Sound 670
Save Palette 672
File menu, Texture Editor
Close 554
Resize 553
Revert 554
Save 553
Save As 553
File menu, Texture palette
Load Palette 442
Read Pattern 424
Find Morph Vertices 717
Fit on Grid (Ctrl+Shift+V) 74, 261
Fit on Pick (Ctrl+F) 73, 260
Fit on Select (V) 73, 261, 333
Fit on Select, Material palette 389, 601
Fixed Elevation contour 951, 995
Fixed List
about 685
combining with BSP 908
Flags, mesh attribute 861

Flat Earth projection 1005, 1060
Flat Earth terrain projection 1006
Flat roof when projected 1138
Flat Shaded
  Modify Delaunay window 970
  object attribute 839
Flat shading model 623, 624
Flight preferences
  Allow Default Parent 63, 226, 291
  Auto Save 61, 247
  Auto Save Checkpoints 247
  Auto Save Confirmation Prompt 61, 247
  Auto Save Every k Minutes 61, 246
  Check Relative Priority When Shifting Nodes Left or Right 62, 230, 231, 288
  Default Database Units 62
  Default Hierarchy Display Level 283
  Enable Switch Nodes 63
  Hierarchy Display Level 62
  Intensities for RGB Matching 64, 621
  Keep Backup Files 61, 247
  Modulate Current Color with Material in Material Palette 62, 387, 394
  Read External References 63
  Update Database Latitude and Longitude 63
  Use Attribute Index Matching 64
Flight preferences panel 60
Flip Book tool 128
  creating animations 128
Flipbook Animation 835
Float2ded conversion utility 1089, 1210
flt1.materials file 890
Focus, setting 895
Fog Controls 267
  Color 268
  Default Settings 268
  Dynamic Update 268
  Fog On 268
  Revert 268
  Set Preference 268
  Update All Graphics Views 268
  Update Now 268
  Visibility Range 267
Fog On, Fog Controls 268
Fog On/Off, View controls 317
Font editor 807
  placing 2D text 812
Force Main Palette Into Subordinate Files 881
Format Origin 832
Format Version 832
Fractal Texture 594
Frame Rate 681
Freeze Bounding Volume 837
Freeze flags 77
Freeze flags, Texture Editor 578
Freeze, vertex attribute 850
Freezing a DOF 745
Front Bottom Left and Back Top Right 838
Full View 273
Full, Batch GeoPut preference 1201

G

Gaming Area 941
Gaming area, Terrain window 990
Gauges
  cylindrical 790
  dial 770
  vertical 784
Generate LOD 720
Generate Only Vertical Planes 914
Generate Report 872
Generic nodes 869
Geocentric projection 1005, 1060
Geocentric terrain projection 1007, 1008
Geodetic projection 1060
GeoFeature action
  creating 1146
  definition 1146
GeoFeature Action Definition preferences
  Action Name 1149
  Enforce Polygon Limits 1153
  Include Bounding Feature (D1) 1151
  Max Feature Polygons Per Group 1153
  Projection 1149
  Raise Flat Projected Features 1151
  Reduce Areal Features 1153
  Reduce Feature Vertex Count 1153
  Reduce Linear Features 1153
  Reduction Tolerance 1153
GeoFeature Action List preferences
  Create 1147
  Delete 1148
  Edit 1148
GeoFeature menu
  Export/Export DFD 1125

Export/Export Select to DFD 1126
Feature Construction/Create Areal Feature 1102
Feature Construction/Create Linear Feature 1104
Feature Construction/Create Point Feature 1101
Feature Construction/New Feature Layer 1099
Feature Construction/Reduce Feature Vertices 1119, 1121
Feature Preferences 1108, 1131
Feature Projection
High Res Library Sub 1158
Project and Cut 1157
Project and Cut Low Z 1157
Project Feature Flat 1158
Project Feature Flat and Cut 1158
Project Feature Flat and Cut Low Z 1158
Project Feature(s) 1157
Project Tops Only 1158
Project Walls Only 1157
Feature Projection/High Res Library Sub 1158
Feature Projection/Low Res Library Sub 1158
Feature Projection/Project Feature(s) 1118
Feature Search 1123
Import/Convert DFAD to DFD 1091
Import/Import DFD 1094
New Feature Search 1123
Projection Preferences 1160
Rules and Actions/Apply Rules to Files 1169
Rules and Actions/Document Rules and Actions 1157
Rules and Actions/Feature Action 1146
Rules and Actions/Rule List 1154
Show Feature Data 1124
Toggle Linear Features 1106
GeoFeature preferences
Plant GeoFeature Models 1160
GeoFeature Projection preferences
Model Path 1160
Use External References 1162
GeoFeature rules
creating 1154
overview 1153
Geometry tools
3D Text 119
Billboard wizard 1293
Curve 116
Flip Book 128
Loft 115
Peak (Shift+P) 103
Revolve About Edge 106
Roof tool 133
Rounded Strip Face 111
Sphere 105
Strip Face 108
Tree wizard 1304
True Type Text 139
Wall (Shift+W) 96
Wall Plus 98
Geometry Type, Feature parameter (Road) 1269
Geometry, in defining models 241
GeoPut Texture tool 1202
Geospecific texture
applying to terrain 1196
applying with Batch GeoPut 1198
effect of face color 1202
overview 1191
setting geospecific information 1192
Get Color From Face tool (Ctrl+Shift+R) 410
Get Face Color From 948
Get IR Color From Face tool 418
Get Line Style From Face, Properties toolbox 206
Get Material From Face tool 395
Get point (Modify Delaunay) 971
Get Texture From Face tool (Ctrl+Shift+T) 462
Get Texture From Face tool (Ctrl-Shift+T) 462
Global - Lights All 616
Global DFAD preferences 1162
Global projection preferences 1144
Gouraud Shade (Modify Delaunay) 970
Gouraud shading mode 620, 621
face colors 625
Gouraud Shading, State preference 621
Graphic View Defaults 261
Graphics (S) 78
Graphics view
selection outline 329
View controls 315
Grid
default settings 304
description of 304
Dimension 325
Grid controls, see "'View controls"
Grid Larger 307, 322
Grid Mode tool 237
Grid On/Off (G) 307, 322

Grid preferences 304
displaying 304
Draw Grid 305
Grid 304
Grid Spacing 305
Snap to Grid 305
Grid Smaller 308, 323
Grid Snap 307, 322
Grid Snap, Grid controls 1247
Grid Spacing Axes 312, 325
Grid Types 311, 324
Grid, area block 1050
Group attributes 835
Calculate 837
Calculate Plateaus From 838
Center 837
Flipbook Animation 835
Freeze Bounding Volume 837
Front Bottom Left and Back Top Right 838
Height 838
Layer 836
Number of Increments 838
Orientation 838
Radius 838
Relative Priority 836
Replication Count 835
Significance 836
Special 1 and 2 836
Type 837
Group Boundary Posts, Delaunay preference 1024
Group Boundary Weighting, Delaunay preference 1025
Group Mode tool (Shift+G) 233
Group node 293
Group Objects, Polymesh preference 1017
Group Size in Latitude Posts (X and Y), Delaunay preference 1020
Group size in terrain 944
effect on quad tree LODs 952, 1026
GVIEWDFADPREFS variable 45

H

Hard Edge, vertex attribute 850
Has Normal, vertex attribute 850
Header attributes
Database Units 832
DB Origin 834
Document External Resources 832
Draw New Textured Polygons White 833
Ellipsoid Model 834
Format Origin 832
Format Version 832
Lambert 834
Last ID 833
Local Origin Offset 835
Map Projection 834
Revision Date 832
Revision Number 832
RGB Mode 833
Save Vertex Normals 625
SW and NE Corner 834
SW Corner 834
Heading, Feature parameter (Road) 1271
Height, group attribute 838
Help
about online help 39
context-sensitive 40
tool tips 40
Help menu
About Creator 40
Contents 40
On Plug-Ins 40
On Version 35
Shortcuts 40
Helper Mode tool 237
Hertz 682
Hidden Faces 272, 846, 857
Hidden Faces submenu - see “View menu, Hidden Faces”
Hidden Polygon, face attribute 846
Hidden Polygon, mesh attribute 857
Hidden, Road Lane parameter 1285
Hide 272
Hierarchy
Database header 292
database header in 280
default 280
designing efficient 299
editing 287
gauge 781
generic nodes 869
in defining models 241
LOD 709
Hierarchy (S) 78, 281
Hierarchy Display Level 62
Hierarchy tools 229

Attach 230
Collect 232
Contract Hierarchy 232
Delete Level 229
Detach 230
Expand Hierarchy 231
Insert Level Above 229
Reorganize Hierarchy 231, 300
Shift Far Left 230
Shift Far Right 231
Shift Left 230
Shift Right 230
Toggle Display 231
Hierarchy view
changing to 281
default 283
expanding and collapsing 281
selecting in 284, 327
High LOD Polygon Count
Delaunay preference 1020
Polymesh preference 1013
High Res Library Sub 1158
high resolution insets
detail texture 530
High resolution library substitution 1158
Highlights
turning off 319, 329
Hill road sections 1236
Hills, in road generation 1249
creating 1247
defining entry slope 1240
defining exit slope 1240
definition 1236, 1239, 1244, 1247
slope 1239
special examples 1249
HLS color palette 413
Horizontal Angle, Preview Tessellation control 1243
Horizontal Break Angle
Road Bed parameter 1259
Road LOD parameter 1259, 1263
Scenario parameter 1282
Horizontal Curve controls
Arc Radius 1252
Construction Tool parameter 1236
Entry Spiral and Exit Spiral 1251
Superelevation 1251
Horizontal Lobe Angle 647, 654
Horizontal Offset, Drive Road parameter 1276
horizontal texture offset 442
HPR Angle 1285

