

Cyclone 5.8

Table Of Contents

Introduction	3
<i>Cyclone</i> Software Introduction	3
Installation.....	5
Included Materials	5
System Requirements.....	5
Installing <i>Cyclone</i>	6
Obtaining your <i>Cyclone</i> License	6
Uninstalling <i>Cyclone</i>	7
Quick Start	9
Launching <i>Cyclone</i>	9
User Configuration Management.....	9
Preferences.....	9
The Navigator Window.....	10
Creating a Database	12
Scanning	12
Registration	14
Modeling.....	15
Scanning with the HDS6000 and HDS4500.....	17
The Scanning Process.....	17
Scanning with the ScanStation, ScanStation 2, and HDS3000	25
The Scanning Process.....	25
Scanning with the HDS2500.....	35
The Scanning Process.....	35
Camera Calibration	38
Scanner Simulation.....	40
Registration.....	41
Introduction	41
The Registration Process.....	41
Cloud Registration	46
Managing ScanWorld Copies	52

Table Of Contents

Modeling	55
Overview	55
Working with Objects	55
Working with the ModelSpace	83
Drawing	89
Pipe Modeling...	91
Piping Mode...	92
Animation.....	92
The Animation Process.....	92
Databases.....	95
Databases	95
Unshared Database Server	95
Remote Administration.....	95
Optimizing a Database.....	97
Virtual Surveyor	99
The Virtual Surveyor Process	99
Sections Manager.....	104
The Sections Process	104
Create Sections Dialog Options.....	106
Sections Manager Dialog Commands	106
Texture Mapping.....	108
The Texture Mapping Process.....	108
Texture Mapping Hints and Tips	111
Texture Map Browser Dialog	112
Texture Editor Dialog	113
Texture Editor (Image) Dialog	114
Texture Map Editor Options Dialog	114
Cyclone Object Exchange (COE)	115
COE for AutoCAD	115
Import Objects (coein).....	122
Explode Blocks (coeXplode)	122
Export Objects (coeout)	122

Table Of Contents

COE for MicroStation	123
OpenGL Modes	137
OpenGL Modes Introduction.....	137
Graphics mode characteristics.....	137
Command Reference	139
2D Drawing Mode	139
2D Extraction.....	139
About <i>Cyclone</i>	139
Above Ref Plane (Mesh).....	140
Acquire Targets.....	140
Activate (Limit Box Manager).....	145
Activate Next (Limit Box Manager)	146
Activate Previous (Limit Box Manager).....	146
Add Both Flat Caps	146
Add Both Semi-Elliptical Heads	146
Add Cloud Constraint.....	147
Add Constraint	148
Add/Edit Annotations...	149
Add/Edit Camera Images.....	152
Add/Edit Colors.....	153
Add/Edit Custom Annotations.....	155
Add/Edit Cutplanes.....	157
Add/Edit Feature Code...	159
Add/Edit Limit Boxes	160
Add/Edit Materials.....	161
Add/Edit Reference Planes.....	161
Add/Edit Registration Label	163
Add/Edit ScanWorld Annotations.....	163
Add/Edit Target Height.....	164
Add Face.....	165
Add Flat Cap Closest to Pick	165
Add Inside Fence	165

Table Of Contents

Add Outside Fence	166
Add/Replace Coordinate.....	166
Add ScanWorld.....	167
Add Semi-Elliptical Head Closest to Pick	167
Adjust Exposure.....	167
Adjust Image.....	168
Align	168
Align Surfaces.....	169
Align to Axis	169
Align to Selection	170
Align Vertices to Axes	170
Align View to Active Plane	170
Alignment and Section.....	171
All in Selection.....	171
Angle	171
Animate.....	172
Animation Editor.....	174
Annotations	176
Appearance.....	176
Append to Alignment.....	176
Apply Color Map.....	177
Apply Multilimage.....	178
Arc (3 Points on Arc).....	178
Arc (Start, Center, End)	178
Atmosphere.....	179
Auto-Add Cloud Constraints	179
Auto-Add Constraints.....	180
Auto-Pick Points.....	181
Auto-Update	181
AutoCAD.....	181
Back	182
Back Angle	182

Table Of Contents

Batch Registration.....	183
Below Ref Plane (Mesh)	183
Breaklines	184
Black/White Target.....	184
Bubble Level.....	184
by Last Selection.....	185
by Ref Plane.....	186
Calibrate Camera.....	186
Cancel Drawing.....	187
Cancel Scan.....	187
Center on Fence	187
Centerline Length.....	188
Check Calibration.....	188
Check Fit Tolerances	188
Check/Re-Level HDS6000.....	189
Check/Re-Level ScanStation.....	189
Circular Fence Mode.....	190
Clear.....	190
Clear Breaklines.....	190
Clearances	191
Clear Calibration Info	194
Clear Image.....	194
Clear Locks	194
Clear Path	195
Clear Target	195
Clear Temporary Measurements	195
Clear Undo/Redo	195
Close	196
Cloud Constraints Wizard.....	196
Codirectional	197
Coincident	198
Colinear.....	198

Table Of Contents

Collapse	198
Collapse (Limit Box Manager).....	199
Collapse Limit Box	199
Connect.....	199
Connect Piping.....	200
Constrain Motion to.....	200
Contours.....	201
Contours to Mesh Deviations.....	201
Convert Elbow to Miter.....	202
Coordinate System	202
Coplanar.....	203
Copy.....	203
Copy (Limit Box Manager)	205
Copy Fenced to New ModelSpace	205
Copy Pick Point(s).....	206
Copy Selection to New ModelSpace	206
Copy to ControlSpace	207
Create... (Contours)	207
Create (Exclusion Volume)	209
Create (Limit Box Manager).....	210
Create Alignment	210
Create and Open ModelSpace	211
Create and Open ModelSpace View.....	211
Create Branch.....	211
Create Drawing	212
Create Drawing from Object	212
Create Lines from Cuts	213
Create Mesh.....	213
Create ModelSpace	214
Create Objects with Random Colors.....	215
Create Path.....	215
Create Path (Loop)	215

Table Of Contents

Create Redlines	215
Create ScanWorld/Freeze Registration	216
Create Sections.....	217
Create Sections from Picks.....	218
Create Simulated Scanner.....	218
Cubic Spline	219
Curve.....	219
Customize Hotkeys.....	220
Customize Lookup Lists.....	221
Customize Toolbars.....	223
Cut.....	225
Cut by Distance from Point	226
Cut by Fence.....	227
Cut by Intensity	227
Cut by Offset from Plane.....	228
Cut Near Ref Plane.....	229
Cut Sub-Selection	229
Cutplane.....	230
Databases.....	230
Datum.....	232
Decimate Contours	232
Decimate Mesh.....	233
Decimate Polyline/Patch	235
Decrease Point Width	236
Defect Report.....	237
Delete.....	237
Delete (Limit Box Manager)	237
Delete Inside	238
Delete Line	238
Delete Outside	238
Delete Selected Inside	238
Delete Selected Outside	239

Table Of Contents

Delete Selection	239
Deselect	239
Deselect Fenced	240
Disable	240
Disassemble.....	240
Disconnect	241
Disconnect (All).....	241
Disconnect (Shared)	241
Display License.....	241
Distance	242
Download HDS6000 Scans.....	243
Draw Arc (3 Points on Arc)	243
Draw Arc (Start, Center, End)	244
Draw Arc (Start, End, Center)	245
Draw Circle (3 Points on Circle).....	245
Draw Circle (Point, Center).....	246
Draw Cubic Spline	246
Draw Cubic Spline Loop	247
Draw Ellipse	247
Draw Isometric Polygon.....	248
Draw Line	248
Draw Polygon.....	249
Draw Polyline	249
Draw Rectangle.....	250
Draw Square	250
Draw Text.....	251
Drawing	251
Eccentric Reducer Connectors	252
Edit Active Plane.....	252
Edit Clipping Planes.....	253
Edit Color Map.....	253
Edit Color/Material.....	255

Table Of Contents

Edit Drawing Parameters.....	257
Edit Elbow BendRatio.....	258
Edit Environmental Lights.....	258
Edit Global Color Map.....	259
Edit Measurements.....	260
Edit Parameters... (Cloud Registration)	262
Edit Parameters... (Region Grow).....	262
Edit Point Properties.....	263
Edit Point/Spot Lights.....	263
Edit Properties.....	264
Edit Properties (Limit Box Manager).....	265
Edit Redlines.....	265
Edit Snap to Object Threshold.....	266
Editor.....	266
Elbow Connectors.....	268
Elevation	269
Elevation Check	269
Enable	271
Enable Dual-Axis Compensator.....	271
Enable Output Box	272
Enable Tilt Sensor.....	272
End Caps	272
Enter Datum.....	272
Exclusion Volume.....	273
Exit	273
Explode	273
Export.....	274
Export (Limit Box Manager)	275
Export Calibration Info	276
Export Customizations	276
Export Leica System 1200	276
Export PCF.....	277

Table Of Contents

Export Template.....	277
Extend	278
Extend All Objects.....	278
Extend TIN to Polyline	279
Extend to Last Selection	279
Extend to Ref Plane	280
Extract Cloud from Selection	280
Extrude.....	280
Extrude Along Axis.....	281
Extrude Perpendicular.....	281
Extrude to Last Pick	282
Fence	282
Fence/View Toggle	283
Field-of-View	283
Field Setup	283
Fill Selected Hole (Mesh).....	289
Fill Selected Hole (Patch)	289
Find High/Low Point.....	290
Fit Edge.....	290
Fit Fenced	293
Fit to Cloud.....	293
Fit to Cloud Options	295
Flip Selected Edge	296
Fog	296
From Curves	296
From Fence Contents.....	297
From Intersections	297
From Pick Points	297
From Selection in Fence.....	298
Front.....	298
Get Image	298
Get Preview.....	299

Table Of Contents

Global Color Map	299
Global Texture Map	299
Grid Settings	300
Group (Command)	300
Group (Menu).....	300
Handles	300
Handles Always Visible.....	301
Hide Point Clouds	301
Home Position.....	301
HDS Target	302
Hide Selected Clouds	302
Hide Unselected Clouds	302
Image Info...	303
Image Resolution	303
Import.....	303
Import (Limit Box Manager)	307
Import Calibration Info.....	307
Import Coordinate List.....	308
Import Coordinate List (Leica System 1200).....	308
Import Customizations	309
Import Leica System 1200	309
Import Registration.....	309
Increase Point Width.....	310
Info (Limit Box Manager).....	310
Info (ModelSpace viewer)	310
Info (Scan Control viewer)	310
Insert	310
Insert Copy of Object's Points.....	312
Insert Point Light.....	312
Insert Spot Light.....	312
Interfering Points.....	313
Invert	313

Table Of Contents

Invert Selection	314
Isometric.....	314
Keep Collapsed.....	314
Keep Viewpoint Upright	314
Launch	315
Layers...	315
License Manager.....	317
Lighting.....	318
Limit Box	318
Limit Box On/Off (Limit Box Manager)	319
Limit Box on Viewer 1	319
Limit Box on Viewer 2	319
Line Segment.....	319
Load Image	320
Load Points within Fence.....	320
Load Points within Fenced Selection.....	320
Lower Active Cutplane	321
Make Available Offline	321
Make Circular.....	322
Make Rectangular	322
Manage.....	322
Measure	323
Merge	323
Merge from ModelSpace.....	324
Merge Sub-Scans	324
Mesh	325
Mesh to Points Deviations.....	325
MicroStation....	326
Minimize	326
Minimize (Limit Box Manager)	327
Minimize Limit Box	327
Miter Connectors.....	327

Table Of Contents

ModelSpace	328
ModelSpace Info	328
ModelSpace View	328
ModelSpaceViews	329
Mouse Modifiers.....	329
Move/Rotate.....	330
Move Vertex	331
Move Viewpoint.....	332
Multi-Pick Mode.....	332
Multi-Pick/View Toggle.....	332
New (Limit Box Manager)	332
New Scanner Position.....	333
Object Coordinate System	333
Object Info.....	334
Object Info... (Navigator Window)	334
Object Preferences.....	334
One-Sided	337
Open	337
Open ModelSpace Viewer	338
Open New Viewer	338
Open Temporary ModelSpace View	338
Optimize	338
Optimize Cloud Alignment	339
Optimize Mesh	340
Orthographic/Perspective	340
Pan (Lock).....	340
Pan (Reference Plane)	341
Panoramic	341
Panoramic Rectangle Fence Mode.....	341
Panoramic Scan Mode.....	342
Parallel	342
Paste	343

Table Of Contents

Paste Link	343
Paste Reference	344
Patch	344
Patch (Submenu)	344
Pause	345
Perpendicular	345
Perspective/Orthographic	345
Pick Mode	346
Pick/Multi-Pick/View	346
Pick/View Toggle	346
Pipe Modeling...	346
Piping	349
Piping Mode...	350
Piping Takeoff	352
Point Cloud Density	353
Point Cloud Rendering	353
Point Cloud Rendering (Registration)	354
Point Cloud Rendering (Scan Control)	354
Point Cloud Sub-Selection	354
Point Distance	355
Pointing	356
Polygonal Fence Mode	358
Polyline (From Curves)	358
Polyline (From Pick Points)	359
Polylines	359
Preferences	359
Print	371
Probe	371
Project	371
Project Polyline to TIN	372
Project Setup	372
Publish Site Map	373

Table Of Contents

Put at Angle.....	378
Put at Distance.....	378
QuickScan (command)	379
Raise Active Cutplane.....	379
Raise/Lower	380
Randomize Colors of Visible (Selected) Objects	380
Rectangle Fence Mode	380
Rectangular.....	381
Redlining	381
Redo.....	382
Reduce Point Cloud.....	382
Reducer Connectors	383
Reference Plane	383
Region Grow	383
Register.....	386
Registration.....	386
Registration (Tools).....	387
Registration Diagnostics	387
Re-Import Scans	387
Remove Both Caps.....	389
Remove Cap Closest to Pick	389
Remove from Group	389
Remove Inside Fence	390
Remove Outside Fence	390
Remove Registration Label.....	390
Rename.....	390
Reset Active Alignment.....	391
Reset Alignment.....	391
Reset ScanWorld Coordinate System	391
Reset Target	391
Resolution (HDS2500)	392
Resolution (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)	392

Table Of Contents

Resolution (HDS4500)	393
Resolution (HDS6000)	393
Restore Default Cloud from Scan	394
Restore Original Scans	394
Resume.....	394
Right Isometric	395
Rotate (Reference Plane)	395
Rotate (View Lock).....	395
Save/Edit Coordinate Systems...	396
Sample Grid	397
Save/Edit Viewpoints.....	398
Save Measurements	400
Save View	400
Save View As.....	400
Scan	401
Scan Filter	401
Scanner.....	402
Scanner Diagnostics	402
Scanner Laser Mode.....	403
Scanner Range Mode	403
Scanner Speed	403
Scanners.....	404
ScanWorld.....	404
ScanWorld Explorer...	405
ScanWorld Info.....	408
Sections Manager	409
Seek Mode	409
Segment Cloud	409
Segment by Polyline	410
Segment Polyline	410
Select a Project.....	411
Select a ScanWorld.....	411

Table Of Contents

Select All	412
Select All (Point Cloud Sub-Selection)	412
Select Connected.....	412
Select Fenced	412
Self-Test.....	413
Servers.....	413
Set Active Alignment.....	414
Set as Default Cloud	414
Set Atmospheric Correction.....	415
Set Datum to Pick Point	415
Set Distance to Next Vertex.....	415
Set from Active Ref Plane (Cutplane).....	416
Set From Points.....	416
Set from Reference Plane.....	417
Set from Selection.....	417
Set Grid Spacing.....	418
Set Half-Space at Pick	418
Set Home ScanWorld	419
Set Limit Box by Cursor	419
Set Limit Box by Fencing	419
Set Object Visibility.....	420
Set Offset.....	420
Set on Object (Cutplane)	421
Set on Object (Reference Plane)	421
Set Origin	421
Set Path	422
Set Plane Origin at Pick Point.....	422
Set Points Selectable.....	422
Set Rotational Spacing.....	422
Set Scan Filters.....	423
Set ScanWorld Coordinate System	423
Set Selectable.....	424

Table Of Contents

Set ScanWorld Default Clouds	424
Set Slice from Picks	425
Set Slice Thickness.....	425
Set Target Height.....	425
Set to { X-Y, X-Z, Y-Z} Plane	426
Set to Object	426
Set to UCS	426
Set to Viewpoint	426
Set Up Direction.....	426
Set Using One Axis.....	427
Set Using Two Axes.....	427
Set Weight.....	428
Shaded.....	429
Shortcut.....	429
Shortcut in Windows	429
Show Active Plane	430
Show All (Limit Box Manager).....	430
Show Annotation Keys.....	430
Show Annotation Points.....	431
Show Annotations	431
Show Constraint.....	431
Show Constraint Viewer 1 (2)	432
Show Coordinate System Axes	432
Show Current Transform.....	432
Show Datum.....	433
Show Diagnostics.....	433
Show Grid	434
Show Handles.....	435
Show Hidden Clouds	435
Show Images	435
Show Incoming Points	435
Show ModelSpace	436

Table Of Contents

Show Object's Cloud.....	436
Show Output Box	436
Show Pick Points	437
Show Pick Polyline.....	437
Show Point Clouds.....	437
Show Rotation Handles	438
Show Scanners.....	438
Show Servers.....	438
Show Shortcuts.....	438
Show Table Matches	438
Show Traverse Report.....	439
Silhouette	439
Slice	440
Snap to Grid	440
Snapping Grid	440
Snapshot.....	441
Sphere Target.....	441
Stakeout.....	442
Standard Viewpoints	444
Station Data (HDS3000)	444
Steel Section.....	445
Subtract From Patch	447
Surface Area	447
Surface Deviation.....	447
Switch Alignment Start/End	449
Synchronize Viewers	450
Target All.....	450
Texture Map Browser.....	450
Texture Map Browser (Scan Control)	450
Tilt	451
TIN Volume	451
Toggle ScanWorld Leveled.....	452

Table Of Contents

Traverse	452
Trim Edges.....	455
Turn Viewpoint.....	455
Undo.....	455
Unextrude.....	456
Unfreeze Registration	456
Ungroup	457
Unify Clouds.....	457
Unify ModelSpace	458
Update Constraint	458
Update Initial Cloud Alignment.....	459
Update Original ModelSpace	459
Update Simulated Scanner Position	460
Use Fit Constraints	461
Use Parts Table	461
User Coordinate System.....	461
User Feedback.....	462
Verify Camera Calibration.....	462
Verify TIN	463
Vertex (From Intersections)	463
Vertex (From Pick Points)	464
View All	464
View Half-Space.....	464
View Interim Results	464
View Lock.....	465
View Mode	466
View ModelSpace	466
View Object As.....	466
View Slice.....	468
Virtual Surveyor.....	468
Volume	469
Zoom	469

Table Of Contents

Zoom (Lock)	470
-------------------	-----

Leica Geosystems HDS LLC provides complete documentation in an accessible HTML-based help system. The help system includes information on all of *Cyclone*'s tools, commands, and features.

Leica HDS Technical Support

Leica Geosystems HDS LLC

4550 Norris Canyon Road

San Ramon, CA 94583

925-790-2300

support@hds.leica-geosystems.com

hds.leica-geosystems.com

Copyright Notice

Copyright © 2005-2007 Leica Geosystems HDS LLC

All rights reserved.

The *Cyclone* documentation is copyrighted and all rights are reserved. Information in this document is subject to change without notice and does not represent a commitment on the part of Leica Geosystems HDS LLC. The document may not, in whole or in part, be copied, photocopied, reproduced, translated, or reduced to any electronic medium or machine-readable form without prior consent, in writing, from Leica Geosystems HDS LLC.

Trademarks

Cyrax is a trademark of Leica Geosystems HDS LLC. Other brands and their products are trademarks of their respective holders and should be noted as such.

Introduction

Cyclone Software Introduction

Most CAD and modeling programs construct geometry from only a few sample vertices and estimate the rest with design assumptions. Design or modeling packages aid the user in creating entirely novel models from a conceptual model or idea, often having little to do with reality.

Cyclone provides a high-performance environment for manipulating point cloud data captured by High Definition Surveying (HDS™) systems. *Cyclone* enables the user to accurately visualize, navigate, measure, and model 3D objects and scenes. See the *Quick Start* chapter for an introduction to the major concepts of the *Cyclone* software system.