I

IDH_texture_compositor 587
Image editors
launching 426
setting up
on Windows 545, 547
Texture Editor 548
Image files
converting to DED 1068
Import Elevation, Terrain menu 989
Import filters
3D Studio 1226
AutoCAD 1228
OBJ 1228
STL 1228
Importing feature data 1094
Importing files 1226
Importing, and color matching 415
Include Bounding Feature (D1)
GeoFeature Action Definition preference 1151
Include Lat/Long in Filename 1092
Index 676
Modify Light Source parameters 606
Indexed Polys, mesh attribute 860
Indirect Texture 948, 996
applying to terrain 1187
associating properties with RGBA values 1189
creating an indirect texture image 1186
required files 1185
terrain 996
Texture panel preference 1048
Infinite light source 599, 607
Infinite Loop Drive, Drive Road parameter 1276
Infinite/Local/Spot light source options 607
Info menu
Display Users 36
Keyboard Shortcuts 903
License 36
List Textures 431
Preferences (Ctrl+Shift+P) 49, 304, 889
Statistics (Ctrl+?) 81
Status Log 82, 247
Info menu, Texture palette 431
Statistics 434

Infrared Color display control 416
Infrared View 316, 415
Inhibit Display, object attribute 840
Input Values, DOF 740
Insert Action Arrow, LOD Rules Definition preference 1155
Insert Color tool (Ctrl+R) 409
Insert Contour 950, 994
Insert IR Color tool 417, 418
Insert Level Above tool 229
Insert Line Style, Properties palette 206
Insert LOD 956
Insert Material tool 395
Insert Material, Edit menu 395
Insert Texture tool (Ctrl+T) 461
Insert Transformation Matrix tool 157, 158
Insert Waveform 674
Instances 368
  changing into a geometry 370
  converting to geometry 370
  creating 369
  duplicating 370
  translating 369, 370
Instruments
  Cylindrical Gauge 790
  Font Editor 807
  Vertical Gauge 784
Instruments menu
  Cylindrical Gauge 790
  Dial Gauge 770
  Font Editor 807
  Place Text 812
  Set Clipping Polygon 802
  Vertical Gauge 784
Intensities for RGB Matching 64, 415, 621
  and Keep Within Color Band 415
Intensity texture pattern 421
Intensity, Face attribute 407
Intensity-alpha texture pattern 421
Interpenetrating hulls 928
Interpenetration Tolerance 914
Interpolate Elevation 950, 995
Interpolate Property 951
Intersection of Two Edges tool 222
Intersection of Two Planes tool 217
Intersection testing - see “Collision detection”
Interval Between Two Vertices tool 223
Invert, Switch attribute 864
IR Code 416
IR Color, face attribute 847
IR Color, mesh attribute 858
IR Material, Face attribute 416
IR palette
  Black Hot/White Hot 417
  Current IR Color 416
  IR Code 416
Irregular Mesh 1014
Isolate
  in Select menu 273, 332
  Live button in window of 332

K

Keep Backup Files 61, 247
Keep Vertical, Feature parameter (Road) 1272
Keyboard shortcuts 38
  creator.ksc file 889
  custom.ksc file 889
  customizing 903

L

Lambert 834
Lambert Conic Conformal projection 1060
Lambert Conic Conformal terrain projectionMap projection
  Geocentric 1007
Lambert Latitude Lines, Map panel preference 1005
Land and Water Edge Weighting, Delaunay preference 1025
Land Posts Color Threshold, Delaunay preference 1020
Lanes
  adding 1285
  Centerline Offset 1285
  centerline reference 1277
  defining 1284
  hiding centerline 1285
  in the Graphics view 1283
  No Passing 1285
  Orientation Type 1285
  overview 1283
  Speed Limit 1285
  writing lane files 1286
Last ID 833
Lat/Long to X/Y, Terrain menu 1058
Lattice Deform tool 177
  parameters 178

procedure 177
Launching Creator
from a project file 988
Layer 836
Least Detail (Ctrl+Shift+L) 715
Length Offset, Road LOD parameter 1265
Less Detail (Ctrl+L) 715
Level of Detail
child edges, CAT 1038
considerations in road tessellation 1275
in CAT terrain 1037
in roads 1259
lofting road sections 1263
morphing in CAT terrain 1037
popping, in terrain 1035
road surface texture 1264
roads, switch in distance 1262
side area color 1265
side area material properties 1265
side area texture 1265
side areas 1262
specifying LOD method 1261
specifying material properties on roads 1264
specifying road color 1264
switching controls 998, 1037
tiling in terrain generation 1035
Walls, in terrain 1036
Level of Detail - see "LOD"
Level, hierarchy
defined 280
Levels of Detail
In TCT conversion 1034
Library Model Format 1142
Library substitution
Feature Preferences 1140
Texture on planted models 1163
Library substitution preferences 1144
License
about 35
different configurations 36
in Info menu 36
licensing
telephone number for 3
Light Activated for Modeling
Modify Light Source parameters 607
Light Grid 636
Light Grid Max. Deviation 637
Light Mode tool 235
Light Name
Modify Light Source parameters 606
Light Point DFAD preferences
Intensity 652
Maximum Defocus 652
Minimum Defocus 652
Proximity Occluding 652
Quality 652
Significance 652
Light Point file preferences
setting file save preferences 247
Light Point Mode tool 236, 660, 661
Light Point palette
Animation attributes
non-Vega Prime 654
Vega Prime 656
Appearance attributes
non-Vega Prime 644
Vega Prime 652
Creating a Sequence animation 657
creating palette entries 643
editing palette entries 644
opening palette files 642
overview 640
using 641
Light Point parameters
Color parameters
Light Ruler 637
Direction parameters
Ambient Intensity 647, 648
Horizontal Lobe Angle 647, 654
Lobe Falloff 647, 653
Lobe Roll Angle 647
Vertical Lobe Angle 647, 654
Display mode
Day/Dusk/Night 645
FID 645
Randomize Intensity Off/Low/Med/High 651
Raster 651
Raster or Calligraphic 651
SMC 645
Terrain Conforming 645

Light Point Controls
Density per sq. KM 637
Distance Between Lights 637
Light Grid Max. Deviation 637
Number of Lights 637
Light Type
Light Grid 636
Light Points 635
Random Circle 636
Random Polygon 636
Random Rectangle 635
String w/Count 635
String w/Delta 635
String w/Delta and Count 635
Performer parameters
Actual Size 650, 653
String Type
Curved String 636
Spiral String 636
Straight String 636
Light Point System
attributes 662
Light Point System tool 227
Light Point tool
overview 631
parameters 635
Light Point/String attributes
Light Point palette 640
Light Points
attributes 661
creating 631
Light point palettes, mgprop.dat file 892
Light Point System node 631
overview 1362
Vega Lightpoint Backcolor 851
Light Ruler 637
Light source attributes
Enabled 616
Export to Realtime 616
Global - Lights All 616
ID 616
Orientation 616
Palette Index 616
Position 616
Light source nodes
creating 612
displaying in the Graphics view 613
enabling/disabling 616
overview 612
positioning 613
viewing and modifying attributes 615
Light Source palette 600, 604
copying a light source 618
Delete Light Source 603
Edit Light Source 604
Index 602
loading 617
mg1.lts file 890
New Light Source 602
saving 617
saving default palette 618
selecting a light source 600
Light Source Size 620
Light sources
infinite 599, 607
local 599, 607
options 603
specifying type 607
spot 599
Light Strings, Feature Preferences 1137
Lighting
constant attenuation factor 610
light source attributes 615
light source nodes 612
linear attenuation factor 610
modeling lights 611
positioning light source nodes 613
quadratic attenuation factor 610
Lighting On, State preference 619
Lighting preferences 619
Lighting Vector Control 606
Lights On/Off 316, 600, 622
Line style
modifying 822
Line Style From Face, Properties toolbox 206
Line style index, face attribute 823
Line Style palette
deleting a line style 823
using Copy and Paste 822
Line Style palette, Palettes menu 821
Line style, definition of 821
Line Style, face attribute 845
Line Style, mesh attribute 857

Linear Bend tool
overview 173
procedure 173
Linear feature
creating 1104
Lit Gouraud shading model 460, 623
Lit shading 620
Lit shading model 623
Live button, isolate window 332
Load
Light Source palette 617
Load Default
Material palette 394
Load Texture palette 442
search rules 541
Loading sounds into database palette 670
Lobe Falloff 647, 653
Lobe Roll Angle 647
Local Coordinate Systems (LCS)
adding geometries to an LCS 367
overview 365
procedure 366, 367
removing 367
Local light source 599, 607
Local Origin Offset 835
Local-DOF menu
Constrain DOF 744
DOF Viewer 745, 746
Freeze DOF 745
Make Geometry 370
Make Local 367
Position DOF 738
Set DOF Limits 739
Set Global 367
Set Local 366
Set Local From Path 367
Un-Constrain DOF 745
LOD
adding to terrain 956
automatic generation of 720
creating for terrain 952
deleting for terrain 956
Modify Delaunay preference 970
morphing 716
preventing popping in terrain 959
quad tree 952
see Level of Detail
setting terrain switch ranges 955
slant range, defined 713
switching 713
tips for terrain ranges 958
LOD attributes 712
Additive LODs Below 714
LOD Center 714
Significant Size 713
Special 1 and 2 714
Switch In 713
Switch Out 713
Transition 713
Use Previous Range 713
LOD Center, LOD attribute 714
LOD Control - Preserve Face 847, 859
LOD hierarchy 709
Additive LODs 711
Independent LODs 710
Nested LODs 710
LOD menu
Auto LOD 715
Eyepoint LOD 715
Find Morph Vertices 717
Generate LOD 720
Least Detail (Ctrl+Shift+L) 715
Less Detail (Ctrl+L) 715
More Detail (Ctrl+M) 714
Most Detail (Ctrl+Shift+M) 714
Place Morph Vertices 718
Remove Morph Vertices 720
Set LOD Distance 715
Show Morph Vertices 719
LOD Method
Distance From Center of Bounding Volume 1261
effect on road attributes 1290
Road LOD parameter 1261
LOD Mode tool 234
LOD Number, Delaunay preference 1025
LOD Range, Road LOD parameter 1262
LOD Rule Name, LOD Rules Definition preference 1155
LOD Rules Definition preferences
Delete 1155
Down 1155
Enforce Polygon Limits 1155
Insert Action Arrow 1155
LOD Rule Name 1155