Installation

Included Materials

CD-ROM
Cyclone 5.8
Installer
Program
On-line help
Documents

System Requirements

PROCESSOR

1.4 GHz Pentium M (Centrino), 2 GHz Pentium 4, or higher

MEMORY

512 MB RAM, 1 GB recommended

HARD DISK

10 GB Storage Space or more

VIDEO

SVGA or OpenGL accelerated graphics card

OPERATING SYSTEM

Microsoft Windows XP (Professional or Home Edition, SP1 or higher), Windows 2000 (SP2 or higher)

ACCESSORIES

Ethernet card (for licensing and communication with scanner)

Installing *Cyclone*

- For *Cyclone* to install correctly under Windows 2000 or Windows XP, use an account with Administrator privileges or equivalent which allows certain system files be installed or updated. Otherwise, *Cyclone* may not install correctly.
- Operation of *Cyclone* under Windows 95, Windows 98, Windows NT, or Windows ME is currently unsupported; if you are attempting to install *Cyclone* on a computer running Windows 95, Windows 98, Windows NT, or Windows ME, the installation will not proceed.

To Install *Cyclone*:

1. Insert the *Cyclone* CD into your CD drive. If **AUTORUN** is enabled, the installer runs automatically. Otherwise, open the Windows **Start** menu and select **Run**, then type the letter of your CD-ROM drive (for example, **D**) followed by **:\\ SETUP.EXE** to run the installer.
2. Follow the directions in each window to proceed with the installation.
- Default installation parameters should be reasonable for most systems; advanced users may select different settings.
- If you have an active firewall, you may see messages about programs attempting to access the network. For *Cyclone* to run properly, these executables, located in the *Cyclone* installation folder, should be permitted access to the network: **Cyclone.exe**, **CyraLicense.exe**, **ptserv32.exe**, **update41.exe**, and **LRCServer.exe** (located in the **HDS4500** sub-folder).
3. When the installation is complete, reboot your workstation if needed and proceed with obtaining a license to run *Cyclone* (see next section).

Obtaining your *Cyclone* License

When you install *Cyclone* for the first time, you must activate it with a license key. If you are installing over an existing *Cyclone* installation, it may not be necessary to re-install your license.

- Your workstation must have an Ethernet card properly installed and activated for the licensing to work.

To obtain your license:

1. Before installing *Cyclone*, run the **HostId.exe** program from the *Cyclone* CD.
2. Email the *HostID.txt* file to Leica Geosystems HDS at <mailto:license@hds.leica-geosystems.com> to request a license for your workstation.
3. Leica Geosystems HDS will create a license for all purchased components and return it to you by email. Save the entire email as "license.dat" in the installation directory (the rest of the email information will be ignored). If **license.dat** already exists, overwrite the existing file with the new file.

Uninstalling *Cyclone*

To uninstall *Cyclone* properly:

1. Open the Windows Control Panel.
2. Run the **Add/Remove Programs** applet. A list of installed programs appears.
3. Select *Cyclone* from the list.
4. Click the **Add/Remove** button to uninstall *Cyclone*.

Note: If a dialog appears asking to remove some components, it is best to leave them. They can be deleted later if they are truly unnecessary.

- Although the files installed by the *Cyclone* installation program are removed from your computer, any *Cyclone* data files or work files created will remain. You may delete those files at your discretion.
- If *Cyclone* does not appear in your **Add/Remove Programs** applet, re-run the original installer and choose the **Remove** option.

Quick Start

Launching *Cyclone*

When you launch the *Cyclone* application, the first window you see is the **Navigator**. The **Navigator** is the main window of the application and serves as a road map for navigating databases on your local machine and remote servers. The **Navigator** window is also used to connect to known scanners and to create objects such as Databases, Projects, ModelSpaces, ModelSpace Views, ScanWorlds, and Registrations. The **Navigator** window is used to manage and organize all of your data. It is also used to exit the application.

To launch *Cyclone*:

1. From the Windows desktop, click the **Start** button.
2. Point to **Programs**, and then point to **Leica HDS Cyclone 5.8**, then select **Cyclone**. The **Navigator** window appears.

User Configuration Management

User Configuration Management helps keeps your *Cyclone*-related files organized. The first time you run *Cyclone* or CloudWorx, the **Cyclone User Configuration Manager** appears. Follow the prompts to specify the locations for your working, library, and log files. Each user account has its own personal configuration.

The **Cyclone User Configuration Manager** can be run at any time, but you should first exit *Cyclone* and CloudWorx before launching the **Cyclone User Configuration Manager**.

To launch Cyclone User Configuration Manager:

1. From the Windows desktop, click the **Start** button.
2. Point to **Programs**, and then point to **Leica HDS Cyclone 5.8**, then point to **Utilities**, then select **Cyclone User Configuration Manager**. The **Cyclone User Configuration Manager** appears.

Preferences

In each *Cyclone* window, the **Preferences** command in the **Edit** menu opens the **Edit Preferences** dialog, which is used to manage preferences throughout the system. Preferences can be edited for the current session only or for the current and all future sessions.

SESSION

These preferences are non-persistent preferences that last for the duration of the current execution of *Cyclone* only.

DEFAULT

These preferences are persistent and applied to the current and all future sessions.

- Several settings throughout *Cyclone* have a range of between 0.0 and 1.0. In these cases, 0.0 denotes the lowest possible setting (none) and 1.0 denotes the highest possible setting. See the [Preferences](#) entry in the *Commands* chapter.

The Navigator Window

The Navigator window opens with three default folders - **Servers**, **Shortcuts**, and **Scanners**.

Servers Folder

The **Servers** folder maintains your list of database servers. You can add and remove servers to and from this folder. Your local machine is added as a server by default.

To add a server:

- From the **Configure** menu, select **Servers....** The **Configure Servers** dialog appears.
- Click **Add**. The **Add Server** dialog appears.
- In the **Server Name** field, type the Microsoft Network name for the server to be added, and then click **Add**. The **Configure Servers** dialog reappears and displays the newly added server.
- Click **Close**. The server is created.

Scanners Folder

The **Scanners** folder maintains a list of known scanners. Add a scanner by entering a name of your choice and a scanner address (IP address). Double-clicking the scanner icon opens a Scan Control window for that scanner.

To add an HDS4500 scanner:

- From the **Configure** menu, select **Scanners....** The **Configure Scanners** dialog appears.
- Click **Add**. The **Add Scanner** dialog appears.
- From the **Scanner Model** list, select the model name of the scanner that you are adding. For example, select **HDS4500** if the scanner is an HDS4500.
- In the **Scanner Name** field, type a logical name for the scanner you are adding (e.g., "Scanner", "My Scanner", "Scanner 123", etc.).
- The HDS4500 does not require an IP address, as it is using Firewire Technology.
- Click **OK**. The scanner is added.
- To close the **Configure Scanners** dialog and return to the **Navigator** window, click **Close**.

To add a ScanStation, ScanStation 2, HDS3000, or HDS6000 scanner:

- From the **Configure** menu, select **Scanners....** The **Configure Scanners** dialog appears.
- Click **Add**. The **Add Scanner** dialog appears.

3. From the **Scanner Model** list, select the model name of the scanner that you are adding. For example, select **HDS3000** if the scanner is a HDS3000, **HDS6000** if the scanner is a HDS6000, **ScanStation** if the scanner is a ScanStation, or **ScanStation 2** if the scanner is a ScanStation 2.
4. In the **Scanner Name** field, type a logical name for the scanner you are adding (e.g., "Scanner", "My Scanner", "Scanner 123", etc.).
5. In the **IP Address** field, enter the IP Address for the scanner you are adding. The IP address of the scanner is printed on the scanner's housing (ScanStation, ScanStation 2, and HDS3000) or in the built-in console (HDS6000).
6. Click **OK**. The scanner is added.
 - To close the **Configure Scanners** dialog and return to the **Navigator** window, click **Close**.

Shortcuts Folder

The **Shortcuts** folder maintains a list of shortcuts to objects that you frequently use. For instance, you may create a shortcut to a ModelSpace or a particular ModelSpace View that is several folders deep in the database, and thus, enable instant access without having to navigate down to the actual folder location in the database.

To create a shortcut:

1. In the **Navigator** window, select the object for which you need a shortcut.
2. From the **Create** menu, select **Shortcut**. The shortcut is added to the **Shortcuts** folder.

Using Links

Links are provided to make navigation easier. They allow you to create a shortcut to any Project, ScanWorld, ModelSpace, or Image as well as allowing you to reference any object in multiple places in the same database.

To create a link:

1. Select the Project, ScanWorld, ModelSpace, or Image to which you want to link.
2. From the **Edit** menu, select **Copy**.
3. Select a location for the link.
 - Links can be pasted into ScanWorlds, Scans, ModelSpaces, and Projects that are located in the same database as the original.
4. From the **Edit** menu, select **Paste Link**. The link appears in the selected location.
 - Links are identical to their source object; renaming a link also renames the source object.
 - Deleting a link does not delete the source object; deleting the source object does not delete any links to that object. Only when the source object and all links are deleted is the object removed from the database.

Object Hierarchy

In *Cyclone*, most objects "belong" to another object, depending on their positions in the hierarchy in the **Navigator** window.

- Servers contain Databases.
- Databases contain Projects.
- Projects contain ModelSpaces, ScanWorlds, Registrations, Images, imported files, and other (subordinate) Projects.
- ModelSpaces contain ModelSpace Views.
- By default, a ScanWorld contains a default ControlSpace and three folders – one each for ModelSpaces, Scans, and Images. Each of these folders may contain additional folders as well. ScanWorlds can also contain Projects. A ScanWorld may contain up to one frozen Registration.

Creating a Database

The first time you run *Cyclone*, you need to create a new database, or connect to an existing database, on the desired server. Databases can reside on your local server or any connected server across a network. A database stores all *Cyclone* data including Projects, ScanWorlds, ModelSpaces, etc..

- For details on using a database, see the [Databases](#) section chapter.

To add a database:

1. Select the server to which you want to add new or existing database.
2. From the **Configure** menu, select **Databases....** The **Configure Databases** dialog appears.
3. Click **Add**. The **Add Databases** dialog appears.
4. In the **Database Name** field, type a logical name as an identifier for your logical database.
 - The logical name is the name displayed in the **Navigator** window.
5. In the **Database Filename** field, type a file name for your database.
 - The database filename is the name used for the database file, which is saved in the *Cyclone*/Databases directory by default.
 - If you leave the **Database Filename** field empty, *Cyclone* bases the file name on the logical name you entered in the **Database Name** field.
 - You may select a new location for the database file by clicking the **Browse** icon to the right of the **Database Name** field.
6. Click **OK**. The database is created.
7. Click **Close** to close the **Configure Databases** dialog.

Scanning

Scanning is the process of using the HDS system to “sample” (take measurements from) a scene in the real world. Each individual measurement results in a 3D point. An entire scan results in collections of these 3D points called “point clouds”. These point clouds provide the basis for surface reconstruction or modeling, which is the main application of *Cyclone*.

The Scanning Process

Scanning Overview

1. From the **Navigator** window, add a scanner object to represent your scanner.
2. Open a **Scan Control** window and connect to the scanner.
3. Organize project folders for your scan data.
4. Adjust settings and target your scan.
5. Execute your scan.
 - For details on the scanning process, see the *Scanning* chapters.

Preparing for Scanning

The first time that you want to use a scanner, it is necessary to create a persistent entry for the scanner. Scanners are added from the **Navigator** window.

To add a scanner:

1. From the **Configure** menu, select **Scanners...** The **Configure Scanners** dialog appears.
2. Click **Add**. The **Add Scanner** dialog appears.
3. From the **Scanner Model** list, select the model of the scanner that you are adding.
4. In the **Scanner Name** field, type a logical name for the scanner you are adding.
5. In the **IP Address** field, enter the IP address for the scanner you are adding.
6. Click **OK**. The scanner is added.
 - To close the **Configure Scanners** dialog and return to the **Navigator** window, click **Close**.

The Scan Control Window

The **Scan Control** window is used to control the scanning process, adjust scan settings, specify scan targets, and organize the resulting scan data. Before scanning, it is necessary to connect to a known scanner (one that has been entered in the database) via the **Scan Control** window.

To launch a Scan Control window:

1. From the **Navigator** window, expand the **Scanners** folder.
2. Double-click the scanner you want. The **Scan Control** window appears with the **Select a Project** dialog open.

To connect to a scanner:

1. From the **Scanner** menu in the **Scan Control** window, select **Connect** or click the **Connect** icon. Connection progress is displayed in the **Scanner Status** bar at the bottom of the **Scan Control** window.
 - When a successful connection is made and the scanner is ready for commands from *Cyclone*, the **Scanner Status** bar at the bottom of the **Scan Control** window reads "Connected and Ready".

Scanner Simulation

Cyclone provides a scanner simulation for the purpose of familiarizing yourself with scan settings and procedures for the HDS2500. Any ModelSpace can be used to launch a **Scan Control** window for a scanner simulation, using the **Create Simulated Scanner** command. The ModelSpace from which the **Scan Control** window is launched serves as the scene to be scanned; the viewpoint in the ModelSpace is the initial viewpoint of the simulated scanner. The **Scan Control** window created by this command operates identically to a **Scan Control** window connected to a real scanner.

Registration

Registration is the process of integrating a project's ScanWorlds into a single coordinate system as a registered ScanWorld. This integration is derived by a system of constraints, which are pairs of equivalent or overlapping objects that exist in two ScanWorlds. The objects involved in these constraints are maintained in a ControlSpace, where they can be reviewed, organized, and removed; they cannot be moved or resized in the ControlSpace. The Registration process computes the optimal overall alignment transformations for each component ScanWorld in the Registration such that the constraints are matched as closely as possible.

The Registration Process

Registration Overview

1. From the **Navigator** window, create a Registration object in the project that you want to register.
2. From the **Registration** window, add the ScanWorlds that you want to register.
3. Add constraints between the component ScanWorlds.
4. Register the ScanWorlds, and then check and adjust constraints as needed.
5. From a successful registration, create a registered ScanWorld/Frozen Registration.
6. Create ModelSpaces from the registered ScanWorld and begin modeling.
 - For details on the registration process, see the *Registration* chapter.

The Registration Window

The **Registration** window consists of a main pane divided into three tabs and two [Constraint Viewer panes](#). The three tabs of the main pane are [ScanWorlds' Constraints](#), [Constraints List](#), and [ModelSpaces](#).

Constraint Viewer Panes

The **Constraint Viewer** panes are used to view constraints. The constraints are selected and centered in each pane.

ScanWorlds' Constraints Tab

The **ScanWorlds Constraints** tab displays each ScanWorld in the registration. Beneath each ScanWorld, a pairing with every other ScanWorld in the Registration is displayed. After constraints have been added, constraints between each ScanWorld pair are displayed. From the **ScanWorlds Constraints** tab, you can: select and view constraints; enable, disable, and delete constraints; set the Home ScanWorld; adjust the weight of individual constraints; and view information for each constraint.

Constraints List Tab

The **Constraints List** tab lists each constraint in the registration and displays current information about each constraint. Constraint information is organized into columns. Each column can be sorted. From the **Constraints** tab, you can: select and view constraints; enable, disable, and delete constraints; set the Home ScanWorld; adjust the weight of individual constraints; and view information for each constraint.

ModelSpaces Tab

The **ModelSpaces** tab displays each component ScanWorld's ModelSpace(s) and ControlSpace. The main applications of the **ModelSpaces** tab are viewing ModelSpaces and ControlSpaces and adding constraints. This tab can also be used to create a new ModelSpace using the objects from existing ModelSpaces after the registration has been frozen.

Modeling

Cyclone's primary application is the conversion of point clouds to CAD object-based line and surface models. The various processes employed to this end are referred to collectively as "modeling" and are applied via the **ModelSpace** window.

Modeling Terms

Several modeling-related terms are very similar, so becoming familiar with the following terms here may avoid confusion: [ModelSpace](#), [ModelSpace View](#), [ModelSpace Window](#), [Viewpoint](#)

ModelSpace

A ModelSpace is a collection of geometry and information about the organization of the geometry. A ModelSpace cannot be viewed directly. To interact with the objects in a ModelSpace, you must open an existing ModelSpace View, create a new ModelSpace View, or open a Temporary ModelSpace View.

ModelSpace View

A ModelSpace View is a collection of settings applied to the particular ModelSpace from which it was created. The settings controlled by the ModelSpace View include the initial viewing position and overriding settings for graphical viewing parameters, such as a user coordinate system. These settings are not shared among different ModelSpace Views. A ModelSpace View can be viewed and manipulated via a **ModelSpace** window.

Changes to the geometry or the layer organization of the geometry in one ModelSpace View affect the ModelSpace from which it was created and all other ModelSpace Views derived from the same ModelSpace.

- Multiple ModelSpace Views can be created from a single ModelSpace.

MODELSPACE WINDOW

A window that displays a ModelSpace View.

VIEWPOINT

A viewing position or “camera” that determines which part of the ModelSpace is displayed. Multiple viewpoints may be saved within a single ModelSpace View.

The ModelSpace Window

The **ModelSpace** window provides visual access to a ModelSpace View, primarily used to view, manipulate, and measure scan data or point clouds. ModelSpace tools allow you to segment the point clouds into smaller subsets, which can then be used to fit specific geometric shapes. Various geometric objects can also be inserted and manipulated. More complex shapes and text annotations can be added using the 2D drawing functions.

The **Status Bar** at the bottom of the **ModelSpace** window displays information about the current ModelSpace, including counts of selected objects, the layer of the object selected last, the distance between the last two picks (if available), the angle between the last two picks (if appropriate), and the progress of the loading of objects.

Basic Viewpoint Controls

Cyclone provides controls in **View** mode that allow you to manually rotate, pan, and zoom the viewpoint using the mouse. The viewpoint rotates around and zooms in/out from a focal point. The focal point is a point that is a certain distance in front of the center of the viewpoint. The distance is controlled by zooming the viewpoint. The focal point can be reset via **Seek** mode, and can be moved by panning the viewpoint.

- To rotate the viewpoint, drag around the focal point while in **View** mode.
- To pan the viewpoint, right-drag (press and hold the *right* mouse button while moving the mouse) in the desired direction while in **View** mode.
- To zoom the viewpoint, press and hold both mouse buttons and move the mouse down to zoom in and up to zoom out.

See the [Changing the Viewpoint](#) section in ModelSpace Navigation for more detailed information.

Scanning with the HDS6000 and HDS4500

The Scanning Process

Scanning can be accomplished in a variety of ways, but all scanning operations generally are divided into the following basic groups:

1. [Creating a Scanner](#) using the **Navigator** window.
2. [Organizing Scan Data](#) by selecting Projects and ScanWorlds (*Cyclone* prompts you to do this automatically when you open the **Scan Control** window).
3. [Preparing to Scan](#) by capturing a preview, targeting and adjusting scan settings.
4. [Executing the scan](#).

Creating a Scanner

The first time that you want to use a scanner, it is necessary to create a persistent entry for it. Scanners are added from the **Navigator** window.

To add an HDS6000 or HDS4500 scanner:

1. From the **Configure** menu, select **Scanners....** The **Configure Scanners** dialog appears.
2. Click **Add**. The **Add Scanner** dialog appears.
3. From the **Scanner Model** list, select the model name of the scanner that you are adding. For example, select **HDS4500** if the scanner is an HDS4500, or **HDS6000** if the scanner is an HDS6000.
4. In the **Scanner Name** field, type a logical name for the scanner you are adding (e.g., "Scanner", "My Scanner", "Scanner 123", etc.).
5. In the **IP Address** field, enter the IP Address for the scanner you are adding. The IP address of the scanner is displayed in the built-in console (HDS6000). The HDS4500 does not require an IP address, as it is using Firewire Technology.
6. Click **OK**. The scanner is added.
 - To close the **Configure Scanners** dialog and return to the **Navigator** window, click **Close**.

Organizing Scan Data

Before scanning can take place, you must first select or create a **Project** folder to store scan data, and then select or create a ScanWorld within that project folder. Individual scans and images are saved in this destination ScanWorld. For example, all of the scans and images that are taken from one scanner position would go into one ScanWorld. The scans and images from another scanner position would go into another ScanWorld, possibly in another project. The organization of the scans is managed by the user.

To select a project for your scan data:

1. When you open a **Scan Control** window, the **Select a Project** dialog appears. To select a different project when the **Scan Control** window is already open, from the **Project Setup** tab in the **Scan Control** window, select **Select a Project**....
- You can create a new project for your scan data by clicking **Create**, and then selecting the created project that appears.
2. Navigate to an existing project, and then click **OK**. Your destination project is selected.
 - When the scanner is not busy, you can select a different destination for the next set of scan data.

Note: If you choose a new project, a new ScanWorld is created and your previous ScanWorld is no longer the destination for subsequent scan data.