Up 1155
Loft Faces in Order Selected 115
Loft tool
example 115
overview 115
Lofting a road 1265, 1266, 1282
Loop Forward 759, 761
Loop Forwards and Backwards 759, 761
Low Res Library Sub 1158
Low resolution library substitution 1158
Lower 3D Tops Below Wall Height 1140

M

Magnification filter 529
mailing address
MultiGen-Paradigm, Inc. 3
Make Geometry, Local-DOF menu 370
Make HLS Map 413
Make Local 367
Make Vectors Equal 629
Making objects from faces 96, 98, 103
Maneuver tools
Insert Transformation Matrix 157
Put 154
Rotate - Scale to Point 151
Rotate About Edge 148
Rotate About Point (Shift+R) 146
Scale (Shift+S) 149
Translate (Shift+T) 144
Manuscript Extents To Process 1091
Map panel
choosing the appropriate settings 1006
Map panel preferences 1003
Map Projection 834
Map projection
Flat Earth 1006
Geocentric 1007
Lambert Conic Conformal 1007
Trapezoidal 1008
UTM 1006
Map Projection, Map panel preference 1004
Map Texture tools
4-Point Put Texture 450
Radial Project Texture 457
Spherical Project Texture 456
Surface Project Texture 455
Mapping a geospecific texture 1202
Mapping texels to pixels 527
Mask Count 863
Mask Name 863
Master file
area block processing 1177
in terrain batch processing 1029
Match Vertices Tolerance 202
Match Vertices tool 202
procedure 203
setting tolerance preference 202
Material
reflection 392
Road LOD parameter 1264
side area property 1265
Material and feature projection 1135
Material Attributes - Index, face attribute 843
Material Attributes - Index, mesh attribute 855
Material Elevation Scale 951
Material ID, face attribute 847
Material ID,mesh attribute 859
Material palette
and MPI_CREATOR_RESOURCEPATH 44
Edit menu, Edit Material 391
File menu, Load Default 394
File menu, Save Palette 393
flt1.materials file 890
opening/closing 387
View menu, Fit on Select 389, 601
View menu, Thumbnail 389, 601
Material Palettet, Project panel preference 1002
Materials 387
applying to a face 395
color and light 393
creating a new definition 389
creating default a Material palette 393
default Material palette 393
deleting unused material definitions 390
glare 393
modifying a definition 390
naming 391
reflection 392
removing 395
selecting 388
shininess 391, 392
transparency 392
Max Feature Polygons Per Group, GeoFeature Action Definition preference 1153
Max Undos 244

Maximum Faces/Group, Delaunay preference 1025
Maximum Poly Count, CAT preference 1040
Maximum Polygons Per Group 944
  effect on quad tree LODs 952, 1026
Maximum Slope, Vertical Curve control 1239
Maximum, DOF attribute 866, 867, 868
Membrane Mode 867
Menu commands, in Creator documentation 38
Mesh
  Decomposing 385
  Editing 385
  Generating 384
Mesh attributes
  Alternate Color Index 854
  Alternate Color Intensity 854
  Alternate Color Name 854
  Alternate Color RGB 854
  Cultural Footprint 859
  DFAD Feature ID 859
  Flags 861
  Hidden Polygon 857
  Indexed Polys 860
  IR Color 858
  Line Style 857
  LOD Control - Preserve Face 859
  Material ID 859
  Number of Vertices 860
  Primary Color Index 853
  Primary Color Intensity 853
  Primary Color Name 853
  Primary Color RGB 853
  Quad Strips 860
  Relative Priority 858
  Render Both Sides Visible 857
  Render Textured Polygons White 857
  Roofline 859
  Shade 856
  SmartScene Palette 859
  Surface Material Code 859
  Terrain Flag 859
  Texture Attributes - Base 854
  Texture Attributes - Detail 854
  Texture Attributes - Mapping 854
  Transparency 857
  Triangle Fans 860
  Triangle Strips 860
  Type 856
  UV Layers 861
Mesh Mode tool 237
Mesh nodes 1016
mg1.lts file 890
mgprop.dat file (light point palettes) 892
Minification filter 528
Minimum Length, Vertical Curve control 1242
Minimum, DOF attribute 866, 867, 868
Minutes/posts, area block size 1176
Mipmaps 958
Mirror tool 170
Missing elements in the Graphics view 263
Mode Select List 73, 357
Mode tools
  BSP Mode 236
  Curve Mode 236
  DOF Mode 235
  Edge Mode 234
  External Reference Mode 235
  Face Mode (Shift+F) 234
  Group Mode (Shift+G) 233
  Light Mode 235
  Light Point Mode 236, 660, 661
  LOD Mode 234
  Object Mode (Shift+O) 234
  overview 233
  Sound Mode 235, 674
  Switch Mode 235
  Vertex Mode 234
Model Path, GeoFeature Projection preference 1160
Modeling lights
  activating 607
  definition of 611
Modeling mode
  changing 357
  Edge Mode 234
  Face Mode 234
  Group Mode 233
  Mode Select List 357
  Object Mode 234
  use in selecting 327
  Vertex Mode 234
Modeling with background images 363
Modify
  face attributes 625
  light source definition 604
Modify All 827

Modify Attributes (Ctrl+=) 826
Modify Color Name window 404
Modify Delaunay 960
  overview 1057
Modify Delaunay window 971
Modify Face tools
  Add Vertex 186
  Combine Faces 189
  Cut Subfaces 190
  Divide Quads 194
  Edge Turn 191
  Orient Faces 196
  Reverse Face 184
  Split Face 185
  Tessellate 193
  Triangulate 188
  T-Vertex Killer 197
Modify Geometry tools
  Bend Deform 178
  Bulldoze 182
  Cookie Cutter 176
  Lattice Deform 177
  Linear Bend 173
  Mirror 170
  Plant 173
  Project 172
  Slice 174
  Taper Deform 181
  Twist Deform 180
Modify Light Source
  Ambient 609
  Attenuation 610
  Azimuth 608
  Diffuse 609
  Elevation 608
  Index 606
  Light Activated for Modeling 607
  Light Name 606
  Lighting Vector Control 606
  Options menu in dialog box 603
  Specular 609
  Spot Cutoff Angle 609
  Spot Dropoff Exponent 610
Modify Mapping in Texture Mapping palette 521
Modify Mapping tool 488, 521
Modify Material
  Alpha 392
  Alpha property 391
  Ambient 392
  Diffuse 392
  Emissive 393
  Material Name 391
  overview 390
  Shininess 391, 392
  Specular 393
  Update 391
Modify Texture tools 481
  aligning vertices 495
  Eight Point Warp Texture 488
  Modify Mapping 488, 521
  Modify UVs tool 490
  moving vertices 493
  Rotate Texture 484
  rotating vertices 494
  Scale Texture 486
  scaling vertices 493
  Texture Composer 499
  Translate Texture tool 481
Modify UVs 490
  aligning vertices 495
  moving vertices 493
  rotating vertices 494
  scaling vertices 493
Modify Vertex Normals
  Make Vectors Equal 629
Modify Vertex tool (Shift+M) 200
Modify Vertex tools
  Dimmer 204
  Match Vertices 202
  Modify Vertex (Shift+M) 200
  Modify Vertex Normals 627, 629
Modifying a line style 822
Modifying a road 1253
Modulate Current Color with Material in Material Palette 62, 387, 394
Modulate texture algorithm
  MultiTexture 470
More Detail (Ctrl+M) 714
Morphing CAT levels of detail 1037
Morphing LODs 716
Most Detail (Ctrl+Shift+M) 714
Mountains
  emphasizing z scale in terrain 944
  giving preference in terrain 945

Moving items in the Graphics view 144
MPI_CREATOR_FLTEXTERNPATH 44, 372
MPI_CREATOR_FLTPATH 44
MPI_CREATOR_PLUGINDIR 43
MPI_CREATOR_RESOURCEPATH 44
MPI_CREATOR_TERPATH 44
MPI_CREATOR_TXTPATH 44, 541
MultiGen Creator
    extending 37
    starting 34
MultiGen-Paradigm, Inc.
    contact information 3
MultiTexture
    applying 470, 473
    copying texture layers 474
    Defined 470
    deleting texture layers 477
    Overview 470
    removing 462
    reordering texture layers 476
    Texture attributes 530
    Texture Environment algorithms 470
    View Control panel 470
MultiTextures
    and bump maps 471
MVF data
    definition 1091