To select a ScanWorld for your scans and images:

1. From the **Project Setup** tab in the **Scan Control** window, select **Select a ScanWorld**. The **Select a ScanWorld** dialog appears.
- You can create a new ScanWorld in the currently selected Project for your scan data by clicking **Create**, and then selecting the created ScanWorld that appears.
2. Navigate to an existing ScanWorld, and then click **OK**. Your destination ScanWorld is selected.
 - When the scanner is not busy, you can select a different destination ScanWorld for the next set of scan data. Note that if the new destination ScanWorld is in another project, you must first select that project via the Select a Project dialog.

Preparing to Scan

Once the destination Project and ScanWorld folders are chosen, the next steps in a typical scanning process are:

- Many of these steps are optional, depending upon the requirements of your specific project.
1. [Capturing a Preview Scan Image](#) of your scanner's field of view.
 2. [Targeting](#) within the scene to select areas of interest.
 3. Adjusting Scan Settings to [specify the resolution](#), [probe the range](#) of your scan, and [set scan filters](#).

Tilt Sensor (HDS6000)

The Leica HDS6000 supports a tilt sensor, which detects how much the scanner “tilts” from perfectly level. The tilt sensor does *not* automatically correct the data to be in a level coordinate system. By default, the tilt sensor is enabled. To toggle the tilt sensor, select **Enable Tilt Sensor** from the **Scanner** menu. You may wish to disable the tilt sensor in some situations, for example, when the scanner is on an unstable platform such as a moving ship.

When enabled, the tilt sensor monitors the inclination of the scanner while the amount of tilt is within the sensor's operating range. If the scanner tilts beyond the sensor's operating range, the current scanner operation is automatically cancelled and you are informed that the HDS6000

must be re-levelled. You can also manually check or re-level the HDS6000 by selecting **Check/Re-Level HDS6000** from the **Scanner** menu.

If the tilt sensor state changes significantly (e.g., a large amount of drift, manual adjustment of the tribrach, or out-of-range), for data integrity, the current ScanWorld is automatically advanced to the next ScanWorld. You may re-select the previous ScanWorld for additional work, but be sure that the data are consistent.

Capturing a Preview Scan

Capturing a Preview Scan

You can capture a Preview Scan of the scene from the point of view of the scanner and display it in the **Scan Control** window. This Preview is useful for determining the field of view of the scanner, as well as for targeting areas of the scene to be scanned. While it can be very helpful for a number of reasons, having a valid Preview Scan in the **Scan Control** window is not required for targeting the scanner.

To capture a Preview Scan of the scanner's current point of view:

1. From the **Scanner Control** menu, click the **Preview** button. The scanner captures a Preview Scan corresponding to the scanner's view within the targeted region. The Preview Scan is optionally displayed in the Preview window via the [Show ModelSpace](#) command and is saved in the corresponding **ModelSpace** under the destination **ScanWorld**.
 - You can also capture Previews by selecting **Get Preview** from the **Preview** menu, or by clicking the **Get Preview** icon on the Scan Control toolbar.
 - If the position or orientation of the scanner has been changed, repeat step 1 to update the Previews. Also, a new ScanWorld should be selected for storing any new data. Use the **New Scanner Position** command to automatically create a new ScanWorld.

Targeting

With a valid Preview in the **Scan Control** window, you can decide which areas of the scene are of interest and how densely they should be sampled.

- To target the entire scene in the **Scan Control** window, select **Target All** from the **Scanner Control** menu. The scan target appears in the **Scan Control** window.
- To target only a portion of the scene, create a target region by dragging one of the **Target** cursors over the area in which you are interested.
- A number of preset target modes can be selected from the Presets dropdown. The horizontal and vertical settings for the selected mode are displayed in the fields below. When you enter targeting settings manually, the setting automatically changes to **Custom**.

Resolution

After determining the area of the scene that you want to scan, adjust the resolution for scanning. Four different Resolution settings are available to the HDS4500. The HDS6000 has an additional Resolution setting.

Probe

- To measure the range and horizontal and vertical direction from the scanner to a point, position the target cross-hair on the point and click the **Probe** button.

Scan Filter

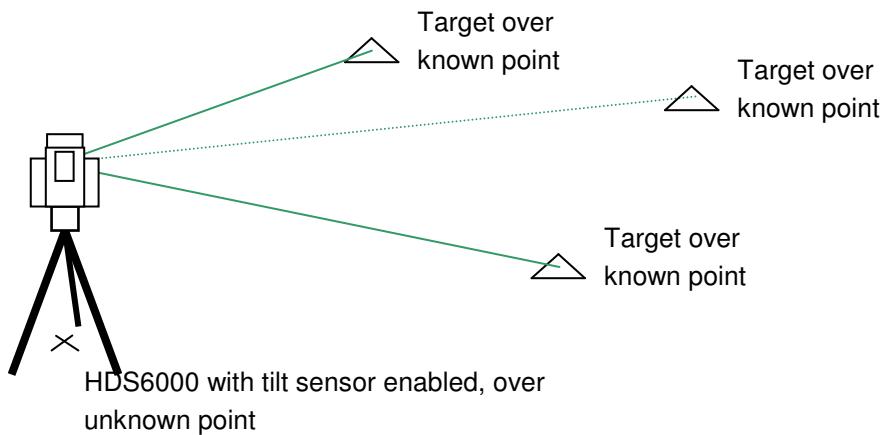
Range and/or intensity parameters can be set via the **Scan Filter** control. Points sampled outside these parameters are discarded without being recorded to the database.

Field Setup (HDS6000)

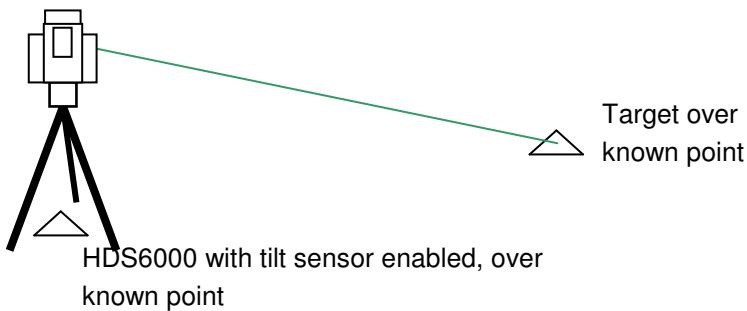
The default scanner coordinate system is based on an origin point within the scanner. The **Field Setup** panel allows you to take advantage of the HDS6000's tilt sensor and use conventional surveying techniques to establish a relationship between the scanner and a known or assumed coordinate system. The known or assumed coordinates are imported by the **Import Coordinate List** or **Import Coordinate List (Leica System 1200)** commands, or are added or replaced by the **Add/Replace Coordinate** command. See the corresponding entries in the *Commands* chapter for more information. Field Setup can also be used to establish the backsight and station position of a new Traverse (see below).

Field Setup Methods

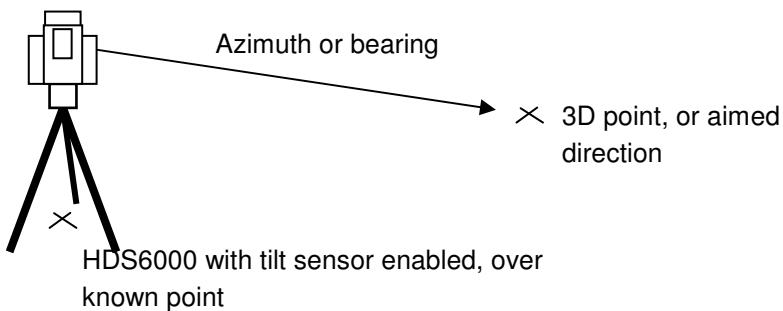
Resection from two or more targets over known coordinates:



Acquiring a target over a known coordinate from a scanner placed over a known coordinate:



Entering an azimuth or bearing to a 3D point or in the direction that the scanner is facing:

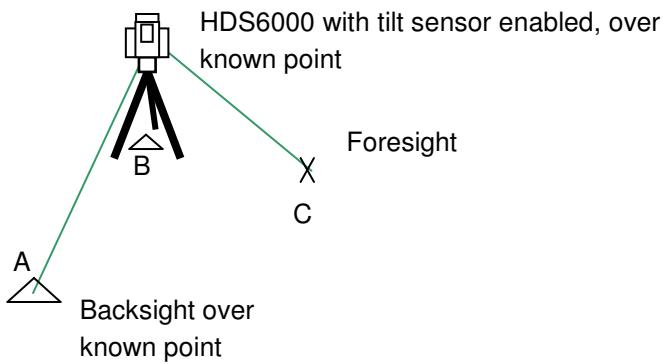


See the [Field Setup](#) entry in the *Commands* chapter for more information.

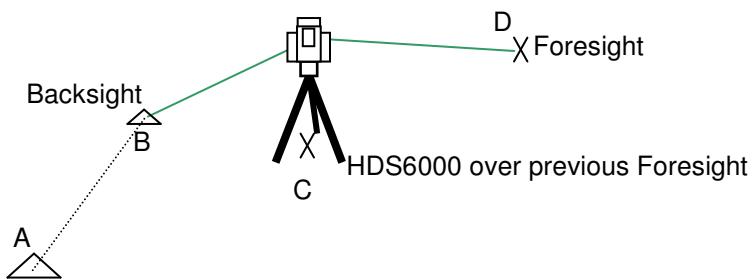
Traverse (HDS6000)

The default scanner coordinate system is based on an origin point within the scanner; the coordinates of the scanner in a global (known or assumed) coordinate system are not known. The **Traverse** control allows you to take advantage of the HDS6000's tilt sensor to use conventional surveying techniques to perform a traverse, establishing relationships between successive scanner positions within a known or assumed coordinate system. Because the tilt sensor detects if the scanner is not level, the traverse procedure does not require as many targets as regular target-based registration. Cyclone supports a linear traverse, open or closed. A closed traverse loops back on itself, whereas an open traverse does not.

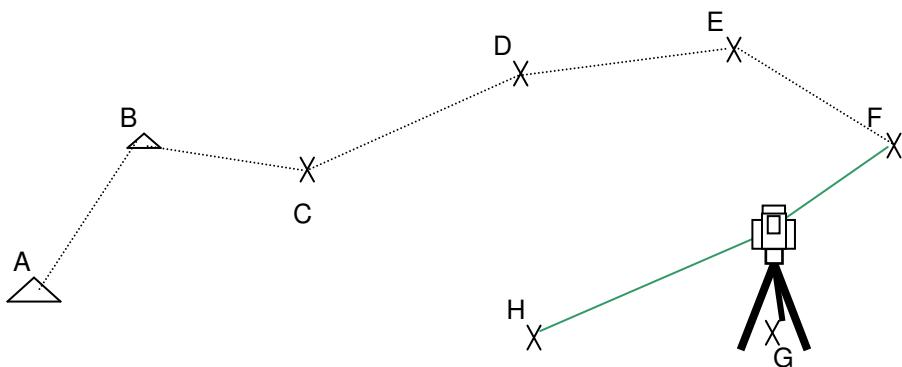
A traverse starts with the scanner over a known point. The user acquires a backsight over a known point, and a foresight, which may be over an arbitrary point. The known or assumed coordinates are imported by the **Import Coordinate List** or **Import Coordinate List (Leica System 1200)** commands, or are added or replaced by the **Add/Replace Coordinate** command. See the corresponding entries in the *Commands* chapter for more information.



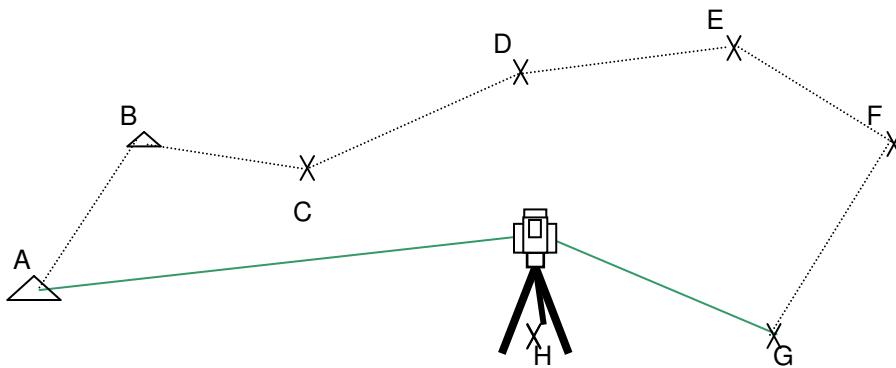
After the scan data for the current scanner position have been acquired, the user moves the scanner to the next position, over the same point as the previous foresight.



The pattern is repeated. Each time, the scanner is moved to the foresight and the previous position becomes the new position's backsight. The new position's foresight is where the scanner will be positioned next.



Eventually, the traverse can end when the foresight is over a known point. (Any foresight can be a known point without ending the traverse.) The traverse is a “closed traverse” if the foresight is the same as the original backsight.



The traverse workflow produces a Registration that tracks the backsight, station, and foresight of each scanner position (ScanWorld). This Registration can be frozen and used as-is, or may be supplemented with additional target and/or cloud constraints.

See the [Traverse](#) entry in the *Commands* chapter for more information.

Scanning

After defining the target region and adjusting the scan settings, the next step is to begin the scan.

In the **Scan Control** window, click the **Scan** button. The scan is saved in the Scans folder under the current ScanWorld.

Using a Scanner Script

It is often necessary to scan several areas within the same scene, each with different scanning parameters. In these cases it can be more efficient to use a scanner script. The scanner script functions allow you to queue up a “batch” of scans, and individually adjust targets and scan densities for each of the intended target areas before beginning the actual scans. This saves you the time it would otherwise take to wait for a scan to finish before setting up the next scan, or perform unattended scanning.

- For instructions on using a script, see the [Editor...](#) entry in the *Commands* chapter.

Importing HDS4500 and HDS6000 Scans

When *Cyclone* completes the HDS4500 or HDS6000 scan, the scan data captured is written to a ZFS format file. This file is stored in the Database Project folder under the HDS4500/HDS6000 Scan Directory.

Note: The **HDS4500/HDS6000 Scan Directory** is the folder location where HDS4500 and HDS6000 scan files are stored. This folder location is specified in the **HDS4500/HDS6000 Scan Directory** setting in the **Scan** tab of the **Edit Preferences** dialog.

For quick data verification and visualization in the field, use the [Re-Import Scans](#) command under the **Tools** menu in the *Cyclone* Navigator to import a reduced subsample of the total available ZFS point cloud data in the *Cyclone* database.

From the **Sampling** list, select the subsampling level (1/4 skips every two points, 1/9 skips every three points, 1/16 skips every four points etc) to import and create a reduced copy of the cloud in the ZFS file. All available point cloud data is imported by selecting 100% in the **Sampling** list.

Note: To remove an HDS4500 or HDS6000 ZFS scan file from the list of available scans in the **Re-Import Scans** dialog, toggle ON Archive. The ZFS scan file is moved from the Database Project Folder in the HDS4500/HDS6000 Scan Directory to the Archive subdirectory.

To make an Archived HDS4500 or HDS6000 scan available in the **Re-Import Scans** dialog, move the ZFS scan file from the Archive folder up into the parent Database Project Folder.

Scanning with the ScanStation, ScanStation 2, and HDS3000

The Scanning Process

Scanning can be accomplished in a variety of ways, but all scanning operations generally are divided into the following basic groups:

1. [Creating a Scanner](#) using the **Navigator** window.
2. [Connecting to a scanner](#).
3. [Organizing Scan Data](#) by selecting Projects and ScanWorlds (*Cyclone* prompts you to do this automatically when you open the **Scan Control** window).
4. [Preparing to Scan](#) by capturing images, targeting and adjusting scan settings.
5. [Executing the scan](#).
6. Merging sub-scans as necessary.

Creating a Scanner

The first time that you want to use a scanner, it is necessary to create a persistent entry for it in the **Navigator** window. Afterwards, the scanner is available each time you start *Cyclone*.

To add a ScanStation, ScanStation 2, or HDS3000 scanner:

1. From the **Configure** menu, select **Scanners....** The **Configure Scanners** dialog appears.
2. Click **Add**. The **Add Scanner** dialog appears.
3. From the **Scanner Model** list, select the model name of the scanner that you are adding. For example, select **HDS3000** if the scanner is a HDS3000, **ScanStation** if the scanner is a ScanStation, or **ScanStation 2** if the scanner is a ScanStation 2.
4. In the **Scanner Name** field, type a logical name for the scanner you are adding (e.g., "Scanner", "My Scanner", "Scanner 123", etc.).
5. In the **IP Address** field, enter the IP Address for the scanner you are adding. The IP address of the scanner is printed on the scanner's housing.
6. Click **OK**. The scanner is added.
 - To close the **Configure Scanners** dialog and return to the **Navigator** window, click **Close**.

Connecting to a Scanner

Connecting to a known scanner (one that has been entered in the database) is done from a **Scan Control** window.

To launch a Scan Control window:

1. From the **Navigator** window, expand the **Scanners** folder.
2. Double-click the scanner to which you want to connect. The **Scan Control** window appears.

To connect to a scanner:

1. From the **Scanner** menu in the **Scan Control** window, select **Connect** or click the **Connect** icon. Connection progress is displayed at the bottom of the **Scan Control** window.
- A successful connection is made and the scanner is ready for commands from *Cyclone* when the **Scanner Status** box at the bottom of the **Scan Control** window reads “Connected and Ready”.

Organizing Scan Data

Before scanning can take place, you must first select or create a **Project** folder to store scan data, and then select or create a ScanWorld within that project folder. Individual scans and images are saved in this destination ScanWorld. For example, all of the scans and images that are taken from one scanner position would go into one ScanWorld. The scans and images from another scanner position would go into another ScanWorld, possibly in another project. The organization of the scans is managed by the user.

To select a project for your scan data:

1. When you open a **Scan Control** window, the **Select a Project** dialog appears. To select a different project when the **Scan Control** window is already open, from the **Project Setup** tab in the **Scan Control** window, select **Select a Project**....
 - You can create a new project for your scan data by clicking **Create**, and then selecting the created project that appears.
2. Navigate to an existing project, and then click **OK**. Your destination project is selected.
 - When the scanner is not busy, you can select a different destination for the next set of scan data.

Note: If you choose a new project, a new ScanWorld is created and your previous ScanWorld is no longer the destination for subsequent scan data.

To select a ScanWorld for your scans and images:

1. From the **Project Setup** tab in the **Scan Control** window, select **Select a ScanWorld**. The **Select a ScanWorld** dialog appears.
 - You can create a new ScanWorld in the currently selected Project for your scan data by clicking **Create**, and then selecting the created ScanWorld that appears.
2. Navigate to an existing ScanWorld, and then click **OK**. Your destination ScanWorld is selected.
 - When the scanner is not busy, you can select a different destination ScanWorld for the next set of scan data. Note that if the new destination ScanWorld is in another project, you must first select that project via the Select a Project dialog.

Preparing to Scan

Once the destination Project and ScanWorld folders are chosen, the next steps in a typical scanning process described below. An alternate scanning method is described in the following *QuickScan* section.

- Many of these steps are optional, depending upon the requirements of your specific project.
1. [Capturing an Image](#) of your scanner's field of view.
 2. [Targeting](#) within the scene to select areas of interest.
 3. Adjusting Scan Settings to [specify the resolution](#), [probe the range](#) of your scan, [set scan filters](#), and compensate for conditions in the [atmosphere](#) that may affect the results of your scan.
 4. [Entering Station Data](#) to locate your scan data in a known or assumed coordinate system.

Dual-Axis Compensator (ScanStation, ScanStation 2)

The Leica ScanStation/ScanStation 2 supports a dual-axis compensator, which automatically corrects scanner data for scanner "tilt" (from perfectly level). By default, the dual-axis compensator is enabled. To toggle the dual-axis compensator, select **Enable Dual-Axis Compensator** from the **Scanner** menu. You may wish to disable the dual-axis compensator in some situations, for example, when the scanner is on an unstable platform such as a moving ship.

When enabled, the dual-axis compensator automatically corrects scanner data while the amount of tilt is within the compensator's operating range. If the scanner tilts beyond the compensator's operating range, the current scanner operation is automatically cancelled and you are informed that the ScanStation/ScanStation 2 must be re-leveled. You can also manually check or re-level the ScanStation/ScanStation 2 by selecting **Check/Re-Level ScanStation** from the **Scanner** menu.

If the dual-axis compensator state changes significantly (e.g., a large amount of drift, manual adjustment of the tribrach, or out-of-range), for data integrity, the current ScanWorld is automatically advanced to the next ScanWorld. You may re-select the previous ScanWorld for additional work, but be sure that the data are consistent.