N

Name, modifying, in roads 1254
Navigating attribute windows 828
Near Switch Distance, Adaptive attribute 1046
Nested face - see “subface”
Nested LODs - see “LOD hierarchy”
New Attribute Search, Attributes menu 870
New Database File (Ctrl+N) 71, 244
New Feature Search 1123
New Light Source 602
New, switch attribute 863
Next Section, Construction Tool parameter 1235
Nintendo preferences
    Draw Bounding Box 65
    Draw Edges 65
    Draw Polygons 65
    Wall Alpha 66
No Passing, Road Lane parameter 1285
Node
    and attributes 825
    changing point of attachment 287
    create light source 612
    creating 226
    defined 279
    face 294
    group 293
    identifiers 241
    IDs (names) 289
    object 293
    relative priority and shifting 230, 231, 288
    renaming 289
    selecting by identifier 333
    selecting in Hierarchy view 284
Node attributes
    about 825
    searching 869
    viewing and changing 826
Nodes
    CAT 1041
    Clip 228
Non-illuminated, object attribute 840
Nonplanar faces, correcting 357
Non-radiant lights 660
Northeast Latitude 1097
Northeast Longitude 1097
Number of Increments, group attribute 838
Number of Objects Per Group X (and Y), Polymesh preference 1018
Number of Vertices 1120
Number of Vertices, mesh attribute 860
Number, Replicate tool parameters 162
Number_Delta, Replicate tool parameters 163

O

OBJ
    importing files from 1228
Object attributes
    Flat Shaded 839
    Inhibit Display 840
    Non-illuminated 840
    Relative Priority 839
    Shadow 840
    Significance 840
    Special 1 and 2 840
    Transparency 839
Object Mode tool (Shift+O) 234
Object nodes 293
Offset Distance, Adaptive attribute 1046

Offset, View controls 311, 324
Omnidirectional 1101, 1106, 1112, 1113
On Version 35
Open Database File (Ctrl+O) 71, 242
OpenFlight 70
Opening new files 244
Operations tools, Texture Editor
  Flip Diagonal 580
  Flip Horizontal 579
  Flip Vertical 579, 585, 586
  Grayscale to Alpha 581
  Histogram 582
  Invert Intensity 581
  Modify Intensity 582
  Operations Target 579
  RGB to Grayscale 581
  Rotate 90 degrees 580
  Swap Quarters 580
Options menu, Texture Editor
  Dynamic Update 557
  Update Database 558
Organize Palette tool 434
Orient Faces tool 196
Orientation 1101
  group attribute 838
  light source attributes 616
Orientation Type, Road Lane parameter 1285
Origin, Adaptive attribute 1045
Origin, centering a model at the origin 352
Orphan 280
Ortho/Perspective View 317
Output Directory, Project panel preference 1002
Output Filename Prefix 1092
Over/Under, View controls 312, 325
Overwriting existing road geometry 1256

P

Pack All Palettes 435
Pack Palette 435
Paint Mode tools, Texture Editor
  Add Mode 571
  Blend Mode 572
  Filter Mode 574
  Max Intensity Mode 572
  Percent of Pen 571
  Replace Mode 571
  Smear Mode 573
  Spray Can Mode 574
  Subtract Mode 571
Palette inconsistencies, resolving 880
Palette Index, light source attributes 616
Palette override, in external references 1162
Palettes
  default.txt (default Texture palette) 890
  flt1.materials(default Material palette) 890
  loading a Material palette 394
  loading Light Source palettes 617
  mg1.lts (default Light Source palette) 890
  Quick 358
  saving a default Light Source palette 618
  saving a Material palette 393
  saving Light Source palettes 617
  Sound palette 669
  terrain contour 951
Palettes menu
  Color (Ctrl+P) 399
  Fractal Texture tool 594
  IR 416
  Light Source 600
  Line Style palette 821
  Material 387
  Sound 669
  Texture Compositor 586
  Texture Mapping 516
  Texture Sequencer 589
Pan View 72, 255, 286
Parallel slopes, in road generation 1239
Parallel to Edge tool 210
Parent 279
  see also “database parent”
Partial, Batch GeoPut preference 1199
Paste (Ctrl+V) 72, 347
Paste Culture Attributes tool 207
Paste GeoFeature Attributes 207, 208, 1115
Path attributes 1289
Path data
  lanes 1286
Path files
  overview 1276
Path Name, Road Lane parameter 1285
Path node 1281
Pathfinder
  modifying texture attributes 1167
  projecting linears as Pathfinder features 1166
  roads as Pathfinder features 1167

Peak tool (Shift+P) 103
Pen tip, Texture Editor 567
Performer parameters
  Light Points tool 648
Perpendicular to Edge tool 212
Perpendicular to Trackplane tool 214
Perpendicular Tracking Plane (P) 310, 324
phone numbers
  MultiGen-Paradigm, Inc. 3
Pick Mode 73, 327
Pitch 257, 262, 739
Pivot v, texture attributes 1168
Pixel fill rate 958
Place in Object, Feature parameter (Road) 1271
Place Light Source 614
Place Morph Vertices 718
Plane Coefficient 865
Plant GeoFeature models 1145, 1162
Plant GeoFeature Models, Projection preference 1160, 1162
Plant tool 173
Plug-ins
  MPI_CREATOR_PLUGINDIR environment variable 43
Point feature
  creating 1100
Polygon control
  in road curves 1282
  in road shoulder curve spirals 1259
  in road side areas 1259
  Spiral poly length 1244
  Twist Type 1238
Polygon Count (Modify Delaunay) 970
Polygon node - see “face node”
Polygon tool (Shift+I) 85
Polygons, limiting in terrain 944
Polymesh
  advantages and disadvantages 983, 1010
  Irregular Mesh terrain processing 1014
Polymesh preferences
  Best Fit 1016
  Bottom Left/Upper Right 1017
  Bottom Right/Upper Left 1017
  Disable LOD Boundary Walls 1014
  Group Objects 1017
  High LOD Polygon Count 1013
  Number of Objects Per Group X (and Y) 1018
  overview 1012
  Post Sampling Rate 1013
  Quads in Flat Area 1017
  Z Scale Multiplier 1014
Polymesh settings
  Calculate RMS Error 1016
  Use Flt Meshes 1016
Polymesh terrain conversion
  overview 1011
Populate tool
  centering models at the origin 352
  definition 164, 168
  using 164
Position DOF 738
Position, light source 616
Post file, saving 1006
Post Sampling Rate, Polymesh preference 1013
Posts (Modify Delaunay) 970
Preference files
  .prefs (Terrain window settings) 891
  1.bat (obsolete) 891
  1.bpf (terrain Batch settings) 891
  creator.ksc (default keyboard shortcuts) 889
  creator1.prefs (application preferences) 889
  custom.ksc (custom keyboard shortcuts) 889
  default.txt (default Texture palette) 890
  dfad.prefs (terrain feature preferences) 891
  dfadbat1.prefs (Rules and Actions) 892
  flt1.materials(default Material palette) 890
  mg1.lts (default Light Source palette) 890
  mgprop.dat (light point palettes) 892
Preferences
  Action Definition 1149
  Batch 1050
  CAT 1040
  Delaunay 1019
  Feature 1055, 1130
  Feature panel 1053
  Feature Projection 1138
  LOD Rules Definition 1154
  Map panel 1004
  Polymesh 1016
Preferences (Ctrl+Shift+P) 49, 889
Preferences, BSP
  Auto Slice if Not Separable 914, 928
  Generate Only Vertical Planes 914
  Interpenetration Tolerance 914

Separate Hierarchy at Lowest Level 914
Skip Disabled Nodes During Separation 914
Preferences, Editing
Auto Apply Texture and Mapping 545
Loft Faces in Order Selected 115
Match Vertices Tolerance 202
Max Undos 244
Shade All if Nothing Selected 619, 621
Slice Displayed Nodes Only 175
Preferences, Flight 60
Allow Default Parent 63, 291
Auto Save 61, 247
Auto Save Checkpoints 247
Auto Save Confirmation Prompt 61, 247
Auto Save Every k Minutes 61, 246
Check Relative Priority When Shifting Nodes Left or Right 62, 230, 231, 288
Default Database Units 62
Default Hierarchy Display Level 283
Enable Switch Nodes 63
Hierarchy Display Level 62
Intensities for RGB Matching 64, 415
Keep Backup Files 61, 247
Modulate Current Color with Material in Material Palette 62
Read External References 63
Update Database Latitude and Longitude 63
Use Attribute Index Matching 64
Preferences, Nintendo
Draw Bounding Box 65
Draw Edges 65
Draw Polygons 65
Wall Alpha 66
Preferences, State
BSP Draw Order 912
Draw Light Sources 619
Draw Mode 321
Draw Order 683
Draw Orders Used 683
Dynamic Texture Default 543
Gouraud Shading 621
Graphic View Defaults 261
Light Source Size 620
Lighting On 619
Preferences, Texture 541
Minification/magnification 527
Prefix
area block naming convention 1050
Preserve Coastline 945
Preserve Coastline, Delaunay preference 1018, 1025
Preview Tessellation controls
Center to Left 1244
description 1243
Horizontal Angle 1243
Spiral Subdivision 1243
Type 1244
using 1274
Vertical Angle 1243
Width 1244
Previewer
Vega Viewer 274
Previous Section, Construction Tool parameter 1235
Primary Color Index 841, 853
Primary Color Intensity 842, 853
Primary Color Name 842, 853
Primary Color RGB 842, 853
Priority listing - see “ Fixed list”
product
update disclaimer 3
Profile Database
in road scenarios 1281
Road LOD parameter 1263, 1266
Scenario parameter 1282
Project and Cut 1157
Project and Cut Low Z 1157
Project Feature(s) 1118, 1157
Project file
Creating 1056
Opening 1056
opening outside Creator 988
Project Flat 1158
Project Flat and Cut 1158
Project Flat and Cut Low Z 1158
Project on Road, Feature parameter (Road) 1271
Project panel, Terrain window 1000
Project tool 172
Project Tops Only 1158
Project Walls Only 1157
Projecting features 1127
Projection Preferences
Feature Preferences 1136, 1138
GeoFeature menu 1160
in Feature Preferences 1134
Plant GeoFeature models 1145, 1162