Capturing an Image

You can capture an image of the scene from the point of view of the scanner and display it in the **Scan Control** window. This image is useful for determining the field of view of the scanner, as well as for targeting areas of the scene to be scanned. While it can be very helpful for a number of reasons, having a valid image in the **Scan Control** window is not required for targeting the scanner or for actual scanning. Note that the colors in the acquired image are mapped to the point cloud upon creation, so it is necessary to acquire an image prior to scanning in order have this visualization option.

Targeting

With a valid image in the **Scan Control** window, you can decide which areas of the scene are of interest and how densely they should be sampled.

- To target the entire scene in the **Scan Control** window, select **Target All** from the **Scanner Control** menu. The scan target appears in the **Scan Control** window.
- To target only a portion of the scene, create a target region by dragging one of the **Target** cursors over the area in which you are interested.
- A number of preset target modes can be selected from the Presets dropdown. The horizontal and vertical settings for the selected mode are displayed in the fields below. When you enter targeting settings manually, the setting automatically changes to **Custom**.
- To quickly and roughly target an area for image acquisition and subsequent scanning, click the **QuickScan** button. The **QuickScan** dialog prompts you to manually aim the scanner to establish left and right limits for scanning and image acquisition. You should aim the scanner by holding the plastic side covers; avoid touching the window area. Note that the top and bottom limits should be defined before acquiring images.

Resolution

After determining the area of the scene that you want to scan, adjust the range, sample spacing, and/or number of points.

Probe

- To measure the range and horizontal and vertical direction from the scanner to a point, position the target cross-hair on the point and click the **Probe** button.

Scan Filter

Range and/or intensity parameters can be set via the **Scan Filter** control. Points sampled outside these parameters are discarded without being recorded to the database.

Atmosphere

Environmental conditions (ambient temperature and atmospheric pressure) can affect the range computations by a small amount. Although any differences are minuscule over the ranges and environmental tolerances supported by the scanners, controls are provided to compensate for such atmospheric conditions. The default values should be acceptable to most users using the **Atmosphere** control.

- To adjust the atmosphere, enter values for ambient temperature and atmospheric pressure in the appropriate fields in the **Atmosphere** tab.

Station Data (HDS3000)

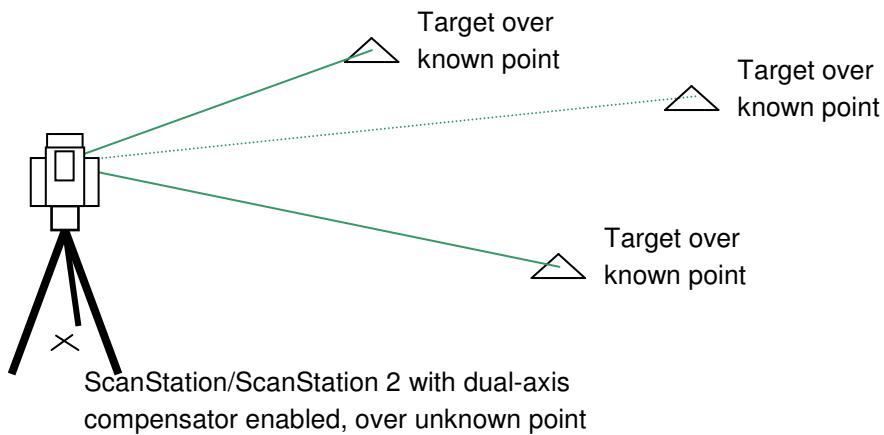
The default scanner coordinate system is based on an origin point within the scanner. The **Station Data** control allows you to enter known or assumed real-world coordinates to establish a relationship between the scanner and that coordinate system. See the [Station Data tab](#) entry in the *Commands* chapter for more information.

Field Setup (ScanStation, ScanStation 2)

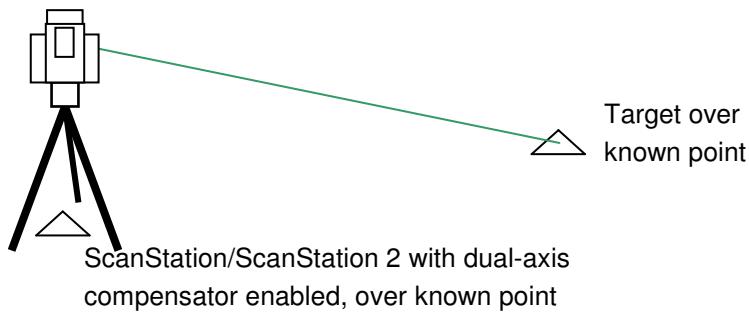
The default scanner coordinate system is based on an origin point within the scanner. The **Field Setup** panel allows you to take advantage of the ScanStation's/ScanStation 2's dual-axis compensator and use conventional surveying techniques to establish a relationship between the scanner and a known or assumed coordinate system. The known or assumed coordinates are imported by the **Import Coordinate List** or **Import Coordinate List (Leica System 1200)** commands, or are added or replaced by the **Add/Replace Coordinate** command. See the corresponding entries in the *Commands* chapter for more information. Field Setup can also be used to establish the backsight and station position of a new Traverse (see below).

Field Setup Methods

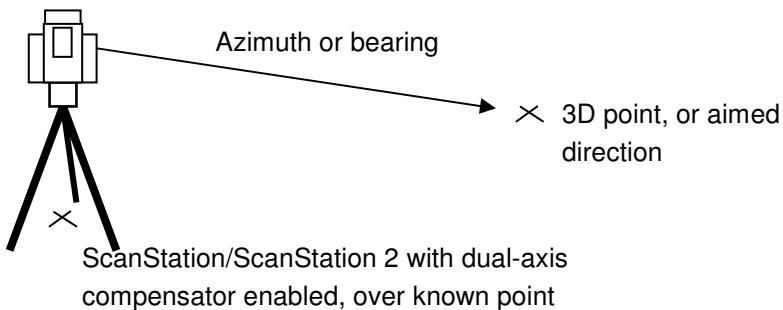
Resection from two or more targets over known coordinates:



Acquiring a target over a known coordinate from a scanner placed over a known coordinate:



Entering an azimuth or bearing to a 3D point or in the direction that the scanner is facing:

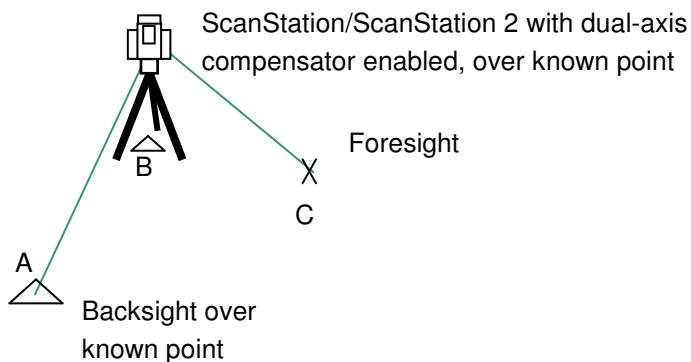


See the [Field Setup](#) entry in the *Commands* chapter for more information.

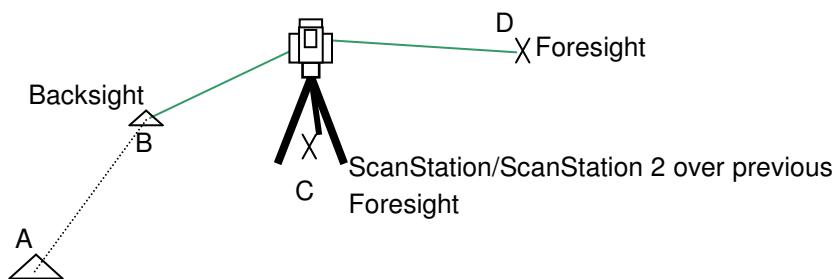
Traverse (ScanStation, ScanStation 2)

The default scanner coordinate system is based on an origin point within the scanner; the coordinates of the scanner in a global (known or assumed) coordinate system are not known. The **Traverse** control allows you to take advantage of the ScanStation's/ScanStation 2's dual-axis compensator to use conventional surveying techniques to perform a traverse, establishing relationships between successive scanner positions within a known or assumed coordinate system. Because the dual-axis compensator automatically corrects scan data for vertical, the traverse procedure does not require as many targets as regular target-based registration. Cyclone supports a linear traverse, open or closed. A closed traverse loops back on itself, whereas an open traverse does not.

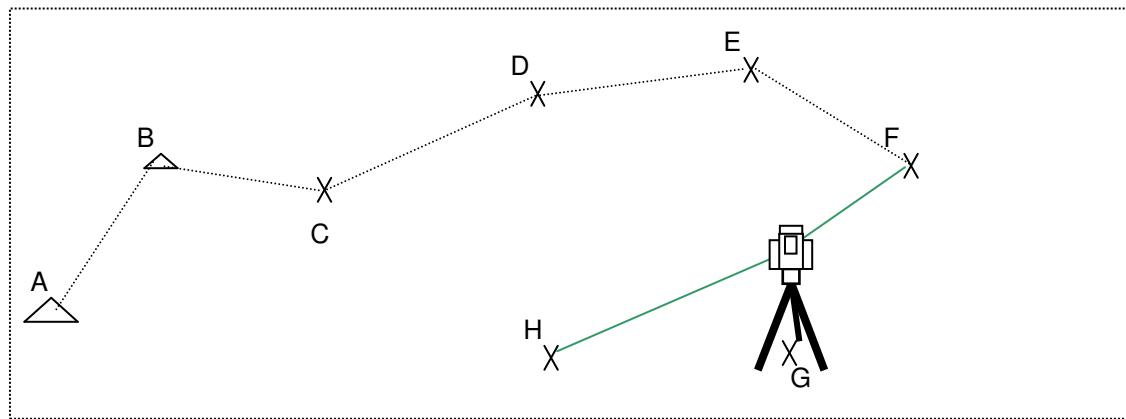
A traverse starts with the scanner over a known point. The user acquires a backsight over a known point, and a foresight, which may be over an arbitrary point. The known or assumed coordinates are imported by the **Import Coordinate List** or **Import Coordinate List (Leica System 1200)** commands, or are added or replaced by the **Add/Replace Coordinate** command. See the corresponding entries in the *Commands* chapter for more information.



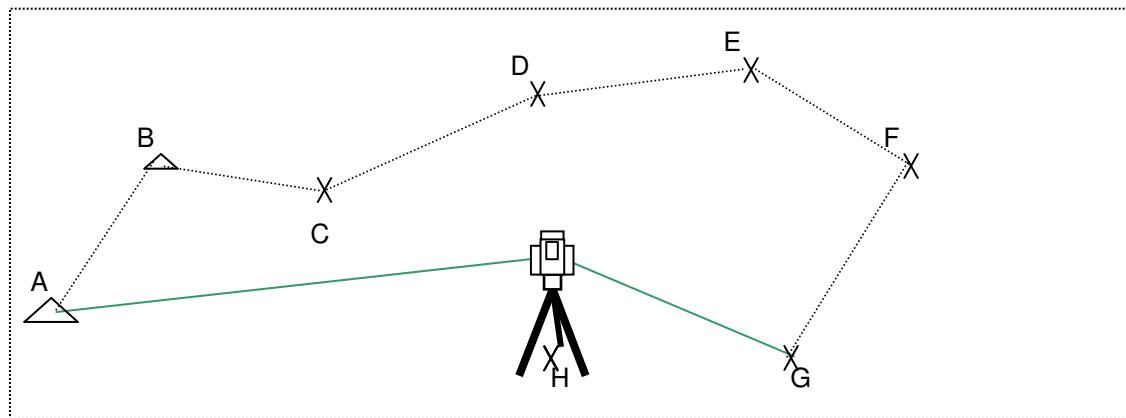
After the scan data for the current scanner position have been acquired, the user moves the scanner to the next position, over the same point as the previous foresight.



The pattern is repeated. Each time, the scanner is moved to the foresight and the previous position becomes the new position's backsight. The new position's foresight is where the scanner will be positioned next.



Eventually, the traverse can end when the foresight is over a known point. (Any foresight can be a known point without ending the traverse.) The traverse is a “closed traverse” if the foresight is the same as the original backsight.



The traverse workflow produces a Registration that tracks the backsight, station, and foresight of each scanner position (ScanWorld). This Registration can be frozen and used as-is, or may be supplemented with additional target and/or cloud constraints.

See the [Traverse](#) entry in the *Commands* chapter for more information.

QuickScan

QuickScan is a convenient way of quickly initiating a scan when your field of view settings do not need to be precise and your other Scan Control settings are already satisfactory. See the [Field-of-View Tab](#) command for instructions.

Scanning

After defining the target region and adjusting the scan settings, the next step is to begin the scan.

In the **Scan Control** window, click the **Scan** button. When a scan is complete, it is saved in the **Scans** folder under the current ScanWorld. During the scan, each increment of the number of points set via the **Max # Points Per Sub-Scan** preference in the **Scan** tab of the **Edit Preferences** dialog is saved as a new sub-scan. A sub-scan can be selected and manipulated while the scan continues. When the scan is complete, sub-scans should be merged via the **Merge Sub-Scans** command. With very large scans, this can be a time-consuming process. To save time, the previous sub-scans can be merged while subsequent scans are being captured.

- If *Cyclone* or the scanner fails during a scan, the scan data captured prior to the crash can be recovered. By default, it is located in the directory in which *Cyclone* is installed in .\Databases\temp\ scanX.scan (in the actual file, the X is replaced by the date and time of the scan). To recover the data, import it using the **Import** command in the **Navigator** window. Data can be imported directly into a project as a new ScanWorld by selecting the Project and then using the **Import** command. It can also be imported into an existing ScanWorld to apply colors from the last multi-image under the ScanWorld by selecting the ScanWorld and then using the **Import** command.

Note: Targets are useful as constraints in the registration process. If you are using either HDS Targets or spheres, a good time to acquire the targets is after the initial scan of the scene. Once acquired, targets are automatically added to the ControlSpace.

Using a Scanner Script

It is often necessary to scan several areas within the same scene, each with different scanning parameters. In these cases it can be more efficient to use a scanner script. The scanner script functions allow you to queue up a “batch” of scans, and individually adjust targets and scan densities for each of the intended target areas before beginning the actual scans. This saves you the time it would otherwise take to wait for a scan to finish before setting up the next scan, or perform unattended scanning.

- For instructions on using a script, see the [Editor...](#) entry in the *Commands* chapter.

Merging Sub-Scans

When creating a large scan, *Cyclone* creates a sub-scan each time it reaches a set increment of maximum cloud points per sub-scan (1.5 million points by default). This allows you to start manipulating point clouds efficiently as the remainder of the scan is being captured. After the full scan is complete, but before modeling or registration, the sub-scans should be merged. When working on a project that includes multiple large scans, it may be more efficient to merge sub-scans while subsequent scans are being captured, taking advantage of processing power not needed to carry out the scan.

- To merge sub-scans: In the **Navigator** window, select the appropriate ScanWorld. Then select **Merge Sub-Scans** from the **Tools** menu. All sub-scans within the selected ScanWorld are combined.
- To adjust the increment above which new sub-scans are created, use the **Max # Points Per Sub-Scan** preference in the **Scan** tab of the **Edit Preferences** dialog.

Scanning with the HDS2500

The Scanning Process

Scanning can be accomplished in a variety of ways, but all scanning operations generally are divided into the following basic groups:

1. [Creating a Scanner](#) using the **Navigator** window.
2. [Connecting to a scanner](#).
3. [Organizing Scan Data](#) by selecting projects and ScanWorlds (*Cyclone* prompts you to do this automatically whenever you open the **Scan Control** window).
4. [Preparing to Scan](#) by capturing images, targeting and adjusting scan settings.
5. Executing the scan.

Creating a Scanner

The first time that you want to use a scanner, it is necessary to create a persistent entry for it. Scanners are added from the **Navigator** window.

To add an HDS2500 scanner:

1. From the **Configure** menu, select **Scanners....** The **Configure Scanners** dialog appears.
2. Click **Add**. The **Add Scanner** dialog appears.
3. From the **Scanner Model** list, select the model name of the scanner that you are adding. For example, select **HDS2500** if the scanner is a HDS2500.
4. In the **Scanner Name** field, type a logical name for the scanner you are adding (e.g., "Scanner", "My Scanner", "Scanner 123", etc.).
5. In the **IP Address** field, enter the IP Address for the scanner you are adding. The IP address of the scanner is printed on the scanner's case.
6. Click **OK**. The scanner is added.
 - To close the **Configure Scanners** dialog and return to the **Navigator** window, click **Close**.

Connecting to a Scanner

Connecting to a known scanner (one that has been entered in the database) is done from a **Scan Control** window.

To launch a Scan Control window:

1. From the **Navigator** window, expand the **Scanners** folder.
2. Double-click the scanner to which you want to connect. The **Scan Control** window appears.

To connect to a scanner:

1. From the **Scanner** menu in the **Scan Control** window, select **Connect** or click the **Connect** icon. Connection progress is displayed at the bottom of the **Scan Control** window.
 - A successful connection is made and the scanner is ready for commands from *Cyclone* when the **Scanner Status** box at the bottom of the **Scan Control** window reads "Connected and Ready".

Organizing Scan Data

Before scanning can take place, you must first select or create a **Project** folder to store scan data, and then select or create a ScanWorld within that project folder. Individual scans and images are saved in this destination ScanWorld. For example, all of the scans and images that are taken from one scanner position would go into one ScanWorld. The scans and images from another scanner position would go into another ScanWorld, possibly in another project. The organization of the scans is managed by the user.

To select a project for your scan data:

1. When you open a **Scan Control** window, the **Select a Project** dialog appears. To select a different project when the **Scan Control** window is already open, from the **Project Setup** tab in the **Scan Control** window, select **Select a Project....**
- You can create a new project for your scan data by clicking **Create**, and then selecting the created project that appears.
2. Navigate to an existing project, and then click **OK**. Your destination project is selected.
- When the scanner is not busy, you can select a different destination for the next set of scan data.

Note: If you choose a new project, a new ScanWorld is created and your previous ScanWorld is no longer the destination for subsequent scan data.

To select a ScanWorld for your scans and images:

1. From the **Project Setup** tab in the **Scan Control** window, select **Select a ScanWorld**. The **Select a ScanWorld** dialog appears.
- You can create a new ScanWorld in the currently selected Project for your scan data by clicking **Create**, and then selecting the created ScanWorld that appears.
2. Navigate to an existing ScanWorld, and then click **OK**. Your destination ScanWorld is selected.
- When the scanner is not busy, you can select a different destination ScanWorld for the next set of scan data. Note that if the new destination ScanWorld is in another project, you must first select that project via the Select a Project dialog.

Preparing to Scan

Once the destination project and ScanWorld folders are chosen, the next step is usually to capture an image of the scene from the point of view of the scanner and display it in the **Scan Control** window. This image is useful for determining the field of view of the scanner, as well as for specifying areas of the scene to be scanned. While it can be very helpful for a number of reasons, having a valid image in the **Scan Control** window is not required for targeting the scanner or actual scanning. Note that the colors in the acquired image are mapped to the point cloud upon creation, so it is necessary to acquire an image prior to scanning in order have this visualization option.

To capture an image of the scanner's current point of view:

1. From the **Scanner Control** menu, select **Get Image**, or click the **Get Image** icon . The image appears in the **Scan Control** window and is saved in the **Images** folder under the parent ScanWorld. Additionally, a link to the current image is created beneath the scan to help keep track of which images go with which scans.
- The prevailing lighting conditions can greatly affect the quality and usability of the image. To compensate for the level of light in the scene, adjust the image exposure via the **Adjust Image Exposure...** command.
- It may take the scanner several seconds to acquire and transfer the image before it is displayed.
- If the position or orientation of the scanner is changed, select **Get Image** again to update the image. Also, a new ScanWorld should be selected for storing any new data. Use the **New Scanner Position** command to automatically create a new ScanWorld.

Adjusting Scan Settings

After determining the area of the scene that you want to scan, the range, sample spacing, and number of points settings can be adjusted. For an explanation of these settings, see the preceding *Scan Settings* section.

Using Targets

Targets are useful as constraints in the registration process. If you are using either HDS Targets or spheres, a good time to acquire the targets is after the initial scan of the scene. For target acquisition procedures, see the [Acquire Targets](#) command.

- Once acquired, Targets are automatically added to the ControlSpace.

Using a Scanner Script

It is often necessary to scan several areas within the same scene, each with different scanning parameters. In these cases it can be more efficient to use a scanner script. The scanner script functions allow you to queue up a “batch” of scans, and individually adjust targets and scan densities for each of the intended target areas before beginning the actual scans. This saves you the time it would otherwise take to wait for a scan to finish before setting up the next scan, or perform unattended scanning.

- For instructions on using a script, see the [Editor...](#) entry in the *Commands* chapter.

Camera Calibration

Cyclone's camera calibration tool is used to calibrate the camera integrated with the HDS2500. Camera calibration provides the following benefits for HDS2500 users:

- ◆ Correction of lens distortion of the image (i.e., straight lines in the scene now appear to be straight in the corrected image).
- ◆ Improvement of the accuracy of scan targeting on the image (i.e., when you target on the image, you generally get a scan of what you targeted, versus the hit-or-miss targeting on an uncalibrated image).
- ◆ Texture mapping of calibrated camera images on point clouds and meshes, providing clearer visualization and navigation.

The following notes apply:

- The HDS2500 requires at least one successful camera calibration before the additional camera functionality is enabled. The camera calibration may need to be re-calibrated periodically as well. See the *Calibrate Camera* entry in the *Commands* chapter for step-by-step instructions.
- After a camera has been successfully calibrated, point clouds and meshes can be texture mapped using the camera images associated with the source scan. Select the point cloud in the ModelSpace viewer, and then selecting **Image Texture Map** from the **View Object As** dialog.
- The camera calibration parameters are specific to each HDS2500. The calibration parameters from one HDS2500 should not be used with data acquired with another HDS2500.

Camera Calibration Process

The HDS2500 camera calibration process requires HDS2500 firmware version **4** or later. The process requires either a dimly lit (or unlit) room large enough to place the scanner at roughly 3-5 meters, then at roughly 6-10 meters from a wall (or ceiling), or an exterior wall against which you can calibrate at night. The view from the scanner to the wall should be unobstructed, and the wall should be a light, diffuse color (for example, white or beige); the process may not work with red, black, or shiny surfaces). Using a wall large enough to cover the entire scanner field of view at the second calibration distance (about 7 meters on each side if 10 meters is used for the second calibration distance, 4 meters on each side if 6 meters is used for the second calibration distance) is recommended. See the [*Calibrate Camera*](#) entry in the *Commands* chapter for step-by-step instructions.

Troubleshooting Camera Calibration

The HDS2500 provides a 480x480 pixel camera image, which is generally a lower resolution than most scans. The camera calibration accuracy is typically within 2 pixels; i.e., when texture mapped, a scan point is textured within 2 pixels of its ideal pixel. The combination of the image resolution and accuracy of camera calibration may result in slightly blurry texture maps. The accuracy of the calibration may be better near the center of the image than near the edges.

Parallax Errors

The HDS2500 laser scanner and internal camera do not share the same optical axis, resulting in the phenomenon of "parallax" (where the scene from the laser scanner's viewpoint is different from the scene from the camera's point of view). The effects of parallax may manifest as a portion of the texture map that appears to be mapped incorrectly onto the points. This may be more pronounced when there is a significant distance between foreground and background objects. As the difference in distances increases, the effects of parallax on the texture mapping increases. Typically, the background texture is projected onto a foreground object, or vice-versa, depending on the arrangement of the objects in the scene.

- When there is a mismatch of texture map and points, segment the point cloud into two pieces, separating the points that are incorrectly mapped from the points that are correctly mapped. Enable texture mapping only on the correctly mapped cloud. Depending on the severity of the mismatch due to parallax, this may or may not be necessary.

Advanced Use of Camera Calibration

When *Cyclone* captures an image from the HDS2500, the image is stored in uncorrected form. Images from a HDS2500 with a calibrated camera contain also the calibration parameters specific to that HDS2500. When the image is displayed, distortion correction is automatically applied (note that the Scan Control **Show ModelSpace on Image** command is incompatible with a calibrated image; selecting this option automatically hides the image). *Cyclone* supports the Import and Export of images and calibration parameters.

- An image, its calibration parameters, and the point cloud or geometry, may each be exported and then converted for use in a third-party application that provides additional texture mapping capabilities. See the *Export Calibration Info* entry in the *Commands* chapter for instructions on exporting calibration parameters. See the *Export Image* entry in the *Commands* chapter for instructions on exporting an image.
- The user can import an image and/or a set of calibration parameters and texture map the calibrated image on a point cloud or mesh. See the *Import Calibration Info* entry in the *Commands* chapter for step-by-step instructions.

Scanner Simulation

Cyclone provides a scanner simulation for the purpose of familiarizing yourself with scan settings and procedures. Any ModelSpace can be used to launch a **Scan Control** window for a scanner simulation by using the **Create Simulated Scanner** command. The ModelSpace from which the **Scan Control** window is launched serves as the scene to be scanned; the viewpoint in the ModelSpace is the initial viewpoint of the simulated scanner. The **Scan Control** window created by this command operates identically to a **Scan Control** window connected to a real scanner.

- To change the position of a simulated scanner to match the current viewpoint of the ModelSpace from which it was launched, use the **Update Simulated Scanner Position** command in the **Scanner** submenu of the original **ModelSpace** window.

Note: The original ModelSpace that serves as the scene to be scanned is not the same as the ModelSpace that can be opened from the **Scan Control** window in order to view the incoming points.

Registration

Introduction

Registration is the process of integrating a project's ScanWorlds into a single coordinate system, as a Registered ScanWorld. This integration is derived using a system of constraints, which are pairs of equivalent objects or overlapping point data that exist in two ScanWorlds. The objects involved in these constraints are maintained in a ControlSpace, where they can be reviewed, organized, and removed; they cannot be moved or resized in a ControlSpace (to ensure data integrity). The Registration process computes the optimal overall alignment transformations for each component ScanWorld in the Registration such that the constrained objects are aligned as closely as possible in the resulting ScanWorld.

The Registration Process

Registering a set of ScanWorlds can be accomplished in a variety of ways. The following series of steps represents a typical project. Your specific projects may be better suited to an order of operations that is slightly different from the following general guidelines:

Note: Before starting the Registration process, it is a good idea to check acquired targets for accuracy. It is also recommended that sub-scans are merged prior to registration (this applies to HDS3000, ScanStation, and ScanStation 2 data only). See the *Merge Sub-Scans* command for details.

1. From the **Navigator** window, [create a Registration](#) object in the project that you want to register.
2. Open the **Registration** window, and [add the ScanWorlds](#) that you want to register to the Registration.
3. [Establish a Home ScanWorld](#) containing a known coordinate system (if available).
4. [Add constraints](#) between the component ScanWorlds.
5. [Register the ScanWorlds](#), and then check and adjust constraints as needed.
6. From a successful registration, freeze the Registration and [create a registered ScanWorld](#).
7. [Create ModelSpaces](#) from the registered ScanWorld.

Creating a Registration Object

The first step in the registration process is to create a Registration object for the project that you want to register. This is done by selecting the desired project in the **Navigator** window, and then selecting the **Registration** command from the **Create** menu. The Registration object does not need to be created in the same project as the participating ScanWorlds, but must be in the same database.

Adding ScanWorlds

The next step in the Registration process is to add the ScanWorlds that you want to register to the Registration. After opening the Registration object, select participating (component) ScanWorlds via the **Add ScanWorld** command.

Using the ControlSpace

A ControlSpace is similar to a ModelSpace View. It is used as a container for all objects attached to its parent ScanWorld that are designated as “constraint” objects or possible constraint objects. The ControlSpace is used to review, organize, or remove objects; it is also used to add constraints manually. However, objects in the ControlSpace cannot be moved or resized. When an object is removed from the ControlSpace, it cannot be used in subsequent registrations.

When a ScanWorld is created, a default ControlSpace is automatically created and placed within the ScanWorld in the **Navigator** window. It cannot be moved, copied, or deleted. When a ScanWorld is added to a Registration, a ControlSpace linked to that specific Registration is created beneath the default ControlSpace. In this manner, a ScanWorld may contain a separate ControlSpace for each Registration that involves that ScanWorld. Operations in a Registration (i.e., adding or deleting constraints) affect only the corresponding subordinate ControlSpace.

Objects are generally added to a ControlSpace in one of four ways:

- ◆ When HDS Target or sphere targets are acquired via the **Scan Control** window.
- ◆ When a registration label is added to an object in a ModelSpace under the ControlSpace’s ScanWorld.
- ◆ When the **HDS Target** or **Sphere Target** commands are used to manually fit a HDS Target or target sphere to a cloud.
- ◆ When an object is copied to the ControlSpace directly from a ModelSpace using the **Copy to ControlSpace** command. The object in the ControlSpace is independent of the object in the ModelSpace from which it originated.

Objects in a ControlSpace can be organized into layers or changed in appearance without affecting their counterparts in the ModelSpace.

- To activate/deactivate a constraint object from within a ControlSpace, toggle the object’s visibility. (This can be done by placing the object in a layer and turning the layer’s visibility OFF.) Only visible objects in the ControlSpace are used in a Registration.

Hint: For trial and error registration, it may be useful to keep several versions of a constraint object (organized by layers, for example) in the ControlSpace and hide all but one for a given trial.

Objects used as constraints in a frozen Registration may not be deleted from the ControlSpace until all constraints which use the object have first been deleted from the Registration. Note that constraints cannot be deleted if a Registration is frozen; unfreeze the Registration in order to delete constraints.

Note: When using a database from an older version of *Cyclone*, running the **Optimize** command from the **Navigator** window on the database automatically creates a default ControlSpace for each ScanWorld. These ControlSpaces are populated with all known constraints and objects with registration labels. Also, a subordinate ControlSpace is created for each Registration that involves the ScanWorld.

Establishing the Home ScanWorld

The home ScanWorld defines the reference coordinate system for the ScanWorld created from a successful registration. By default, the first ScanWorld added to the Registration is set as the home ScanWorld. However, the home ScanWorld can be re-selected using the **Set Home ScanWorld** command.

- The Home ScanWorld is displayed in **bold** face.
- If you have a known coordinate system, set the ScanWorld containing that coordinate system as the Home ScanWorld. A known coordinate system can be established within a ScanWorld via the **Set ScanWorld Coordinate System** command or by creating a new ScanWorld and inserting known coordinates.

Making a ScanWorld “Leveled”

Depending on the source of the data in a ScanWorld, the ScanWorld may be considered to be “leveled”. That is, the data are known to be in a coordinate system where “vertical” is “true vertical” and “up” is in the positive direction of elevation. A ScanWorld known to be “leveled” is registered such that its “up” vector remains the same.

To manually toggle a ScanWorld’s “leveled” property, select **Toggle ScanWorld Leveled** from the **ScanWorld** menu. This command should be used with care, since an incorrect property may adversely affect the registration results.

Adding Constraints

Cyclone provides several methods for adding constraints. If you have modeled a sufficient number of objects that exist in multiple ScanWorlds, *Cyclone* can usually match the modeled objects to form your registration constraints without using registration labels or manually adding constraints, provided that you have added those objects to the ControlSpace for each ScanWorld.

Auto-Add Constraints

During the **Auto-Add Constraints** process, *Cyclone* first incorporates all previously added Registration pairs, and then searches for objects with matching Registration labels. *Cyclone* also searches the ControlSpaces for geometrically consistent objects that can be used in registration constraints (if the **Auto-Add Constraint Tolerance** is set at a value greater than zero in the **Registration** tab of the **Edit Preferences** dialog).

Note: The search only looks through objects in the ScanWorld’s subordinate ControlSpace corresponding to the Registration.

Given that a large set of constraints is unlikely to occur randomly, the search picks the largest mutually consistent constraints from the possible constraints. If the **Auto-Add Constraints** command gives an incorrect result (this would usually be discovered during the actual Registration by a registration failure or by large errors), undo the last operation, manually add one or two constraints, and then try again. The **Auto-Add Constraints** process is performed from the **Registration** window using the **Auto-Add Constraints** command.

- After using the **Auto-Add Constraints** command, it is recommended that you check to make sure that constraints have not been added between ScanWorlds known to not overlap.

Constraints occurring in this manner should be deleted.

Matching Registration Labels

When two ScanWorlds' ControlSpaces contain objects with the same Registration label, a constraint is automatically created, relating each of these object pairs. In most cases, constraints that result from Registration labels come from HDS Targets that have been placed in overlapping areas of scans for the specific purpose of using them for Registration. Some HDS Targets may have been imported from a reference coordinate system (e.g., survey data). Registration labels are created during the modeling process via the **Add/Edit Registration Label** dialog in the **ModelSpace** window.

Manually Adding Constraints

Manually adding a constraint establishes that two objects are equivalent. For example, a constraint formed from two patches found in two different ScanWorlds declares that the patches should lie in the same plane after the two ScanWorlds have been registered.

Constraints are manually added using the **Add Constraint** command. The process involves viewing both ScanWorlds' ControlSpaces and selecting the objects to be used in the constraint. Note that both objects must be copied to their respective ControlSpaces before a constraint can be specified.

The types of geometry that can be used for constraints include: Patch, Line, Vertex, Sphere, Cylinder, and Steel Sections.

Register

After the constraints have been added, the next step is to register the ScanWorlds using the **Register** command in the **Registration** menu. The **Registration** command computes the optimal alignment transformations for each ScanWorld in the Registration so that all constrained objects are aligned as closely as possible. Statistics for the successful Registration can be viewed using the **Registration Diagnostics** command in the **Registration** menu. More constraints than are strictly required can be added and may improve results or be used to help check the registration (i.e., if the extra constraints are disabled, their errors are still computed; they should have a low error under a "good" registration, even if they were not used to compute the registration).

Adjusting Constraints

Fine-tuning a Registration for maximum accuracy is accomplished by reviewing and adjusting constraints. After a successful registration, the overall accuracy of the registration can be viewed from the **Registration Diagnostics** dialog, where the error of each constraint is displayed in the **Constraints** tab of the **Registration** window. The **Error** column displays the error for each constraint and can be used to sort constraints by error magnitude by clicking the column heading.

When a constraint is found with a larger than average error, there are several possible courses of action. First, the constraint can be visually checked using the **Show Constraints** command, which displays the two objects involved in the constraint in the Constraint viewers. When it is determined that a constraint's error is unexpectedly high, it can be disabled (see the *Disable* entry in the *Commands* chapter) or deleted, or its influence can be decreased (see the

Set Weight entry in the *Commands* chapter). The constraint can also be disabled by making the constraint object invisible in the ControlSpace. Possible reasons for a high error include: incorrect constraint, poorly-fit objects, poorly-sampled objects, or some motion of the scanner during data acquisition. However, a correct constraint may have a high error if the registration favored other, incorrect constraints (e.g., if the incorrect constraints all somehow "agreed" with each other).

Note: The practice of decreasing the weight of all constraints with higher than average error is not recommended as it can lead to lower overall registration accuracy. Decreasing the weight of a constraint should only be done if you have reason to believe that the measurement of one or both of the objects in the constraint is less accurate than other measured objects.

If the **Auto-Update** command in the **Registration** menu is ON, the registration is updated after each change to the system of constraints. Otherwise, use the **Register** command in the **Registration** menu to update the registration when ready.

Creating a Registered ScanWorld and Freezing the Registration

After a successful registration, the **Create ScanWorld/Freeze Registration** command can be used to create a single ScanWorld that includes the registered (component) ScanWorlds in a unified (common) coordinate system. It also locks all constraints in place. At this point constraints can no longer be modified unless the Registration is unfrozen via the **Unfreeze Registration** command. The Registration object is relocated beneath the new ScanWorld. The new ScanWorld's default ControlSpace consists of all of the objects in each ControlSpace linked to this Registration for each ScanWorld used in the Registration.

Creating a ModelSpace from the Registration

There are two ways to create a ModelSpace from a Registration, depending on the desired content of the resulting ModelSpace.

- A. The ModelSpace is created using the **Create | ModelSpace** command in the **Navigator** window. A ModelSpace created in this manner includes only the original scan data and target objects from all of the registration's component ScanWorlds. The current preferences determine which objects are included. These settings are found in the **ModelSpace** tab of the **Edit Preferences** dialog. None of the modeling done in any of the component ScanWorlds' individual ModelSpaces is included in the new ModelSpace. The same result is obtained by using the **Create ModelSpace** (or **Create and Open ModelSpace**) command under the **Registration** menu in the **Registration** window.
- B. The ModelSpace is created from the **ModelSpaces** tab of the **Registration** window by selecting all ModelSpaces containing objects you want to include in the resulting ModelSpace, and then using the **Create ModelSpace** command. The ModelSpace created in this manner includes ModelSpace data from each of the selected ModelSpaces.
 - In each case, individual scans can be added to the ModelSpace by dragging the desired scan onto the ModelSpace in the **Navigator** window. Scans may also be restored using the **ScanWorld Explorer**.
 - In a ModelSpace containing a registered set of scans and a large number of point clouds, performance is improved by the cloud unification process. See the [Unify Clouds](#) entry in the *Command Reference* chapter.

Checking the Registration

Mistakes can still happen, even with acceptable registration metrics. When QC (quality control) points are available, the **Elevation Check** dialog can help discover mistakes by measuring the vertical deviation of each QC point from the registered cloud points in its neighborhood. The QC points are typically measured on arbitrarily-selected ground points with an independent survey instrument and placed in the same coordinate system as the Home ScanWorld. With a good registration, the QC points will be close in elevation to the registered point cloud. See the *Elevation Check* entry in the *Command Reference* chapter.

Cloud Registration

Cloud Registration supports the creation and management of *cloud constraints* between overlapping point clouds and meshes ("clouds" and "meshes" are used interchangeably in this topic), without the need for targets. The overall registration process remains the same (see the *Registration* topic in this chapter); the user may also create cloud constraints in addition to the geometric constraints between HDS Targets, spheres, cylinders, planes, etc. Cloud constraint commands are found in the **Cloud Constraint** menu of the **Registration** window.

Cloud Constraints

A cloud constraint is a user-defined constraint between two ScanWorlds, which indicates that the point clouds in the ControlSpace of the first ScanWorld overlap with the point clouds in the ControlSpace of the second ScanWorld. Just as an object-object constraint tells the registration to align the objects, a cloud constraint tells the registration to align the overlapping areas of two or more point clouds.

It is possible to register two ScanWorlds together with a single cloud constraint if the overlapping points constrains the registration in all six degrees of freedom (three translation directions and three rotation angles); e.g., if the overlap is in a corner. However, two scans may only overlap in a single plane; in this case, the cloud constraint behaves much like a plane-plane constraint, with the unconstrained degrees of freedom inherent in a plane-plane constraint.

Creating a Cloud Constraint

Cloud constraints do not differentiate between clouds and meshes. One ControlSpace can have a mesh while another has a cloud (or any combination thereof). This means you can use cloud constraints to align a cloud with a mesh model. For best results, the mesh should be sampled similarly to the density of the point cloud. The three methods of creating a cloud constraint are: 1. assisted creation via the **Cloud Constraints Wizard**; 2. manual creation via the **Add Cloud Constraint** command; and 3. addition of cloud constraints to an existing registration ("post registration") or to geo-referenced ScanWorlds using either the **Add Cloud Constraint** command or the **Auto-Add Cloud Constraints** command.

Note: Cloud constraints are not created by the **Auto-Add Constraints** command.

This is the *recommended* method for creating cloud constraints. The Cloud Constraints Wizard guides you through the entire process of creating cloud constraints and can save considerable time compared to the other methods. First, specify which ScanWorld pairs are overlapping, and

then the Wizard steps through the process of creating a cloud constraint for each specified pair of ScanWorlds. See the *Cloud Constraints Wizard* entry in the *Commands* chapter for detailed instructions. The Cloud Constraints Wizard offers the following advantages:

- It automatically copies all scan clouds (except HDS Target clouds) into the ControlSpace for the selected ScanWorlds.
- It automatically opens the ControlSpaces side-by-side in the Constraint viewers, and advances to the next ScanWorld pair after the user has picked corresponding points and pushed the Cloud Constraints Wizard's **Create Constraint** button.
- It provides a quick way to check for overlap between two ScanWorlds. Pressing the eyeglass button in the Cloud Constraint Wizard's grid control displays the corresponding ScanWorlds' ControlSpaces in the Constraint viewers.

Manual Creation

The manual process is similar to creating a sphere-sphere constraint:

1. In the Constraint viewers in the **Registration** window, show two ModelSpaces or ControlSpaces that contain overlapping clouds.
2. Pick three or more matching points on the two clouds. (See below for hints on selecting these points.)
3. Select the **Add Cloud Constraint** command.
 - If the picked points match closely enough, the cloud constraint is created and the selected clouds are copied as needed into their respective ControlSpaces. (See the *Add Cloud Constraint* entry in the *Commands* chapter for detailed instructions.)