Projection types
Flat Earth 1005
Geocentric 1005
Trapezoidal 1005
UTM 1005
Projection, GeoFeature Action Definition preference 1149
Properties toolbox
Copy GeoFeature Attributes 207, 208, 1115
Line Style From Face 206
Paste GeoFeature Attributes 207, 208, 1115
Properties tools
Get Color From Face (Ctrl+Shift+R) 410
Get IR Color From Face 418
Get Material From Face 395, 396
Get Texture From Face (Ctrl+Shift+T) 462
Get Texture From Face (Ctrl-Shift+T) 462
Insert Color (Ctrl+R) 409
Insert IR Color 417, 418
Insert Material 395
Insert Texture (Ctrl+T) 461
PRP - see " Perpendicular Tracking Plane"
Put Bounding Volume 698
Put Texture tool (Shift+,)
mapping onto faces 445
Put tool 154

Q

Quad Strips, mesh attribute 860
Quad Tree LOD Boundary Posts, Delaunay preference 1024
Quad tree LODs 952, 1026
Quadrants On/Off 312, 326
Quads in Flat Area, Polymesh preference 1017
Quick palettes 358
adding an item 359
deleting an item 360
removing/restoring multiple items 360
selecting an item 359
Quitting the program 35

R

Radial Project Texture tool 457
Radius, group attribute 838
Raise Flat Projected Feature, GeoFeature Action Definition preference 1151
Random Circle 636
Random Polygon 636
Random Rectangle 635
Randomize Intensity 651
Randomize Selection 1141
Randomize Vectors 630
Raster display mode 651
Raster or Calligraphic display mode 651
Raster To DED 1067
converting ASCII or binary files 1067
converting image files 1068
Read External References 63
Read Select, Batch preference 1052
readdma conversion utility 1087, 1207
Reading setup files 1274
readusgs conversion utility 1088, 1209
Real world texture size 527
Realtime considerations
applications processing 957
collision detection 957
culling 957
optimization issues 681
rendering 957
Realtime systems
displaying CAT terrain 1038
Recalling tracking planes 313
Recent Files 243
Rect. High Res 1022
Rectangle Plus tool 87
Rectangle tool
creating rectangle 87, 92
Rectangle, creating with Coordinate panel 342
Recursive Separation 913, 920
Reduce Areal Features, GeoFeature Action Definition preference 1153
Reduce Areals 1098
Reduce Features Vertex Count, GeoFeature Action Definition preference 1153
Reduce Linear Features, GeoFeature Action Definition preference 1153
Reduce Linears 1098
Reducing vertices
after features are imported 1119, 1121
while importing features 1098
Reduction Tolerance 1098
Reduction Tolerance, GeoFeature Action Definition preference 1153
Referencing a vertex 355
Referencing an edge 356

Reflect Current Selection 827
Reflecting geometry - see “Mirror”
Reflection map - see “Environment Map Texture tool”
Relative Priority
about 688
face attribute 846
group attribute 836
mesh attribute 858
object attribute 839
Reloading textures in the Texture palette 427
Remove Material 395
Remove Morph Vertices 720
Remove Plane/Hulls 926
Remove Texture 462
Remove Unused in Subordinate Files 881
Remove Waveform 674
Removing items from Select List 330
Removing unused textures 430
Rename (Ctrl+J) 241, 289
Render Both Sides Visible 846, 857
Render Textured Polygons White 846, 857
Rendering, effect on terrain 957
Reorder texture layers 476
Reorganize Hierarchy tool 231
overview 300
parameters 301
Repeat Texture 867
Repeated texture pattern 445
Repeating Geometry, in road generation 1269
Repeating XREF, in road generation 1269
Repetition Factor 458
Replace Graphics 347
Replace texture algorithm
MultiTexture 470
Replicate tool 162
procedure 162
Replication
creating 162
defined 162
overview 162
Replication Count 835
Report Generator 872
Requirements, system 33
Reread Externals, File menu 374
Reset CAD View 79
Reset View (U) 50, 74, 261, 262
Reset, Terrain window 941
Resizing - see “Scale”
Resolve Color Palette Inconsistencies 881
Resolve Material Palette Inconsistencies 881
Resolve Texture Palette Inconsistencies 881
Resolving palette inconsistencies 880
Reverse Face tool
example 184
overview 184
Revert
fog control 268
Revision Date 832
Revision Number 832
Revolve About Edge tool
overview 106
parameters 108
procedure 107
RGB Mode 408, 833
RGB texture pattern 421
RGB-alpha texture pattern 421
Ridge and Valley Detection 945, 946
Ridge and Valley Detection, Delaunay preference 1018, 1023
Ridge and Valley Preservation, Delaunay preference 1023
Ridge and Valley Tolerance, Delaunay preference 1023
Rigid Body, Adaptive scope 1046
RMS Error 1016, 1022
Road
specifying section type 1236
Road attributes 1289
Road Bed parameters
Center to Left 1257
Create Side Areas 1257
Edge Z Value Basis 1258
Horizontal /Vertical Break Angle 1259
Road Width 1257
Shoulder Width 1258
Spiral Subdivision 1259
Spiral Type 1259
Road data
generating path files 1276
Road generation
180 degree curves 1245
adding new road sections 1236
centerline location 1243
compound radial curves 1246
construction overview 1232

creating hills 1247
creating straight sections 1250
defining section type 1236
defining width during construction 1243
entry and exit slopes 1239
modifying sections 1253
polygon control 1238
selecting a road to modify 1253
special hills 1249
superelevation 1251
types of sections 1232
width of road 1244
Road Lane parameters
Centerline Offset 1285
Hidden 1285
No Passing 1285
Orientation Type 1285
Path Name 1285
Speed Limit 1285
Road LOD attributes 1290
Road LOD parameters
Close Profile 1263, 1266
Color 1264
Horizontal Break Angle 1259, 1263
Length Offset 1265
LOD Method 1261
LOD Range 1262
Material 1264
Profile Database 1263, 1266
Side Area 1262
Side Area Color 1265
Side Area Material 1265
Side Area Texture 1265
Spiral Subdivision 1252, 1263
Spiral Type 1252, 1263
Texture 1264
Texture Length 1264
Vertical Break Angle 1263
Road menu
Auto Tessellate 1273
Drive Road 1275
Select Road 1253
Tessellation Tool 1253
Write Paths 1276
Road scenario
creating a definition 1281
defining lanes 1284
definition 1280
modifying a definition 1281
parameters 1282
profile database 1281
Road Lane parameters 1285
Road tessellation
applying a road definition 1273
Auto Tessellate 1273
banking 1258
changing a component name 1254
defining levels of detail 1259
defining the road bed 1257
Feature parameters 1268
features, definition 1267
Level of Detail parameters 1261
LOD switch in distance 1275
lofting 1265
overview 1252
Preview Tessellation controls 1274
reading setup files 1274
Road Bed parameters 1257
saving a setup file 1273
selecting a road 1274
Tessellation Tool 1253
Tessellation Tool controls 1255
Road Width
Road Bed parameter 1257
Scenario parameter 1282
Roads
default.rds road setup definitions file 893
Roadway centerline files 1277
description 1277
sample 1278
Roll 257, 262, 739
Roof coverage 1104
Roof tool 133
Roofline, face attribute 848
Roofline, mesh attribute 859
Rotate - Scale to Point tool 151
Rotate About Edge tool 148
Rotate About Point tool (Shift+R) 146
Rotate Grid 308, 323
Rotate Texture 484
Rotate View 255
Rotate, DOF transformation 741
setting limits 739
Rotating features 1271

Round Earth projection 1060
Rounded Strip Face tool 111
Rules
    assigning to LODs 1156
Rules and Actions
    dfadbat1.prefs file 892
Rules and Actions, Project panel preferences 1001

S

Sampling Tolerance 623
Save As 247
Save Checkpoint 248
Save Database File (Ctrl+S) 72, 244
Save default Light Source palette 618
Save Palette, Material palette 393
Save Vertex Normals 625
Save, Eyepoint menu 253
Save, Light Source palette 617
Saving a DED file 1006
Saving and loading sound palettes 671
Saving files automatically 245
Saving new files 245
Scale Model to Feature Size 1141
Scale Texture 486
Scale tool (Shift+S) 149
Scatter Points in Areals 1118
Scatter tool
    centering models at the origin 352
    definition 164, 168
    using 168
Scenario data 1280
Scenario lane files 1283, 1287
Scenario parameters
    Center to Left 1282
    Close Profile 1282
    Horizontal Break Angle 1282
    Profile Database 1282
    Road Width 1282
    Spiral Subdivision 1282
    Spiral Type 1283
    Vertical Break Angle 1282
Searching
    node attributes 869
Searching and modifying external references 375
Section Creation, Construction Tool parameter 1236
Select Adjacent Faces 336
Select All (Ctrl+Shift+A) 333
Select All Features 1095, 1097
Select All Lights 334
Select All, Batch preference 1052
Select Backfacing Faces 335
Select By Attributes 1095, 1097
Select Concave Faces 335
Select Concave Objects 335
Select Concurrent Faces 339
Select Concurrent Vertices 334
Select Connected Faces 337
Select Empty Nodes 334
Select List
    displaying 74
    overview 329
    removing items 330
    using 329
Select menu
    Change mode (Ctrl+Shift+C) 233, 339
    Deselect All 333
    Fence Touching 328
    Isolate 273, 332, 930
    Select Adjacent Faces 336
    Select All (Ctrl+Shift+A) 333
    Select All Lights 334
    Select Backfacing Faces 335
    Select Concave Faces 335
    Select Concave Objects 335
    Select Concurrent Faces 339
    Select Concurrent Vertices 334
    Select Connected Faces 337
    Select Empty Nodes 334
    Select From ID 333
    Select List 329
    Select Non-Manifold Faces 338
    Select Nonplanar Faces 334
    Select Thin Faces 335
Select menu, Texture Editor
    Deselect All 557
    Invert Select 557
    Select All 557
    Show Select Outline 557
Select Non-Manifold Faces 338
Select Nonplanar Faces 334
Select Planes 926
Select Road, Road menu 1253
Select Thin Faces 335
Select tools, Texture Editor
    Color From Texel 566