Post Registration: Adding Cloud Constraints to Improve an Existing Registration or Geo-referenced Registration

If you have pre-aligned ScanWorlds (ScanWorlds that are already in the same coordinate system either through registration using objects or geo-referencing), you can add cloud constraints and re-register the ScanWorlds to optimize the registration results. Since the ScanWorlds have already been pre-aligned or registered, each cloud constraint added uses the existing registration to derive its initial alignment. This does not require you to open each ModelSpace to specify the initial alignment hints via pick points.

There are two methods for adding cloud constraints to the registration in this case:

1. Using the **Add Cloud Constraint** command: This process requires only that you select two ScanWorlds that share a common coordinate system and overlap, and then select the **Add Cloud Constraint** command. Note that you must ensure that the ControlSpaces of these ScanWorlds already contain the necessary point clouds. See the *Copy to ControlSpace* entry in the *Commands* chapter for detailed instructions.
2. Using the **Auto-Add Cloud Constraints** command: This command will search all pairs of ScanWorlds for overlapping point clouds. If overlap is found, a cloud constraint is created and the alignment is optimized. The user can reduce the number of ScanWorld pairs searched by selecting one or more ScanWorlds. If a selection exists, the command will create cloud constraints only for ScanWorlds that overlap the selected ScanWorlds.

In general, object-object constraints cover a limited area of the scene, while point cloud overlap may cover a far greater extent. Thus, adding a cloud constraint can improve a registration in regions where shared targets or objects do not exist but where overlap between different scans does exist.

Picking Points for Initial Alignment Hints

The initial alignment hint should be as accurate as possible to produce faster and more reliable **Optimize Cloud Alignment** results. The **Optimize Cloud Alignment** command can fail if the picked points do not provide good correspondences. When picking the *minimum* of three points, use the following guidelines:

- Pick points that are easily identified. Good choices include: corners, intensity markings (such as a painted sign on a wall), the top of a pole, and where a pole enters the ground. Bad choices include: the middle of a flat wall, and a smoothly curving surface without an identifiable feature.
- Pick points that match the pick points in the other ScanWorld as closely as possible. We recommend that your pick points in one cloud match their corresponding pick points in the other cloud within four inches, which is the default **Cloud Reg Max Search Distance**. You may increase this setting (see the *Registration* tab of the *Preferences* entry in the *Commands* chapter), but doing so may decrease the speed and reliability of the **Optimize Cloud Alignment** command.
- Avoid geometrically symmetric picks. The three points should not form a symmetric shape (such as a line, equilateral triangle, or isosceles triangle). In these cases, the initial alignment may rotate and still match the three points, producing unexpected results. In this case, add a geometrically suitable fourth point to eliminate the symmetry.
- Spatially distribute the picks over a large volume. Pick three or more points that form as large a triangle as possible. Look at the entire area of overlap in the two clouds and try to pick three points or more that cover as much of that area as possible (i.e., pick one point near the front of the overlap, one near the back of the overlap and one somewhere in between). This ensures that the initial alignment aligns all the overlapping areas as closely as possible.

Note: The order of the picks in each viewer does not matter, as long as there are at least three asymmetric picks.

Using Cloud Constraints

The optimal alignment of the two point clouds used in a cloud constraint must be determined before the cloud constraint can be used in global registration. When creating a cloud constraint, three or more matching pick points (called *Initial Alignment Hints*) on the clouds are specified in the **Registration** window. These hints are used to determine the initial alignment of the clouds and to create the cloud constraint. This initial alignment is used as a starting point for the optimal alignment. Note that if the initial alignment is not accurate enough (e.g., the initial alignment hints are not within the **Cloud Reg Max Search Distance**), the optimal alignment search may fail.

Optimal alignment is computed by the **Optimize Cloud Alignment** command. (See the *Optimize Cloud Alignment* entry in the *Commands* chapter for details.) However, if you are confident that your cloud constraints are set up properly, you can immediately select the **Register** command. Any cloud constraints marked as **not aligned** are aligned before global registration is performed.

Optimize Cloud Alignment Parameters

In the **Registration** tab of the **Preferences** dialog, the **Cloud Reg Subsampling Percentage** parameter specifies a speed-versus-accuracy trade-off in the performance of the **Optimize Cloud Alignment** command. The default is 3%, which is effective and efficient for typical scan cloud sizes (i.e., a point cloud with one million points). In general, a value of 100% may give results up to two times as accurate as a value of 3%, but may take approximately ten times as long.

The recommended process is to:

1. Align all cloud constraints using the default 3% parameter and check that global registration gives a satisfactory result.
2. If you then desire more accuracy, change the parameter to 100% (or other value). Select all the cloud constraints, and then select the **Optimize Cloud Alignment** command.

Cloud Constraint Status

In the **Constraint List** tab of the **Registration** window, the **Error Vector** column for cloud constraints shows one of the three following values:

Create Sections Dialog Options

NOT ALIGNED

The **Optimize Cloud Alignment** command has not been run successfully on the cloud constraint.

ALIGNED

The **Optimize Cloud Alignment** command has been executed satisfactorily, and the cloud constraint constrains all six degrees of freedom between the two ScanWorlds.

ALIGNED/UNDERCONSTRAINED

The **Optimize Cloud Alignment** command has been executed satisfactorily, but the geometry of the overlap area in the two clouds may not constrain all six degrees of freedom (e.g., if the overlap area is a flat plane, it would only constrain 1 translation and 1 rotation degree of freedom). In these cases, you may want to visually verify that it is indeed underconstrained or if the initial alignment guess was inaccurate.

Cloud Alignment Accuracy

Although there is no fail-safe method of determining if the cloud alignment result succeeded in finding the optimum alignment (within accuracy limits of the data), the following methods can be used to check for accuracy.

ALIGNMENT RMS ERROR

One of the first things to look at is the RMS (root mean square) error of the alignment (in the **Error Vector** column in the **Constraint List** tab). If the RMS error value is in the 1 cm range for HDS data, the alignment is likely to be good. HDS3000, ScanStation, and ScanStation 2 data has an RMS of 6 mm in general, but since the overlap measurement is often imprecise,

the alignment RMS is generally above 6 mm depending on the geometry of the scene. For example, if the scene contains large smooth surfaces, the overlap measurement is very good, and the RMS may be 6mm or lower. However, for scans of more complex geometry such as a leafy tree or a field of grass, the RMS will be higher since the overlapping points may not always be from the same surface (e.g., a nearby leaf or blade of grass).

ERROR HISTOGRAM

The second thing to look at is the shape of the error histogram. When observing the histogram during the **Optimize Cloud Alignment** procedure, the histogram normally begins as a horizontal line, and then gradually tightens up to the shape of a vertical half-bell curve. The half-bell curve shape indicates that the alignment has been optimized properly. The width (one-sigma) of the half-bell curve should be approximately matching the accuracy of the scanner used to scan the cloud (6 mm for the HDS2500, HDS3000, ScanStation, and ScanStation 2). A histogram that remains flat indicates that the initial guess was too far away for the **Optimize Cloud Alignment** algorithm to align the clouds.

VIEW THE ALIGNMENT

Finally, viewing the results of the alignment will give you a good sense of the accuracy. Select a cloud constraint and then select the **View Interim Results** command to open a temporary ModelSpace containing the aligned objects and clouds from the ControlSpaces of the two ScanWorlds. Focus on corners and surfaces to check the alignment. If you see any gaps between surfaces that were stationary during the scans, you should use the **Update Initial Cloud Alignment** command and optimize again. The **Show Overlapping Points** function in the **Show Diagnostics** dialog is another tool for viewing the alignment. The **Show Overlapping Points** function creates a temporary ModelSpace containing four point sets: blue, dark blue, green, and dark green. The dark blue and dark green points are the points in ScanWorlds A and B respectively that are not found to be overlapping. The remaining colored points are the overlapping points. This gives you a better idea of what data the **Optimize Cloud Alignment** method is working with and what it considers overlapping. If you delete or hide the dark blue and dark green points, you are left with the overlapping regions of the two clouds. If you see that any obvious surfaces are missing from this overlap region, that may indicate that the initial alignment is inaccurate.

Improving Initial Alignment Hints

Select the cloud constraint you want to fix and then select **Show Constraint** from the **Constraint** menu. The ControlSpaces of the two ScanWorlds are shown and the Initial Alignment Hints pick points are shown as well.

At this time you can add or replace pick points, with the goal of improving initial alignment accuracy. If the initial alignment was satisfactory, but you want to improve it in a single region, add a pick point within that region in each Constraint viewer (hold **SHIFT** to retain the previous picks). To show the original picks again, double click the constraint in the **Constraint List** tab. When finished, select **Update Initial Cloud Alignment**. The constraint is reset to use the updated set of pick points and its status is set to "not aligned".

Max Iterations Warning

When running the **Optimize Cloud Alignment** command, you may occasionally see the warning: "The following cloud constraint alignments were halted by the Cloud Reg Max Iterations limit." This indicates that the **Optimize Cloud Alignment** command did not arrive at a solution before reaching the **Cloud Reg Max Iterations** limit.

This may be addressed by:

1. Repeating the **Optimize Cloud Alignment** command; this continues the optimization where it left off last. In general, you can run the Optimize Cloud Alignment command any number of times to improve the result, (if improvement is possible).
2. Increasing the Cloud Reg Max Iterations preference and repeating the Optimize Cloud Alignment command.
3. Improving your initial alignment hint as described above and repeating the **Optimize Cloud Alignment** command.

Notes on Global Registration with Cloud Constraints

In general, global registration treats cloud constraints just like any other type of constraint. Several items related to global registration deserve more explanation, described below:

Cloud Constraint Registration Error

In the **Constraint List**, the error column lists the error of each constraint after global registration. In general, this is the distance between two constraint objects after the optimal registration transform has been computed for their ScanWorlds. For cloud constraints, the error value indicates the increase in the error metric of the cloud alignment when offset by the global registration from the optimal pair-wise alignment (i.e., the alignment computed by the **Optimize Cloud Alignment** command). If you register two ScanWorlds using only a cloud constraint, the error for the cloud constraint will be 0. The reason is that since there is no other constraint in the registration, the global registration does not need to adjust the cloud alignment. If you then add a HDS Target constraint and Register, the global registration starts spreading the errors among the constraints and you should see a non-zero error for the cloud constraint.

Underconstrained Constraints

You will occasionally see constraints that generate a warning upon completing the **Optimize Cloud Alignment** command. As described above, these constraints are labeled "aligned/underconstrained" in the **Error Vector** column. This occurs when the overlap geometry of the clouds does not fully constrain the alignment in all six degrees of freedom. In such cases, an alignment was found, but that alignment can slide along or rotate around one or more dimensions without affecting the cloud alignment error. For example, although we can align two scans of a flat floor, we can also slide them around in the plane without affecting the alignment error.

In these cases, the cloud constraint does the right thing; it behaves (numerically) as a plane-plane constraint and allows the global registration to slide the ScanWorlds relative to each other

as long as they stay in the same plane. Similarly, you could scan a large cylinder from two locations and the cloud constraint would be underconstrained, allowing the ScanWorlds to slide along and rotate around the cylinder's center axis.

When a cloud constraint is underconstrained, you may need additional constraints to lock down the registration of the two ScanWorlds. In the planar case, you may need two additional HDS Target constraints. In the linear case, two additional asymmetric HDS Targets would be sufficient to lock it down.

Weights on Constraints

Cloud constraints are created such that their error contribution to the global registration is scaled relative to the error contribution of HDS Target constraints with weights of 1.0.

When you create a registration with a combination of cloud constraints and HDS Target constraints, we recommend that you not adjust the weights of these constraints unless you have a specific reason, such as:

- You know that a particular HDS Target (or other constraint object) is not as accurate as a standard HDS Target (roughly 3 mm accuracy) and you have a good estimate of how accurate your method for establishing the coordinate was.
- You used a cloud in a cloud constraint that was acquired using a scanner that has a different per-sample accuracy.

In these cases, the weight should be proportional to the accuracy of your HDS Target or scan data. For example, if the HDS Target was 10 mm accurate, a good weight would be $0.3 = 3 \text{ mm} / 10 \text{ mm}$, where 3 mm is the rough accuracy of a HDS Target. If a scan cloud with 3 cm accuracy was used, the weight should be around $0.2 = 6 \text{ mm} / 30 \text{ mm}$, where 6 mm is the per-sample accuracy of a scan.

Managing ScanWorld Copies

Each ScanWorld is given a unique identifier (not visible to the user) when it is first created. When a ScanWorld is copied, the new ScanWorld shares the same identifier. Copying a ScanWorld behaves differently depending on whether the ScanWorld is pasted into the same, source database or into a different database.

ScanWorld Copy Behavior within a Single Database

When a ScanWorld is pasted into the same database, it is created as a link to the source ScanWorld, not as a new, distinct entity, and any change to one affects the other.

Example:

1. Select a ScanWorld.
2. From the **Edit** menu, select **Copy**.
3. Select a location within the same database, and then select **Paste** from the **Edit** menu. A copy (link) of the ScanWorld appears.
4. Change the name of either ScanWorld.
5. Notice that the name of the other ScanWorld is also updated.

ScanWorld Copy Behavior Between Databases

When a ScanWorld copy is pasted from one database to another, it is created as a new entity. Because the ScanWorlds are in different databases, each ScanWorld can be changed without affecting the other. However, when a ScanWorld is pasted into a database that already contains a ScanWorld with the same identifier, differences in annotations, ModelSpaces, and ControlSpaces must be resolved before the merge (copy) is completed.

Example:

1. Select a ScanWorld.
2. From the **Edit** menu, select **Copy**.
3. Select a location in another database, and then select **Paste** from the **Edit** menu. A copy of the ScanWorld appears.
4. Change the name of either ScanWorld.
 - Notice that the name of the other ScanWorld is *not* updated.
5. Add, edit, or delete an annotation and a ModelSpace in the copied ScanWorld.
6. Select either ScanWorld. For merging purposes, this is the “source” ScanWorld.
7. From the **Edit** menu, select **Copy**.
8. Select any location in the database containing the other ScanWorld. For merging purposes, this is the “target” ScanWorld, which has the same identifier as the source ScanWorld.
9. Select **Paste** from the **Edit** menu. The **Merging ScanWorlds: Resolve Differences** dialog appears and displays discrepancies in the annotations.
10. Select the annotations you wish to have in the target ScanWorld, and then click **OK**. The **Merging ScanWorlds: Select ModelSpaces** dialog appears.
11. Select the default ControlSpace and ModelSpaces you wish to have in the target ScanWorld, and then click **OK**. The source and target ScanWorlds are merged into the target ScanWorld’s database.

Modeling

Overview

Cyclone's tools analyze and extract information from point cloud data and convert point clouds to CAD object-based line and surface models. There are many advantages to using *Cyclone* for this purpose:

- Most current CAD programs do not support massive clouds of points gracefully.
- Modeled objects can provide dimensions and measurements that the clouds cannot provide directly (e.g., the outside diameter of a pipe, or the radius of a pipe bend).
- A form of data compression can be realized, since a pipe has only a few parameters, as compared with hundreds or thousands of points sampled on the surface of the pipe.
- Best-fit objects are often more accurate than the individual point samples, since the fitting tends to average out the individual sampling noise.

Working with Objects

Region Growing

The **Region Grow** submenu commands are used to fit a patch, cylinder, or sphere to an appropriate group of points within a point cloud without first having to segment the points. This is done by picking one or more points within the region that contains a supported shape (patch, cylinder, or sphere), and then choosing the **Region Grow** submenu command appropriate for that shape. For example, if there is a wall behind an assortment of conduit and control boxes in the ModelSpace, you can pick a few points on the wall and direct the program to fit a patch via region growing. The algorithm determines the plane of the wall and ignores the points on the conduit and control boxes, since they do not fit to the plane.

The **Region Grow** submenu also contains the **Smooth Surface** command. Rather than fitting an object to a group of points, this command is used to identify points on a connected smooth surface. For example, the lawn of a golf course is smooth but not flat enough for **Region Grow Patch**.

In the **Region Grow: Smooth Surface** dialog, the **Hardtop Surface** check box enables improved performance when segmenting points on a road from points on passing vehicles. When using this option, you should use a fence to select only road points and associated noise points. All other points (curbs, center dividers, sign posts, etc) should be excluded from the selection. A single pick point on the road surface is required. For **Hardtop Surface**, the **Gap to Span** defines the scale on which the road looks approximately flat. When the segmentation is complete, the noise can be removed from the ModelSpace.

- Default region growing parameters can be preset to initial values in the **Modeling** tab of the **Edit Preferences** dialog.
- Region growing can also be used to fit objects from parts tables. See the *Region Grow* entry in the *Commands* chapter for instructions.
- Sometimes, the scanner's line of sight to a surface is partially blocked by an obstruction (another object). Region growing stops when it encounters this type of occlusion, unless there are multiple pick points on each part of the object separated by the shadow or other occlusion.
- When region growing, multiple clouds may be selected as long as there is at least one pick point.
- When enabled, the Limit Box can improve performance by reducing the amount of points processed; only those visible points are used.

Fitting Objects

A point cloud can be quickly best-fit by using *Cyclone*'s **Fit Fenced to Cloud** and **Fit to Cloud** submenu commands. The fitters typically take only a few seconds to generate a best-fit object. The user indicates the type of surface to be fit for the selected point cloud, and the fitting algorithms perform a multi-dimensional minimization over the parameter space for the surface type. For example, a sphere in 3D has four parameters: X, Y, Z coordinates, and a radius, so the algorithm finds the combination of X, Y, Z coordinates and radius values for a sphere that best fits the point cloud (i.e., the average offset of the points to the sphere's surface is zero).

- Object fitting can be adjusted by applying constraints and catalog/parts tables via the **Object Preferences** dialog.
- The user is notified of potentially bad fits based on the fit tolerances specified in the Object Preferences dialog.
- After an object has been fit, the point cloud(s) that were used to create the object can be displayed using the **Show Object's Cloud** command. This provides access to the points for visual comparison or verification.
- The point cloud used in the fit may be retrieved for additional use through the **Insert Copy of Object's Points** command. Point clouds that have been automatically merged through the fitting process may be split into the component clouds via the **Explode** command.

Common Fitting Objects:

PATCH

flat surfaces: walls, floors; boxes and slabs when extruded

CYLINDER

vessels, pipes, flanges, columns

SPHERE

vessels, targets

CONE

lights, funnels

LINE SEGMENT

conduit, railing, cabling

ELBOW

bend, elbow

CORNER

three intersecting planes

Box

rectangular slabs, crates, shipping containers

STEEL SECTION

angle, channel, tee, rectangular tube, and wide-flange

Using Fit Tolerances

Object types that support fitting can be tested against user-specified tolerances in order to verify that the quality of a fit does not exceed the allowable quantities. Fit quality tolerances can be set in the **Object Preferences** dialog for each object type that can be fit. When fitting using tolerances, a warning message appears if the statistical errors of a fit are beyond one or more of the set tolerances. For example, if the **Std Deviation** value is set to 4 mm for cylinders, each time a cylinder is fit to a cloud of points, the standard deviation of the fit vs. the points in the cloud is compared with the "Std Deviation" fit tolerance for spheres; if it is greater than 4 mm, *Cyclone* warns that the fit may not be acceptable to the user.

- To enable fitting to tolerances, first enable and set the tolerances within the **Object Preferences** dialog, and then toggle the **Check Fit Tolerances** command ON from the **Fit to Cloud Options** submenu.

Using Fit Constraints

Some object types that can be fit to a point cloud also support the constraining of the fitting parameters to specific values. Fitting constraints can be set for each object type in the **Object Preferences** dialog. Each constraint may be **Unspecified**, **Initial**, or **Fixed**. The value of an **Unspecified** constraint is found by the fitting algorithm. An **Initial** value can help the fitter as a hint of what the value might be. The fitter uses a **Fixed** constraint as the final value of that parameter. For example: if you know that a cylinder to be fit must have a certain diameter, you could set the **Fit Diameter** to **Fixed** at that diameter. On the other hand, if you believe that the cylinder might be around a certain diameter, but aren't sure, you could set the **Initial** value to that diameter, but the final cylinder may have a different diameter. Finally, if you don't know what the diameter is, you can set the **Fit Diameter** to **Unspecified**.

- To fit with constraints, first enable **Fit Constraints** with **Initial** or **Fixed** values within the **Object Preferences** dialog, and then toggle the **Use Fit Constraints** command ON from the **Fit to Cloud Options** submenu. Otherwise, the fitter determines the final values entirely on its own.
- The **Fit Constraints** area of the **Object Preferences** dialog is disabled for objects that do not support fit constraints.