Select by Color 566
Select by Drawing 565
Select by Ellipse 565
Select by Line 565
Select by Polygon 565
Select by Rectangle 565
Select Lasso 565
Selecting
edges and vertices 328
fence touching 328, 340
in Hierarchy view 327
items in the database 327
light source 600
turning constructions off and on 319
turning highlighting off and on 319, 329
using Select From ID 333
with arrow keys 331
Selection
displaying outline 329
line preferences 328
Selection controls
Fence Touch On 73
Fence Within 73
Fit on Grid (Ctrl+Shift+V) 74
Fit on Pick (Ctrl+F) 73
Fit on Select (V) 73
Mode Select List 73
Pick Mode 73, 327
Reset View (U) 74
Separate Children 913, 919
Separate Hierarchy at Lowest Level 914
Separate Selected 913, 916
Set DOF Limits parameters 740
Set Global 367
Set Local From Path 367
Set LOD Distance 715
Set Preference, fog control 268
Setup files
reading 1274
saving 1273
Shade All if Nothing Selected 619, 621
Shade, face attribute 625, 845, 856
Shading a database 620
Shading, modifying vertex normals 627, 629
Shadow, object attribute 840
Shift Far Left tool 230
Shift Far Right tool 231
Shift Left tool 230
Shift Right tool 230
Shininess 392
Modify Material parameters 391
Shoulder Width, Road Bed parameter 1258
Shoulders, on roads 1257
Show Feature Data 1124
Show Feature Summary 1097
Show Grid, Batch preference 1050
Show Morph Vertices 719
Shrink Mode (A) 318, 328
Siblings 280
Side Area Color, Road LOD parameter 1265
Side Area Material, Road LOD parameter 1265
Side Area Texture, Road LOD parameter 1265
Side Area, Road LOD parameter 1262
Significance 836, 840
Significant Size, LOD attribute 713
Simplifying geometry - see “Generate LOD”
Simulation data
roadway centerline files 1277
scenario lane files 1283
Single geometry, in road generation 1269
Single XREF, in road generation 1269
Skip Disabled Nodes During Separation 914
Sky color 269
controls 270
parameters 270
Sky Color control parameters
Baseline Controls 271
time of day 271
Slant range, defined 713
Slice Displayed Nodes Only 175
Slice tool 174
SmartScene Palette, face attribute 848
SmartScene Palette, mesh attribute 859
SMC 645
face attribute 848
matching Feature Preference setting 1101, 1103, 1105
mesh attribute 859
selecting in a dialog box 1101, 1103, 1105
Smooth shading 624
Snapshot 248
Solid 322
Sound
editing with external editor 671

files 669
nodes 672
Sound attributes 675, 868
Absorption 677
Active 676
Amplitude 676
and realtime system 677
Delay 678
Direction 677
Doppler 677
Falloff 677
File 676
ID 676
Index 676
Pitch Bend 676
Position 677
Priority 677
Wave Index 670
Width 677
Sound ID field 676
Sound menu
Insert Waveform 674
Play Selected 675
Position Sound 675
Remove Waveform 674
Sound Mode tool 235, 674
Sound palette 669
button 75
Index parameter 670
Name 670
Number of Samples 670
Samples Per Second 670
Southwest Latitude 1097
Southwest Longitude 1097
Spacing Type, Feature parameter (Road) 1269
Special 1 and 2 836, 840
LOD attribute 714
Specular
Modify Light Source parameters 609
Modify Material parameters 393
Speed Limit, Road Lane parameter 1285
Speed, Drive Road parameter 1276
Sphere tool 105
Spherical Project Texture tool 456
Spiral String 636
Spiral Subdivision
Preview Tessellation control 1243
Road Bed parameter 1259
Road LOD parameter 1252
Scenario parameter 1282
Spiral Subdivision, Road LOD parameter 1263
Spiral Type
Angle 1244, 1259, 1264, 1283
Distance 1244, 1259, 1283
Length 1264
Road Bed parameter 1259
Road LOD parameter 1252
Scenario parameter 1283
Spiral Type, Road LOD preference 1263
Split Face tool 185
Spot Cutoff Angle
Modify Light source parameters 609
Spot Dropoff Exponent
Modify Light Source parameters 610
Spot light source 599
Standard View (Shift+F9) 80, 281
Starting the software 34
State preferences
Ambient Value for Gouraud Shading 621
BSP Draw Order 912
Draw Light Sources 619
Draw Mode 321
Draw Order 683
Draw Orders Used 683
Gouraud Shading 621
Graphic View Defaults 261
Light Source Size 620
Lighting On 619
Statistics (Ctrl+?) 81
Statistics, Texture palette 434
Status Log 82, 247
Step
DOF attribute 866, 867, 868
DOF parameter 740
STL
importing files from 1228
STL export 1229
Stop driving, in driving simulation 1276
Storing tracking planes 313
Straight road sections
creating 1250
definition 1236
straight road sections 1236
Straight String 636

String w/Count 635
String w/Delt and Count 635
String w/Delta 635
Strip Face tool 108
Subfaces
  defined 294
  with z-buffer drawing 687
Subordinate Files to Merge List 882
Subtexture
  3-Point Put and 4-Point Put texture tools 480
  Defining and applying 479
  Overview 478, 1370
  Purpose 478
  Texture collage 478, 479
Subtree 280
Sun Raster
  supported file format 421, 551
Sunraster
  supported file format 421, 551
Superelevation
  spiral poly length 1259
support, technical
  telephone number for 3
Surface Mode tool 237
Surface Project Texture tool 455
SW and NE Corner 834
SW Corner 834
Switch attributes
  All Off 864
  All On 864
  Current Mask Index 863
  Delete 864
  Invert 864
  Mask Count 863
  Mask Name 863
  New 863
  Update 864
Switch In, LOD attribute 713
Switch LODs During Placement 718
Switch Mode tool 235
Switch nodes 861, 862
Switch Out, LOD attribute 713
Switch range, setting for terrain 955
Switching distance, level of detail 998, 1037
Switching LODs 713
System requirements 33

T

Taper Deform tool
  overview 181
  parameters 182
  procedure 181
TCT
  advantages and disadvantages 1011
  bridge checking 1179
TCT conversion preferences
  Accuracy Tolerance (Lowest LOD) 1033
  Clear Heights button 1034
  Cutout List 1034
  Maximum Poly Count (Lowest LOD) 1033
  Project Feature Only (No Terrain Polygons) 1034
technical support
  telephone number for 3
Technical support phone number 41
telephone numbers
  MultiGen-Paradigm, Inc. 3
Terrain
  and MPI_CREATOR_TERPATH 44
  applying a geospecific texture 1196
  applying texture 1183
  applying texture with Batch GeoPut 1198
  balancing culling and drawing 957, 1057
  CAT conversion preferences 1039
  converting to DED format 1085
  Creating a project file 1056
  creating LODs 952
  creation process 984
  Creator Terrain Studio (CTS) 1204
  defining contours 1063
  elevation data 1063
  Gaming Area 941
  geospecific texture, description 1191
  Group Size parameters 944
  indirect texture 1185
  Maximum Polygons Per Group 944
  Modify Delaunay 960
  Modifying a database 1057
  Opening a project file 1056
  Polymesh 1011

preference files
.prefs 891
1.bpf 891
dfad.prefs 891
dfadbat1.prefs 892
mgprop.dat 892
Preserve Coastline 945
quad tree LODs (defined) 952, 1026
Ridge and Valley Detection 945, 946
switch range setting 955
texture considerations 958
tips for building 956
triangulation parameters 941
Use Terrain Error Correction 946
Z Scale Factor 944
Terrain Accuracy Tolerance, Delaunay preference 1026
Terrain Conforming light points 645
Terrain conversion utilities
catdma 1205, 1212
float2ded 1089, 1210
image2ded 1205
readdma 1087, 1205, 1207
readusgs 1088, 1205, 1209
Terrain Culture Triangulation
automatically creating Pathfinder roads 1152
cutout files 1034
cutting culture into terrain 1151
errors in cutout files 1179
inserting OpenFlight files 1034
LODs 1034
modifying Pathfinder texture attributes 1167
projecting only features 1179
projecting Pathfinder features 1166
setting error tolerance 1033
setting polygon budget 1033
Triangle panel 1031
using culture only 1034
Terrain Flag, face attribute 848
Terrain Flag,mesh attribute 859
Terrain menu
Batch GeoPut 1197
Convert Database Projection 1059
Convert Database to DED 1214
Import Elevation 940, 989
Lat/Long to X/Y 1058
Modify Delaunay 968
Terrain option 939
Terrain processing
advantages of Batch processing 1171
choosing a terrain algorithm 983
choosing type of processing 1171
ellipsoid projection 1008
Map panel settings 1006
Terrain window 989
contours, description 993
Gaming Area 990
preferences panels 1000
Project panel 1000
selecting an area for terrain conversion 990
TCT conversion preferences 1031
Tessellate tool 193
Tessellation
in roads, overview 1252
Tessellation Tool 239, 1253
applying a road definition 1273
reading setup files 1274
saving a setup file 1273
Tessellation Tool controls
Apply 1256
Copy 1256
Create 1255
Create Definition 1255
Cut 1256
Delete 1256
Edit Properties 1255
Move Down 1256
Move Up 1255
Paste 1256
Read Setup File 1255
Replace Existing Road Geometry 1256
Toggle Display 1255
Write Setup File 1255
Text
attributes 818
Text Mode tool 237
Text nodes
creating 815
Text, 2D 812
Texture
alignment 447
and MPI_CREATOR_TXTPATH 44
applying geospecific texture 1196
applying to elevation contours 1183
applying to Pathfinder features 1167

assigning default texture attributes 425
blending transparency attribute 522
calculating texture memory requirements 466
creating a mirror effect 459
default Texture palette file 441
displaing path and file name 442
editing 545
full-size texture display 527
geospecific texture, description 1191
horizontal offset 442
indirect 1185
indirect texture 996
magnified texture display 527
mapping from and to points 448
Mapping Name 450
mapping onto faces 445
mapping texels to pixels 527
memory 464, 958
modifying geospecific coordinates 1192
Modifying Pivot v value 1168
moving a pattern to another palette 428
MultiTexture 470
Organize Palette tool 434
Pack All Palettes 435
Pack Palette 435
palette order number 441
pattern ID number 442
preferences 541
reduced texture display 527
removing 462
Repeating a texture pattern 448, 453
Road LOD parameter 1264
selecting a texture pattern 423
supported file formats 421, 551
terrain considerations 958
Texture palette 422
texture palette file 441
tracking plane use during texture application 448
Use Attribute Index Mapping preference 64
using the current texture mapping 517
vertical offset 442
View control 321
Texture and feature projection 1135
Texture and library substitution 1163
Texture application preferences
Effect of u and v wrap method 1264
Texture length 1264
Texture attributes
Blend, texture environment 530
DBL origin 526
Decal, texture environment 530
detail texture 530
External Format 526
File Type 525
geospecific coordinates 1192
Intensity 525
Intensity-alpha 525
Magnification filter 529
Minification filter 528
Modulate, texture environment 530
MultiTextures 530
Primary 524
Real World Size 447, 527
Replace, texture environment 530
RGB file type 525
RGB-alpha 525
Texture Size 527
Up Vector 527
Wrap Method u and v 526
Texture Attributes - Base, face attribute 843
Texture Attributes - Base, mesh attribute 854
Texture Attributes - Detail, face attribute 843
Texture Attributes - Detail, mesh attribute 854
Texture Attributes - Mapping, face attribute 843
Texture Attributes - Mapping, mesh attribute 854
Texture Composer tool
composite textures 507
modifying texture mappings 504
overview 499
parameters 510
templates 500
Texture Compositor 586
Texture Editor
defining the pen tip
overview 567
pen color 567
pen selection 569
pen style 568
displaying and protecting texels 577
Draw tools 558
Edit menu 554
Exercise
Drawing With a Pen Tip 575
Selecting and Flipping Texels 583

eyepoint control 550
file formats 551
File menu 553
Maneuver tools 584
Operations tools 578, 584
Options menu 557
overview 548
Paint Mode tools 570
selecting texels
Select menu 556
Select tools 563
setting up
on Windows 545, 547
starting 549
Texel display 550
View menu 555
Texture Elevation Scale 951
Texture layers
copying 474
deleting 477
reordering 476
Texture Length, Road LOD parameter 1264
Texture mapping
GeoPut Texture tool 1202
offsetting a texture pattern 1265
Texture Mapping palette 516
applying the current mapping 517
changing the current mapping 518
creating a new mapping 519
loading 519
menus 520
overview 516
saving 519
Texture Mapping, vertex attribute 851
Texture palette
adding patterns to 424
Center Select 430
creating a default Texture palette 443
default.txt file 890
deleting all textures 430
deleting textures from palette 430
dividing a texture 464
File (name) 423
Fit on Select 430
loading a palette file 442
organizing 434
overview 422
Pack All Palettes 435
Pack Palette 435
removing unapplied textures 430
saving a palette 442
saving a palette file 442
using with wizards 1294, 1305, 1325
View All 430
viewing a texture list 431
viewing texture statistics 434
Texture palette file 441
Filename 442
palnum 441
patnum 442
x 442
y 442
Texture Palette, Project panel preference 1002
Texture pattern types
Intensity 421
Intensity-alpha 421
RGB 421
RGB-alpha 421
Texture patterns, adding to a palette 424
Texture Sequencer 589
The 643
Thumbnail 389, 601
Tile (Shift+F4) 78
Tiles U, V
Four Point Put 453
Three Point Put 448
Toggle Alpha Blending 522
Toggle Display tool 231
Toggle Linear Features 1106
Toolbar
Quick palettes 358
Toolbox caddy, location of 69
Toolbox tear-off windows 81, 85
Tools
4-Point Put Texture 450
Adaptive Mode 237, 1043
Add Vertex 186
Attach 230
Average Vertices 204
Bend Deform 178
BSP Mode 236
Bulldoze 182
Centerline 216
Circle 91

Circle Plus 92
Clip Mode 236
Closest Point on a Line 222
Closest Point on a Plane 224
Collect 232
Combine Faces 189
Construct Average Vertex 221
Construction Curve 218
Contract Hierarchy 232, 281
Cookie Cutter 176
Copy Culture Attributes 208
Create (Shift+C) 226, 290
Create BSP 228
Create Clip 228
Create DOF 227, 736
Create External Reference 228, 373
Create Group 226
Create Instance 228, 369
Create Light Points 227, 660
Create Light Source 228, 599
Create LOD 227
Create Object 227
Create Sound 227
Create Switch 228
Curve 116
Curve Mode 118, 236
Custom Toolbox 238
Cut Subfaces 190
Delete Level 229
Detach 230
Dimmer 204
Divide Quads 194
DOF Mode 235
Duplicate
    copying an instance 370
Duplicate (Shift+D) 160
Edge From Mouse 209
Edge Mode 234
Edge Turn 191
Environment Map Texture 459
Expand Hierarchy 231, 281
External Reference Mode 235
Extrusion 121
Face Mode (Shift+F) 234
Flip Book 128
GeoPut Texture 1202
Get Color From Face (Ctrl+Shift+R) 410
Get IR Color From Face 418
Get Line Style From Face 206
Get Material From Face 395, 396
Get Texture From Face (Ctrl+Shift+T) 462
Grid Mode 237
Group Mode (Shift+G) 233
Helper Mode 237
Insert Color (Ctrl+R) 409
Insert IR Color 417
Insert Level Above 229
Insert Line Style 206
Insert Material 395
Insert Texture (Ctrl+T) 461
Insert Transformation Matrix 157, 158
Intersection of Two Edges 222
Intersection of Two Planes 217
Interval Between Two Vertices 223
Lattice Deform 177
Light Mode 235
Light Point 631
Light Point Mode 236, 660, 661
Linear Bend 173
LOD Mode 234
Loft 115
Match Vertices 202
Mesh Mode 237
Mirror 170
Modify Mapping 488
Modify Vertex (Shift+M) 200
Modify Vertex Normals 627, 629
Object Mode (Shift+O) 234
Organize Palette 434
Orient Faces 196
Parallel to Edge 210
Paste Culture Attributes 207
Peak (Shift+P) 103
Perpendicular to Edge 212
Perpendicular to Trackplane 214
Plant 173
Polygon (Shift+I) 85
Populate 164, 168
Project 172
Put 154
Put Texture (Shift+,) 445
Radial Project Texture 457
Rectangle 86, 92
Rectangle Plus 87

Reorganize Hierarchy 231
Replicate 162
Reverse Face 184
Revolve About Edge 106
Roof tool 133
Rotate - Scale to Point 151
Rotate About Edge 148
Rotate About Point (Shift+R) 146
Rounded Strip Face 111
Scale (Shift+S) 149
Scatter 164, 168
Shift Far Left 230
Shift Far Right 231
Shift Left 230
Shift Right 230
Slice 174
Sound Mode 235
Sphere 105
Spherical Project Texture 456
Split Face 185
Strip Face 108
Surface Mode 237
Surface Project Texture 455
Switch Mode 235
Taper Deform 181
Tessellate 193
Text Mode 237
Toggle Display 231
Torsion Mode 237
Translate
translating an instance 369
Translate (Shift+T) 144
Triangulate 188
True Type Text 139
T-Vertex Killer 197
Twist Deform 180
Vertex From Mouse 220
Vertex Mode 234
Wall (Shift+W) 96
Wall Plus 98
Torsion Mode tool 237
Total ambient light computation 621
Tracking plane
components 303
custom 313
menu 77, 313
overview 303
use during texture application 448
Trackplane Center 309, 323
Trackplane Control
Construction Tool parameter 1235
Trackplane From Face 308, 323
Trackplane From Vertex 310, 323
Trackplane From View 311, 324
Transfer Properties
Modify Delaunay window 970
Transfer Subfaces
Modify Delaunay window 971
Transferring Feature Preferences 1142
Transformation edits
applying 158
assigning to DOFs 739
overview 157
Transition, LOD attribute 713
Translate Texture 481
Translate tool (Shift+T) 144
Translate tools
translating an instance 369
Translate, DOF transformation
about 740
setting limits 739
Transparency
face attribute 845
mesh attribute 857
object attribute 839
Transparency in materials 391
Trapezoidal projection 1005, 1060
Trapezoidal terrain projectionprojection
Geocentric 1008
Traversal, database
overview 688
Tree coverage 1103
Tree wizard
tutorial 1304
Triangle Area Weighting 623
Triangle Fans, mesh attribute 860
Triangle Strips, mesh attribute 860
Triangulate tool
overview 188
procedure 188
Triangulation parameters 941
choosing 941
Group Size in Latitude and Longitude Posts 944
Maximum Polygons Per Group 944