Point Cloud Segmentation

It is often necessary to divide (or segment) a point cloud into one or more subsets in order to focus further operations on a subset, remove unwanted data, etc. Segmentation provides a process for grouping different regions of point clouds based on shared characteristics. For example, points may be segmented because they are on the same pipe or on the same wall. The points may also be segmented because they are unnecessary for subsequent modeling or visualization (they may be stray or unnecessary points).

Simple Segmentation

Simple segmentation involves using the fence tool to manually separate the points of interest. The fence can be manipulated for the precise outline required to segment one set of points from another or to clear unwanted points. Once the fence contains the desired points, the **Cut by Fence** command creates at most two point clouds – one inside, and one outside the fence. Simple segmentation is limited in that it uses a single fence from a single viewpoint to create segments, and if further refinements are needed, additional segments are created. When more precise segmentation is needed, use the sub-selection function (see [Advanced Segmentation](#) in the *Modeling* chapter and [Point Cloud Sub-Selection](#) in the Commands chapter) before creating segments.

Advanced Segmentation

Advanced segmentation involves using multiple sub-selections to refine a selection (add or remove points) before segmentation, fitting, or other processes are performed. After the initial selection, the viewpoint can be changed and a new fence can be drawn in order to make a more precise selection. When the sub-selection contains only the desired points, the selected points are segmented via the **Cut Sub-Selection** command.

To make a precise segment from a point cloud:

1. Draw a rough fence around the points that you want to segment.
2. Using the **Selection | Point Cloud Sub-Selection** submenu commands, create a temporary selection.
3. Continue to refine the temporary selection (add or subtract points) by manipulating the viewpoint, drawing a new fence, and using subsequent **Selection | Point Cloud Sub-Selection** submenu commands.
4. From the **Create Object** menu, point to **Segment Cloud**, and then select **Cut Sub-Selection**. The points within the temporary subset are now a separate point cloud and can be selected independently.

Segmenting by Intensity

The **Cut Cloud by Intensity** dialog can be used to manually segment a point cloud above and below a user-specified intensity. When a point cloud is viewed with a color map, the **Cut by Intensity** command provides a slider that interactively updates the point cloud and allows you to specify an intensity range for segmentation. This can be useful to separate points sampled on surfaces with different (light, dark, etc.) materials.

Segmenting by Interfering Points

The **Highlight Interfering Points** dialog is used to highlight points that are within a user-specified distance of a selected object and segment those points from the point cloud. For example, if you place a beam into a ModelSpace View using the **Insert Object** command, and you want to identify any points in the point cloud that may interfere with the beam, this command is used to identify and segment the interfering points. For directions on identifying and segmenting interfering points, see the [*Interfering Points*](#) entry in the *Commands* chapter.

Trim Edges

Occasionally, when scanning, the laser strikes partially on and partially off the edge of an object's surface, which can negatively impact the accuracy of the captured point. The **Trim Edges** command creates a new subset that consists of all the points near the cloud's boundary. The points left in the interior have fewer "partial" samples.

Inserting Objects

Not all objects can be produced by fitting to a point cloud. Sometimes the number of points sampled on the surface is insufficient for a good statistical fit, the distribution of points is unbalanced, or perhaps an object was never directly sampled. *Cyclone* provides manual insertion of objects as an alternative to automatic fitting via the **Insert** submenu command in the **Create Object** menu. Manually inserted objects behave identically to all other objects, except they are not based on sampled points.

- Object insertion parameters can be adjusted via the **Creation/Initial Values** area of the **Object Preferences** dialog. (If there is a pick point or object selected in the ModelSpace, some of these settings may be overridden. The picked point or object serves as a reference

for the inserted object's parameters.)

- Object insertion parameters can also be set via standard parts tables. To insert objects from a parts table: First, enable **Catalog/Parts Table** and make **Table** and **Item** selections within the **Object Preferences** dialog (different tables and parts can be selected at the time of insertion). Then toggle the **Use Parts Table** command ON from the **Create Object** menu.

Object Insertion Rules

- **The first object inserted** into an empty scene is inserted at (0, 0, 0).
- **If there is no selected object and no pick point** when an object is inserted, the size and alignment of the new object are defined in the **Creation/Initial Values** area of the **Object Preferences** dialog. The object is located in front of the current view at the focal point.
- When **an existing object is picked**, the picked object provides a plane, direction, and/or vertex for the inserted object to base its initial position, orientation, and dimensions. For example, a patch readily defines a plane, a direction (plane normal), and a point (origin). The rules can be sophisticated (e.g., when inserting an object on a cylinder, picking near the end point inserts the object aligned with the cylinder's axis, but picking away from the end point inserts the object perpendicular to the cylinder's surface at the picked point). Exception: cameras are always inserted at the current viewpoint regardless of picks.
- When **inserting a series of objects**, the previous object provides a plane, direction, and/or vertex for the inserted object to base its initial position, orientation, and dimensions. Using this method, it is possible to build a chain of objects, such as a piping run, from scratch. However, when using auto-pick (see the *Auto-Pick Points* entry in the *Commands* chapter), the specified auto-pick point overrides any selection point.
- **The inserted object is initially selected with a pick point** based on the previous selection. If there was no previous selection, a default pick point is provided.

Using Parts Tables

Manufacturers of standard equipment fabricate the parts based on published specifications. These specifications can be formatted as tables, including geometric data (e.g., dimensions), derived properties (e.g., surface area, mass), and nominal information for each standard "part". *Cyclone* supports standard Parts Tables for pipes (cylinder), standard piping components, and steel sections (angle, channel, rectangular tube, tee, wide flange). The tables are ASCII text files organized under the **Tables** subdirectory under the "Library folder" of the *Cyclone User Configuration Manager*. *Cyclone* ships with standard tables for pipes, tubes, and steel sections. Do not edit these standard tables. Tables for other shapes are also supported, although standard tables are not provided. See the templates in the **Lib\Tables** subdirectory.

Disclaimer:

Although Leica Geosystems HDS provides standard tables, the contents of the tables are provided to the user as-is. The correctness of the tables should be checked to the user's satisfaction.

Fitting and Inserting Objects from Parts Tables

Many object types support the use of tables for insertion; some support the use of tables in fitting to clouds. For those object types that support the use of tables, the **Catalog/Parts Table** area of the **Object Preferences** dialog is activated when that object is selected in the **Object Type** field.

- To enable object fitting from a parts table: First, open the **Object Preferences** dialog, and then enable **Catalog/Parts Table** and make **Table** and **Item** selections. Next, toggle the **Show Table Matches** command ON from the **Fit to Cloud Options** submenu. See the *Fit to Cloud* entry in the *Commands* chapter for specific procedures.
- To enable object insertion from a parts table: First, open the **Object Preferences** dialog, and then enable **Catalog/Parts Table** and make **Table** and **Item** selections (these selections can be changed at the time of insertion). Next, toggle the **Use Parts Table** command ON from the **Create Object** menu. See the *Insert* entry in the *Commands* chapter for specific procedures.

Selection

Selection Basics

Generally speaking, an object is selected when you have indicated (in some way) that it will participate in the next command operation. Selected surface objects are indicated with a checkerboard texture overlay, while point clouds and lines temporarily become thicker while selected.

- The **Show Pick Polyline** command toggles the display of a polyline along the current series of pick points. This is useful for keeping track of the order of picks for operations like creating a cubic spline or a polyline from pick points. It does not create a polyline - it only displays the order of the pick points.
- The **Show Pick Points** command toggles the display of pick points as white points.
- The **Invert Selection** command can be particularly helpful for selecting all but a few objects in a ModelSpace. This is accomplished by first selecting only the objects that you eventually do *not* want selected, and then using the **Invert Selection** command.
- The ability to select any object of a specified type can be toggled ON/OFF from the **Selectable/Visible** tab of the **View Properties** dialog. (See the *Set Selectable* entry in the *Commands* chapter.)
- The **Auto-Pick** function (see the *Auto-Pick Points* entry in the *Commands* chapter), allows you to specify a significant point (i.e., the end of a cylinder) that is automatically picked when that object type is selected. When using auto-pick, the specified point is always the only picked point regardless of whether the object was selected by fence or picked at any other point on the object.

Object Selection Methods

- A. In **Pick** mode, click the object with the **Pick** mode cursor . This is the only way to select objects that also allows you to indicate (or “pick”) a specific point on the selected object. Picking is useful for indicating where an action might take place next and is required for some operations. For example, if you pick an object, then insert an object via **Insert** submenu commands, the new object’s position and orientation may be based upon where you picked the first object.
- Multiple objects can be selected in **Pick** mode by holding **SHIFT** and clicking the objects you want to select or by entering **Multi-Pick** mode and clicking objects with the **Multi-Pick** mode cursor .
- Holding **SHIFT** and clicking an object that is already selected produces additional pick points.
- Holding **CTRL** and clicking an object toggles the selection state of the object without affecting the selection state of other objects.
- B. Use the **Select Fenced/Deselect Fenced** commands to select/deselect every object within or overlapping the drawn rectangular data fence. See the *Using a Fence* section in this chapter for more information. Previous selections are not affected by the fence selection.
- C. All objects of a specified type can be selected/deselected from the **Selectable/Visible** tab of the **View Properties** dialog. (See the *Set Selectable* entry in the *Commands* chapter.)
- D. All objects assigned to a layer can be selected/deselected simultaneously from the **Layers** dialog.

Point Cloud Selection

Selecting Whole Point Clouds

- A. In **Pick** mode, click the point cloud with the **Pick** mode cursor. This is the only way to select point clouds that also allows you to indicate (or “pick”) a specific point. Picking is useful for indicating where an action might take place next and is required for some operations.
- Multiple point clouds can be selected in **Pick** mode by holding **SHIFT** and clicking the point clouds you want to select or by entering **Multi-Pick** mode and clicking objects with the **Multi-Pick** mode cursor .
- Holding **SHIFT** and clicking an object that is already selected produces additional pick points.
- Holding **CTRL** and clicking an object toggles the selection state of the object without affecting the selection state of other objects.
- B. Draw a fence and then use the **Select Fenced/Deselect Fenced** commands to select/deselect every point cloud within or overlapping the drawn rectangular data fence.
- C. All point clouds can be selected/deselected from the **Selectable/Visible** tab of the **View Properties** dialog. (See the *Set Selectable* entry in the *Commands* chapter.)

Point Cloud Sub-Selection

Point Cloud Sub-Selection involves using multiple sub-selections to refine an initial selection (add or remove points) before segmentation, fitting, or other processes are performed. After the initial selection, the viewpoint can be changed and a new fence can be drawn to add or subtract points with subsequent point cloud sub-selection commands (See the [Point Cloud Sub-Selection](#) command entry). When the sub-selection contains only the desired points, the selected points can be manipulated independently of other points in the ModelSpace.

To make a point cloud sub-selection:

1. Draw a rough fence containing all the points that you may want to use.
2. Using the **Selection | Point Cloud Sub-Selection** submenu commands, create a temporary selection.
3. Continue to refine the temporary selection (add or subtract points) by manipulating the viewpoint, drawing a new fence, and using subsequent **Selection | Point Cloud Sub-Selection** submenu commands. The points within the temporary subset can now be segmented, fit, or used for other processes.

Using Object Handles

Most objects in *Cyclone* provide handles that can be used to adjust the object's position, orientation, and/or dimensions.

Displaying Handles

Handles are color-coded as follows to indicate the type of handle behavior.

Red rotate

Orange resize, translate, snap

Blue translate, snap

When an object is selected, the settings in the **Handles** submenu determine which handles are displayed.

- The **Show Handles** command toggles the display of handles. This command overrides the **Show Rotation Handles** command.
- The **Show Rotation Handles** command toggles the display of rotation handles.
- The **Handles Always Visible** command toggles the constant visibility of handles even when hidden from view by an object. When this command is checked, it allows convenient access to handles that are behind or inside of objects, which would otherwise be hidden from view.

Using Handles to Translate, Resize, or Rotate Objects

All object types in *Cyclone* can be translated, resized, and rotated using their handles.

- To translate an object via its handles, drag one of the object's blue handles, or press and hold **CTRL** and then drag one of the object's orange handles.
- To resize an object using its handles, drag one of the object's orange handles.
- To rotate an object using its handles, drag one of the object's red handles.
- When an object in a group is selected, both that object's handles and the handles on the group's bounding box are displayed and available for manipulating the group as a whole. When the group is selected in any other way (e.g., by fence), only the handles on the group's bounding box are shown. Orange resizing handles are not available for objects in a group.

Resizing Steel Shapes

- To resize a steel shape's profile while maintaining the position of the face opposite the handle, drag one of the object's handles.
- To resize a steel shape's profile around its midline, press and hold **ALT** while dragging one of the object's edge handles.

Constraining Handle Movement

Object handle motion can be constrained to a specified axis or plane in two ways. The first method is to set the constraint *before* manipulation via the **Edit Object | Handles | Constrain Motion to** submenu commands. The second method cycles constraints *during* handle manipulation via keyboard modifiers. Additionally, object rotation via red handles can be restricted to user-specified angular intervals.

Constraining Before Manipulation

- To select the plane or axis along which the handle movement is restricted, use the **Edit Object | Handles | Constrain Motion to** submenu commands. The **Screen Plane** setting constrains motion to be perpendicular to the viewpoint, while the other settings use coordinate system axes.
- Handles can also be constrained to move by user-specified increments by using the **Snap to Grid** command.
- To constrain rotation handle movement, the **Snap to Grid** command must be ON. To set the rotation interval, use the **Set Rotational Spacing** command.

Constraining During Manipulation

- To constrain handle movement to an axis, pick a handle and press the **X**, **Y**, or **Z** key. The first time the key is pressed, handle movement is constrained to be along the corresponding axis (i.e., pick a handle and press **X** and handle movement is constrained along the **X** axis).
- To constrain handle movement to a plane, pick a handle and press the **X**, **Y**, or **Z** key a second time. The handle is constrained to the plane that does *not* include the axis of the key pressed. Pressing the **X** key a third time removes the constraint and reverts to the default behavior.
- To toggle the **Screen Plane** constraint ON/OFF, press **S**.
- To remove any active handle restraints, pick a handle and press **Spacebar**.

Note: Select either **User Coordinate System** or the **Object Coordinate System** axes for handle constraints in the **Edit Object | Handles | Constrain Motion to** submenu.

Using Handles to Snap Objects

Using Handles to Snap Objects

Most objects in *Cyclone* provide one or more points, lines, and/or planes that can be used in addition to its handles as snapping points when moving or resizing other objects via handles. (Note that some objects provide bounding box handles only.) Objects that are being manipulated snap to other objects whose handles or other snapping points, lines, and/or planes come within the user-specified snapping threshold. The snapping threshold can be adjusted using the **Snap to Object Threshold** dialog.

To snap selected objects:

1. Multi-select the objects you want to snap.
2. Press and hold **SHIFT** and drag the second object's handle towards the reference object to resize the object. Press and hold **SHIFT + CTRL** while dragging the second object's handle towards the reference object to translate the object. When the dragged handle passes over one of the reference object's handles, it snaps to that handle. When any of the reference object's snapping points are within the snapping threshold (specified in the **Edit Snap to Object Threshold** dialog), the object being manipulated jumps to the snapping point defined by the reference object.
- When the snapping point is “handle-to-handle”, the handles appear larger when they snap. The dragged handle can be moved away from the snap when the cursor passes beyond the snapping threshold.
- To make the object you are moving or resizing snap to the reference object's snapping point at a perpendicular or parallel angle, press and hold **SHIFT + ALT** and drag the object handle towards the reference object. If the angle of intersection is at least 45 degrees, the second object is snapped at a perpendicular angle. If the angle is less than 45 degrees, the second object snaps parallel with the reference object.

Adding and Deleting Boundary Vertices to or from a Patch, Extrusion Face, Polygon, or Polyline

Vertices can be added to or deleted from a selected patch, extruded face (top and bottom faces), polygon, or polyline while in **Pick** mode.

- To add a vertex to the selected patch, extrusion face, polygon, or polyline, press and hold **ALT** and click along the edge of the selected object without moving the mouse. A vertex (and a handle) is added to the object.
- To remove a vertex from the object, press and hold **ALT** and click the vertex you want to remove without moving the mouse. The vertex (and the handle) is removed.

Editing Objects

The **Color and Material Editor** dialog is used to change the display properties of the selected object(s). Color and material settings can be adjusted individually. For a vertex, line, or point cloud, only the diffuse material property affects how it is displayed.

The **Align** submenu commands and dialogs are used to align objects in a variety of ways, including: aligning objects to a reference surface or axis; using a reference object to make objects coincident, colinear, coplanar, parallel, perpendicular, or codirectional; and placing selected objects at a specified distance or angle from each other. Many common modeling tasks can be accomplished with **Align** submenu commands and dialogs (e.g., making walls perpendicular to floors or making a ceiling parallel to a floor.)

The **Extend** submenu commands extend a single object to the intersection with a reference object or extend the boundary of all selected objects to the planar or linear intersection with the other selected objects. A common use would be extending a column to a floor.

Extrude

The **Extrude** submenu commands are used to create an extrusion object from a patch or a face of an extrusion object. The extrusion is created with a specified thickness either along a user-specified axis, along the patch's normal towards the viewpoint, or towards a pick point. An extrusion can be used to create a box from a patch to represent an electrical box, for example.

Merge

The **Merge** command combines two or more selected objects of the same type. In the process of creating a single new object, the merging function fills in space between the selected objects. Use this command to create objects such as wide floors and ceilings and long pipes. This feature is particularly useful when filling areas that have not been scanned directly. If the objects were created from point clouds, the merged object is refit to the original point clouds. The current settings and Object Preferences for use of Parts Table, Fit Tolerances, and Fit Constraints are used in this fit.

Copy

The **Copy** dialog is used to create a series of translated copies of the selected object(s). The **Copy at Offset** tab is used to specify the number of copies and the axis along which they are translated. This option is commonly used to model ceiling beams, pipe racks, or stairs.

The **Copy at Angle** tab is used to make a series of copies of the selected object(s) rotated around a specified axis. The number of copies, the angle between each copy, the center point, and the axis of rotation can be specified in this tab. This option can be used to model the bolts on a flange face, for example.

Move/Rotate

The **Move/Rotate** dialog is used to rotate or move one or more selected objects. When moving an object, the distance and axis of the motion are user-specified. When rotating an object, the center point, angle of rotation, and axis around which the object is rotated are user-specified.

Using a Fence

A fence is a two-dimensional, rectangle or polygon that is used as a boundary to select and deselect objects and point clouds, segment point clouds and other objects, and create 2D line drawings. Fences can be created in **Rectangular** or **Polygonal Fence Mode**, or linear objects (such as polygons, ellipses, arcs, etc...) can be used to define the boundaries of a fence.

Creating a Fence

To create a fence from a linear object:

- This function provides a convenient method for repeatedly creating the same fence.
1. Select the linear object you wish to use to define a data fence.
 - Linear objects can be created by a variety of methods including via the 2D drawing tools. If an arc or other open curvilinear object is selected, the end points are connected for purposes of defining the data fence. 2. From the **Edit** menu, point to **Fence**, and then select **Set from Selection**. The fence is created.

To draw a rectangular fence:

1. From the **Edit** menu, point to **Modes**, and then select **Rectangle Fence Mode**. The **Rectangle Fence** cursor appears.
2. Drag the cursor around the area that you want to fence.
 - The newly drawn fence can adjusted by dragging its sides or vertices with the **Rectangle Fence** cursor.
 - To clear the newly drawn fence, select **Clear** from the **Fence** submenu.

To draw a polygonal fence:

1. From the **Edit** menu, point to **Modes**, and then select **Polygonal Fence Mode**. The **Polygonal Fence** cursor appears.
2. Click to place the first vertex.
 - Drag to adjust the vertex position.
3. Continue adding vertices as needed.
 - A vertex can also be added by clicking anywhere along the line segment between existing vertices.
 - An existing vertex can be deleted by clicking the vertex without moving the mouse.
 - The newly drawn fence can adjusted by dragging its vertices with the **Polygonal Fence** cursor.
- To clear the newly drawn fence, select **Clear** from the **Fence** submenu.

Fence Operations

Deleting with a Fence

When a fence has been drawn, cloud points, mesh triangles, and line segments (in contour lines and polylines) can be deleted relative to the fence and according to their selection status using the **Fence** submenu commands in the **Edit** menu. Point clouds, meshes, contour lines, or polylines that are overlapping a fence are first segmented, and then the appropriate segment is deleted. Other objects are not affected by these fence operations.