Preserve Coastline 945
Ridge and Valley Detection and Tolerance 945
Use Terrain Error Correction 946
Z Scale Factor 944
True Type Text tool 139
T-Vertex Killer tool 197
Twist Deform tool
overview 180
parameters 180
procedure 180
Twist Type
Construction Tool parameter 1238
Cosine Function with Length 1238
Linear with Curve 1237
Linear with Length 1237
Type
face attribute 845
group attribute 837
mesh attribute 856
Type, Preview Tessellation control 1244

U

Un-Constrain DOF 745
Undo (Ctrl+Z) 72, 244
Undo History 244
Unidirectional 1105, 1113
Units, Batch preference 1050
Universal Transverse Mercator
see UTM terrain projection
Up, LOD Rules Definition preference 1155
Up-and-down hills, in road design 1249
Update
Modify Materials 391
Update All Graphics Views 265
Update All Graphics Views, Fog Controls 268
Update Database Latitude and Longitude 63
Update Now, fog control 268
Update Vertex Colors 625
Update Vertex Normals 623
Update, switch attribute 864
Use Attribute Index Matching 64
Use Center Point, Vertical Curve control 1239
Use Edge Z, Feature parameter (Road) 1272
Use Entered Prefix Only 1092
Use External References, GeoFeature Projection preference 1162
Use Face Color Intensities 625
Use Feature Data Extents 1095, 1097
Use File Name as Prefix 299
Use Flt Meshes 1016
Use Geospecific Texture, Texture panel preference 1047
Use Lower Left Lat/Long, area block naming convention 1051
Use Maximum, Vertical Curve control 1242
Use Node's ID 299
Use Post Weighting Multipliers, Delaunay preference 1024
Use Post Weighting Parameters 1018
Use Previous Range, LOD attribute 713
Use Strip Face Texture On 3D Tops 1139
Use Terrain Error Correction 946
USGS raw data conversion 1209
UTM projection 1005, 1060
UTM terrain projection 1006
UV Layers, mesh attribute 861

V

Valid Surface Material Codes 1132, 1133
Valleys, giving preference in terrain 945
Vector data - see Feature data
Vector, Adaptive attribute 1046
Vega Viewer
for previewing databases 274
Vehicle dynamics
lane files 1286
Version number 35
Vertex
aligning with Modify UVs texture tool 495
defined 295
displaying normals 626
moving with Modify UVs texture tool 493
referencing 355
rotating with Modify UVs texture tool 494
scaling with Modify UVs texture tool 493
selecting 328
Vertex attributes 848
Freeze 850
Hard Edge 850
Has Normal 850
Texture Mapping 851
Vertex Color Index 850
Vertex Color Intensity 850
Vertex Color Name 850
Vertex Color RGB 850, 851
Vertex Location 849
Vertex Normals 850

Vertex Color Index 850
Vertex Color Intensity 850
Vertex Color Name 850
Vertex Color RGB 850, 851
Vertex From Mouse tool 220
Vertex Location 849
Vertex Mode tool 234
Vertex Normals, vertex attribute 850
Vertex toolbox
    Construct Average Vertex 221
    Vertex From Mouse 220
Vertex tools
    Closest Point on a Line 222
    Closest Point on a Plane 224
    Intersection of Two Edges 222
    Interval Between Two Vertices 223
Vertical Angle, Preview Tessellation control 1243
Vertical Break Angle
    Road Bed parameter 1259
    Road LOD parameter 1263
    Scenario parameter 1282
Vertical Curve controls
    Construction Tool parameter 1239
    Curve Length 1241, 1242
    Entry Slope 1240
    Exit Slope 1240
    Maximum Slope 1239
    Minimum Length 1242
    Use Center Point 1239
    Use Maximum 1242
Vertical curves
    in hills 1247
Vertical gauge, creating 784
Vertical Lobe Angle 647, 654
Vertical Offset, Drive Road parameter 1276
Vertical texturre offset 442
View controls
    Antialias Lines 316
    Background Image 318, 362
    Dimension, grid 312, 325
    Display Update Rate 319
    displaying 74
    Draw Bounding Volumes 319, 695
    Draw Clip Nodes 319, 807
    Draw Constructions 319
    Draw Control Points 318
    Draw DOF Axes 316, 737
    Draw Lights Large 317, 661
    Draw Mode (D) 321
    Draw Order 682
    Draw Select Outline 319, 329
    Draw Texture (T) 521
    Draw Vertex Large 317
    Draw Vertex Normals 316, 626
        Light points 661
    Draw Vertex Numbers 318
    Draw Wire Backface 317
    Dynamic Texture 523
    Dynamic Wireframe 255, 318
    Fog On/Off 317
    Grid Larger 307, 322
    Grid On/Off (G) 307, 322
    Grid Smaller 308, 323
    Grid Snap 307, 322
    Grid Spacing Axes 312, 325
    Grid Types 311, 324
    Infrared View 316, 415
    Lights On/Off 316, 600, 622
    Offset 311, 324
    Ortho/Perspective View 317
    Over/Under 312, 325
    Perpendicular Tracking Plane (P) 310, 324
    Quadrants On/Off 312, 326
    Rotate Grid 308, 323
    Shrink Mode (A) 318, 328
    Solid, Draw Mode 322
    Texture 321
    Toggle Alpha Blending 522
    Trackplane Center 309, 323
    Trackplane From Face 308, 323
    Trackplane From Vertex 310, 323
    Trackplane From View 311, 324
    Wire Solid, Draw Mode 322
    Wireframe, Draw Mode 322
    XY coordinate plane 308, 323
    XZ coordinate plane 308, 323
    YZ coordinate plane 308, 323
View Graphics Buffer 345
VIEW icon - see “Trackplane From View”
View menu
    Background Image 361
    Center Select (C) 261, 285
    Fit on Grid (Ctrl+Shift+V) 261
    Fit on Pick (Ctrl+F) 260

Fit on Select (V) 261, 333
Fog Controls 267
Full View 273
Graphics (S) 78
Hidden Faces
and Hidden Polygon 846, 857
Draw Normal 272
Draw Wireframe 273
Hide 272
Hierarchy (S) 78, 281
Reset (U) 50, 261, 262, 283
Sky Color 269
Sky Color Controls 270
Snapshot 248
Track Coordinate 341
Viewing Volume 264
Zoom Control, speed 254
View menu, Texture Editor
Fit on Select 556
Pen Window 556
Render 556
Reset View 556
Tile 556
View menu, Texture palette
Center Select 430
Fit on Select 430
View All 430
View panel
Draw Light Sources 600
Draw Sound 673
Viewing vector 251
Viewing volume
changing size and scale 691
changing the clipping plane 689, 691, 692
controls 264
defined 263
far clipping plane 263
field of view (FOV) 263
near clipping plane 263
overview 689
procedure 689, 691, 692
setting for multiple eyepoints 692
terrain considerations 958
Viewing volume parameters
Far 266
Field of view 266
Near 266
Reset to Defaults 266
Scale 266
Set Preferences 266
Update All Graphics Views 265
Visibility Range, Fog Controls 267
VRML export 1229
VTX icon - see “ Trackplane From Vertex”

W

Wall Alpha Nintendo preference 66
Wall Plus tool 98
Wall tool (Shift+W) 96
Walls, in terrain 1036
Wave File Name 676
Wave Index 670
Waveform
definition of 671
inserting into sound nodes 674
Wavefront OBJ
importing files from 1228
web site
MultiGen-Paradigm, Inc. 3
Width, Preview Tessellation control 1244
Window menu
Arrange Icons 78
CAD View (Shift+F10) 79
Cascade (Shift+F5) 77
Close All 78
Reset CAD View 79
Standard View (Shift+F9) 80, 281
Tile (Shift+F4) 78
Zoom View (Shift+F6) 78
Wire Solid 322
Wireframe 322
Wizards
Billboard wizard tutorial 1293
Building wizard tutorial 1324
Tree wizard tutorial 1304
Write Current Selection to Post File, Map panel preference 1006
Write Paths, Road menu 1276
Write Select 296, 298
Writing out GeoFeature rules and actions 1157
WWW site 42

X

XREF
repeating 1269

XY coordinate plane 308, 323
XZ coordinate plane 308, 323

Y

Yaw 257, 262, 739
YZ coordinate plane 308, 323

Z

Z Scale Factor 944
Z Scale Multiplier
    CAT preference 1041
    Delaunay preference 1020
    Polymesh preference 1014
Z-buffer drawing order 686
Zoom
    in batch mode 1172
Zoom speed control, graphics view 254
Zoom View (Shift+F6) 78
Zoom View control 73, 254, 285
Zoom, Terrain window 941