- The **Delete Inside** command removes all cloud points, mesh triangles, or line segments within or partially within the data fence.
- The **Delete Outside** command removes all cloud points, mesh triangles, or line segments entirely outside of the data fence. If a triangle or line segment is partially within the fence, it is not deleted.
- The **Delete Selected Inside** command removes every selected cloud point, mesh triangle, or line segment within or partially within the data fence.
- The **Delete Selected Outside** command removes every selected cloud point, mesh triangle, or line segment entirely outside of the data fence. If a triangle or line segment is partially within the fence, it is not deleted.
- When enabled, only those portions within the Limit Box are affected by these operations.

Selecting Objects with a Fence

A fence can be used as a boundary for selecting objects (including mesh triangles and line segments) and point clouds using commands in the **Selection** menu. These commands do not segment selected objects.

- The **Select Fenced** command selects every object or point cloud (the entire point cloud is selected) entirely within or partially overlapping a fence. In the case of a mesh object, the entire mesh object is selected, as are all triangles within or partially within the fence. Subsequent commands performed on the selected mesh/triangles may pertain to the entire mesh if the commands are not mesh-specific (e.g., **Edit | Delete**), and to the triangles if the commands are mesh-specific (e.g., **Mesh | Delete Selection**). In the case of contour lines and polylines, line segments within or partially within the fence are selected.
- The **Deselect Fenced** command deselects every object or point cloud (the entire point cloud is deselected) entirely within or partially overlapping the fence.

Segmenting with a Fence

A fence can be used as a boundary to segment the selected cloud, mesh object, contour line, or polyline into two subsets using the **Cut by Fence** command. More than one point cloud, mesh object, contour line, or polyline can be segmented simultaneously. For more information, see the [Point Cloud Segmentation](#) section in the *Modeling* chapter.

Copy Fenced Objects to a New ModelSpace

The **Copy Fenced to New ModelSpace** command is very useful for editing objects in a separate ModelSpace containing only the objects with which you are currently working. This command creates a new ModelSpace, copies the fenced object(s) into the new ModelSpace, and opens a ModelSpace View for the new ModelSpace. Partially fenced objects are included in the new ModelSpace. Partially fenced clouds are not segmented, but only the points within the fence are copied. See the [Copy Fenced to New ModelSpace](#) entry in the *Commands* chapter for instructions.

Extracting 2D Lines from the Contents of a Fence

A 2D line drawing can be created from the contents of a **rectangle** fence using the **From Fence Contents** command. The 2D drawing is created and displayed in a new window. See the *From Fence Contents* entry in the *Commands* chapter for more information.

Taking Measurements

Cyclone provides a variety of object and linear measurement tools in the **Measure** submenu. Measurements can be viewed, saved, deleted, and copied to the Windows clipboard using the **Measurements** dialog. It is often unnecessary to save each measurement, so measurements are considered “temporary” unless the **Save Measurements** option is ON. Temporary measurements

are replaced by each newly executed measurement command, and can be cleared using the **Clear Temporary Measurements** command. Some objects must be picked to display a measurement graphic.

Object Measurements

ELEVATION

This command calculates the distance between a picked point and the origin along the current vertical direction. If multiple points are being measured, the results are displayed in the order in which the points are picked.

CENTERLINE LENGTH

This command calculates the length along the centerline of the selected object(s). When more than one object is selected, this command sums the centerline length of each object. You can use this command to measure the distance along a pipe run .

PIPING TAKEOFF

This command calculates the takeoff for the selected piping run. The piping takeoff is similar to the centerline length, except that the length of an elbow or bend is not the arc length, but rather the sum of the distances from each of its endpoints to where its endpoint tangents cross.

SURFACE AREA

This command computes the surface area of the selected object(s). If more than one object is selected, individual and collective measurement results are displayed.

VOLUME

This command computes the volume of the selected object(s). If more than one object is selected, individual and collective measurement results are displayed. Mesh objects and point clouds viewed as a mesh do not contribute to this volume.

Surface Deviation

Measure surface-to-surface differences in elevation between a TIN and the active reference plane or another TIN.

TIN Volume

Measure cut/fill volume between a TIN and the active reference plane or another TIN.

MESH VOLUME: ABOVE/BELOW REF PLANE

This command computes the volume of a mesh to the active Reference Plane. For example, if you have a scan of a mound of dirt and need to know its volume, you can set the active Reference Plane to the ground level, create a mesh object from the point cloud (or view the point cloud as a mesh), and compute the mesh volume.

INTERFERING POINTS

This command is used to highlight and segment points that are within a user-specified distance of a selected object. If the selected object defines a volume, the points contained inside the object are also highlighted and segmented.

MESH TO POINTS DEVIATIONS

This command compares the faces in a selected mesh object with the points in the selected point cloud, and uses the data to calculate deviations in accordance with user-defined parameters. This command is applicable to mesh objects only (as opposed to a point cloud viewed as a mesh).

CONTOURS TO MESH DEVIATIONS

This command compares the lines in a selected contour set with the faces in the selected mesh object, and uses the data to calculate deviations in accordance with user-defined parameters. This command is applicable to contours and mesh objects only (as opposed to a point cloud viewed as a mesh).

FIND HIGH/LOW POINTS

This command is used to locate the highest or lowest point in a point cloud within a specified area around a picked point. Vertices can be created when high/low points are found.

Linear Distance Measurements

POINT TO POINT

This command calculates the shortest distance between picked points (a straight line) as well as the component offsets along the major axes.

POINT TO CENTER LINE/POINT

This command calculates the distance between the first picked point and the center point or the nearest intersection with the centerline of the object selected last.

- If the second object is a sphere, it does not matter where it was picked or if it was selected by other means, since the center point of a sphere is its exact center.
- If the measurement is to the *centerline* of the second object, it does not matter where the object was picked or if it was selected by other means.
- For measurements, centerlines are extended beyond the endpoint or boundary of the actual object to the nearest point of intersection. For example, a measurement to the centerline of a cylinder is made to the nearest point on the underlying axis of the cylinder, which is not necessarily a point within the length of the cylinder.

POINT TO UNBOUNDED SURFACE

This command calculates the distance between the first picked point and the nearest intersection with the surface of the object selected last. Since the measurement is to the *unbounded* surface of the second object, it does not matter where it was picked or if it was selected by other means.

- For measurements, surfaces are extended beyond the endpoint or boundary of the actual object to the nearest point of intersection. For example, a measurement to the surface of a

patch is made to the nearest point on the underlying plane of the patch, which is not necessarily a point within the boundary of the patch.

CENTER TO CENTER

This command calculates the distance between the centers of the first selected object and the nearest intersection with the center of the second selected object. The object type determines whether the measurement is to a centerline or to a center point (e.g., a sphere is measured to its center point and a cylinder is measured to its centerline.). It does not matter where the objects were picked or if they were selected by other means.

POINT TO SCANNER

This command calculates the distance between a picked point and the selected scanner. If a point cloud is picked and a scanner object is not selected, *Cyclone* determines the distance from the picked point to the scanner that was used to capture the point cloud. This can be used to get a sense of the “depth” of a scanned scene, and whether or not the point is within the calibrated range of the scanner.

Datum Measurements

A “datum” is a user-defined vertical line used to take measurements of other objects and features. You create a datum by picking a point in the ModelSpace and selecting the **Set Datum to Pick Point** command, or by selecting the **Enter Datum** command, then entering the (X, Y, Z) coordinates of the desired point; the datum is created as a vertical line passing through that point. Until a datum is defined by the user, the default datum passes through the point (0, 0, 0). You can measure the distance from the datum to any picked point(s) in the ModelSpace using the **Point Distance** command; the shortest distance from the point to the datum line is computed.

Working with Meshes

A mesh is a series of triangles created using the points in a point cloud, vertices, polylines, or any combinations of the three as vertices. For each adjacent trio of points in a cloud, a triangle is created. This has the effect of generating a coherent visual surface from a point cloud.

Cyclone enables you to create dynamic mesh objects that can be manipulated extensively to achieve the visual results you need. Triangles can be added, edited, or removed. Mesh objects can be decimated using a dynamic real-time interface that allows you to adjust and fine-tune your mesh. New vertices can be incorporated into the mesh from user-drawn polylines, and breaklines can be placed into the mesh to preserve geometric features during a decimation.

Sample Grid

Resamples TIN meshes on a regular grid.

Creating a Mesh Object

Mesh objects are created using the **Create Mesh...** command.

Note: If you merely wish to view the point cloud as a mesh, it is recommended that you use the **View Object As...** functionality to view the point cloud as a mesh. See the [View Object As](#) entry in the *Commands* chapter for more information.

- If you are creating a TIN, set the “up” direction as desired before creating a mesh object.

To create a mesh object:

1. Select the point cloud (vertices, or polylines) to be used for creating the mesh object.
2. From the **Create Object** menu, select **Create Mesh....** The **Meshing Style** dialog opens.
3. Select the type of mesh object you wish to create.

Note: When creating a TIN, any selected polylines are incorporated as breaklines in the new TIN.

- Select **Basic Meshing** to create a basic mesh consisting of triangles using trios of adjacent points as triangle vertices. Select **Complex Meshing** to create a mesh consisting of triangles using trios of adjacent points that are likely to lie on the same surface. Select **TIN Meshing** to create a mesh such that there are no overlapping triangles with respect to the vertical direction; the current coordinate system determines the “up” direction for a TIN.

Note: **Basic Meshing** and **Complex Meshing** cannot be done with more than one point cloud.

4. Click **OK**. The mesh object is created.
- Click **Cancel** to abort the command.
- Use the ScanWorld Explorer to make the point cloud visible again. See the *ScanWorld Explorer* entry in the *Commands* chapter for more information.

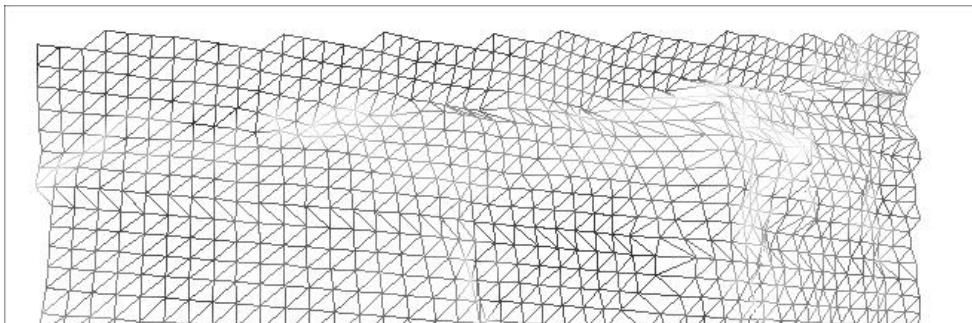
Tip: If your mesh object seems to have deep shadows or what seem to be large gaps, go to the **ModelSpace** tab in the **Edit Preferences** dialog, change the lighting preferences to **Point Source**, and click **Apply**.

Mesh Viewing Options

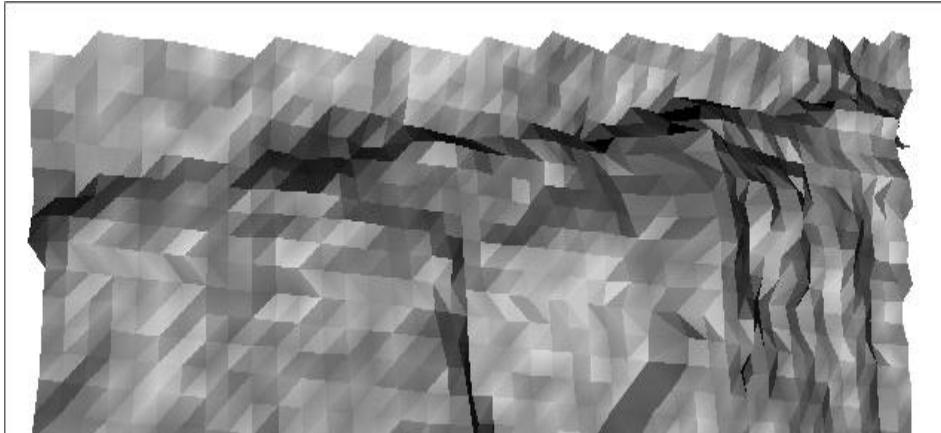
Cyclone provides you with a variety of options for adjusting and fine tuning the view of your mesh object. Viewing a mesh in wireframe can be helpful, especially when you are editing individual triangles and need to see triangle vertices and edges ([See Illustration 1](#)).

Similarly, the "Per Face Normals" view option displays individual triangles such that they may be easier to distinguish. Each triangle is rendered as an individual surface with its own surface normal. As with wireframe, this is a useful view option when editing a mesh ([See Illustration 2](#)).

Mesh Object Illustration 1



Mesh Object Illustration 2



Editing a Mesh Object

Once created, a mesh object can be edited and fine-tuned to produce a specific deliverable.

Note: Only mesh objects can be edited. Point clouds viewed as meshes (via the **View Objects As** dialog) cannot be changed or edited using mesh tools.

Editing Triangles

Each triangle in the mesh object can be edited as an individual object using commands in the **Mesh** submenu. Use the **Move Vertex** command to change the shape of a triangle. You can also use the **Flip Selected Edge** command to make subtle changes in the mesh; this command flips the selected edge, redrawing the two triangles sharing the selected edge.

Use the **Delete Selection** command in the **Mesh** submenu to remove unwanted triangles. Be careful not to use the **DELETE** key on your keyboard. It will result in deleting the entire mesh object. Similarly, avoid using the “Cut” and “Copy” functions in the **Edit** menu unless you intend to use them on the mesh object as a whole.

Filling in Holes

Holes in the meshing can be filled by using the **Fill Selected Hole** command. However, if a hole is not enclosed by the mesh (e.g., it coincides with the outer edge of the mesh), this command may not work as expected. A good technique for filling these types of open-ended gaps is to close off the gap at the edge of the mesh, either by moving a vertex across the gap to close it in, or by adding a face based on picks to either side of the open end of the gap. Once the open end has been closed off, the remaining hole can be filled using the **Fill Selected Hole** command.

Polylines and Meshes

Polylines can be used to enhance a mesh object. A common use for a polyline is to create a breakline in the mesh. A breakline is a polyline that has been incorporated into a mesh object.

The vertices of a breakline are “locked” during decimation; this can be a useful feature when you want to preserve a particular geometric feature of the mesh (e.g., a curb edge). In addition, breaklines are exported with a mesh object when exporting a TIN using the **Export...** command.

Note: A polyline that is projected onto a mesh as a *polyline*, is merely a relocation of that individual polyline. It is not incorporated into the mesh as a breakline, and it is not used during decimation.

To incorporate a breakline into a mesh object (e.g., to preserve a curb edge):

1. Identify the geometric feature to be preserved, and multi-select several points within the mesh along the edge of the feature.
■ It is not necessary to pick vertices within the mesh.
2. From the **Create Object** menu, point to **From Pick Points** and select **Polyline**. A polyline is drawn using the pick points as vertices, and the newly created polyline remains selected.
■ You can also use polylines created from imported files (i.e., from survey data).
3. Multi-select the mesh object.
4. From the **Tools** menu, point to **Mesh**, then to **Breaklines**, and select either **Project Polyline to Mesh** or **Extend Mesh to Polyline** as appropriate. The polyline is incorporated into the mesh, the polyline vertices are added into the mesh, and triangles are redrawn as appropriate to accommodate the breakline.
■ See the *Project Polyline to Mesh* and *Extend Mesh to Polyline* entries in the *Commands* chapter for more information on incorporating polylines into a mesh object.
■ The **Clear Breaklines** command can be used to remove all breaklines from a mesh object.
■ When decimating a mesh containing breaklines, be sure to select either **Maintain** or **Keep Exact** in the **Breaklines** field of the **Decimate Mesh** dialog if you want to preserve breakline integrity.

Decimating Meshes

Mesh decimation is the process by which the number of triangles in a mesh is reduced according to user-defined parameters. The **Decimate Mesh** dialog supports interactive decimation; as you move the **Percentage** slider, the mesh in the ModelSpace view is continually redrawn to reflect the changing percentage value when the **Auto Preview** checkbox is selected.

You control the decimation by determining the level of decimation that is to occur, and whether or not to maintain vertices, breaklines, and boundary edges. For example, when **Maintain Vertices** is enabled, the positioning of mesh vertices is preserved as closely as possible to their original locations, given the amount of decimation being performed; the higher the percentage, the more likely it is that vertices will remain at or near their original positions. You can choose to “maintain” breaklines and/or boundary edges: The boundary edges/breaklines are preserved geometrically relative to the level of decimation. However, you can also choose to ignore breaklines and/or boundary edges, in which case no attempt is made to preserve original positioning, or you can choose **Keep Exact**, in which case the vertices in breaklines and/or boundary edges are frozen in place and are not modified, no matter how heavily decimated the mesh is.

- For detailed instruction on decimating a mesh object, see the *Mesh* entry in the *Commands* chapter.

Selecting Parts of a Mesh

When editing a mesh object, it is necessary to select triangles, edges, holes, or vertices, depending on the function to be initiated. For example, an edge must be selected to use the **Flip Selected Edge** command. When a selection is made within a mesh object, the triangle (to which the selection pertains) is shown in a different color. Selection coloring can be re-defined as desired in the Object Properties dialog (see the *Edit Properties* entry in the *Commands* chapter).

- To select a triangle, click on the desired triangle. Picking triangles while holding the **SHIFT** key allows you to select multiple triangles. If the selected triangle is adjacent to a hole in the mesh, the hole is selected as well.
- For operations that require selection of a vertex or a triangle edge in a mesh object, the vertex/edge closest to your pick on the mesh will be selected and used.

Using a Mesh to Create Other Objects

Once a mesh is created, it can be used for creating other objects. Contours are drawn using a mesh as a basis; the mesh can be either a mesh object or a point cloud viewed as a mesh.

- For detailed instructions on creating contours, see the *Create* entry in the *Commands* chapter.

In addition to contour lines, you can use the **Extract Cloud from Selection** command to create a new point cloud containing the vertices in a mesh object. The resulting point cloud is a *new* cloud; it is not a retrieval of the original cloud(s) used in the creation of the mesh.

Note: To retrieve the original point cloud from which a mesh object was created, you can use the [ScanWorld Explorer](#) to make the points visible, or you can use the [Insert Copy of Object's Points](#) command.

Using Cutplanes

The Cutplane is a 3D plane with the primary function of "cutting" through objects in a ModelSpace to produce a 2D cross-section. There may be more than one Cutplane cutting through the ModelSpace, although only one is the "active" Cutplane at any given time. The active Cutplane is the only Cutplane affected by user commands. The Cutplane can be used to create polylines from its intersections with objects and meshes, and the active Cutplane can be used to hide all geometry above itself, or to hide all geometry beyond a user-specified distance from its plane.

- The **Cutplanes** dialog is used to create, delete, and edit Cutplanes.

Positioning the Cutplane

Raising and Lowering

The Cutplane can be raised/lowered by user-defined increments using the **Raise Active Cutplane** and **Lower Active Cutplane** commands. The increment by which the active Cutplane

is raised and lowered using these commands is set in the **Cutplane Offset** dialog. This offset is used each time you raise or lower the Cutplane.

Tip: When manipulating a Cutplane repeatedly, it can be more efficient to assign hotkeys (see the *Customize Hotkeys* entry in the *Commands* chapter) to the **Raise Active Cutplane** and **Lower Active Cutplane** commands.

Aligning

The active Cutplane can be aligned to the active Reference Plane using the **Set from Ref Plane** command. The Cutplane can also be aligned to a reference object using the **Set on Object** command. This command aligns the active Cutplane through a pick point and along the axis of the selected object (for point clouds, the surface normal of the picked point is used as the axis).

Cutplane Operations

View Half-Space

Once the Cutplane has been positioned, you can visually hide all geometry above the active Cutplane using the **View Half-Space** command. This produces a helpful visualization of the cross-section and everything below. It can be useful to see "inside" the surfaces that are left visible.

- To toggle the visibility of the geometry, use the **Flip Cutplane Normal** icon in the **Cutplanes** dialog.

Slice

It can be useful to create a "slice" around the active Cutplane, beyond which all geometry is hidden. The results of the **View Slice** command resemble a cross-section that can be used for quick 2D plots directly from point clouds and surfaces.

- To set the thickness of the slice, use the **Set Slice Thickness...** command.

Create Lines

The Cutplane can also be used to create polylines at the intersection of the active Cutplane with objects and meshes. The polylines that result from using the **Create Lines from Cuts** command are separate from the objects used to create them and can be manipulated independently. Likewise, changes to the source objects do not change the polylines once they have been created.

Fit Edge

Cyclone's **Fit Edge** tool can be used to generate polylines describing a swept edge best-fit to a point cloud.

The Fit Edge Process

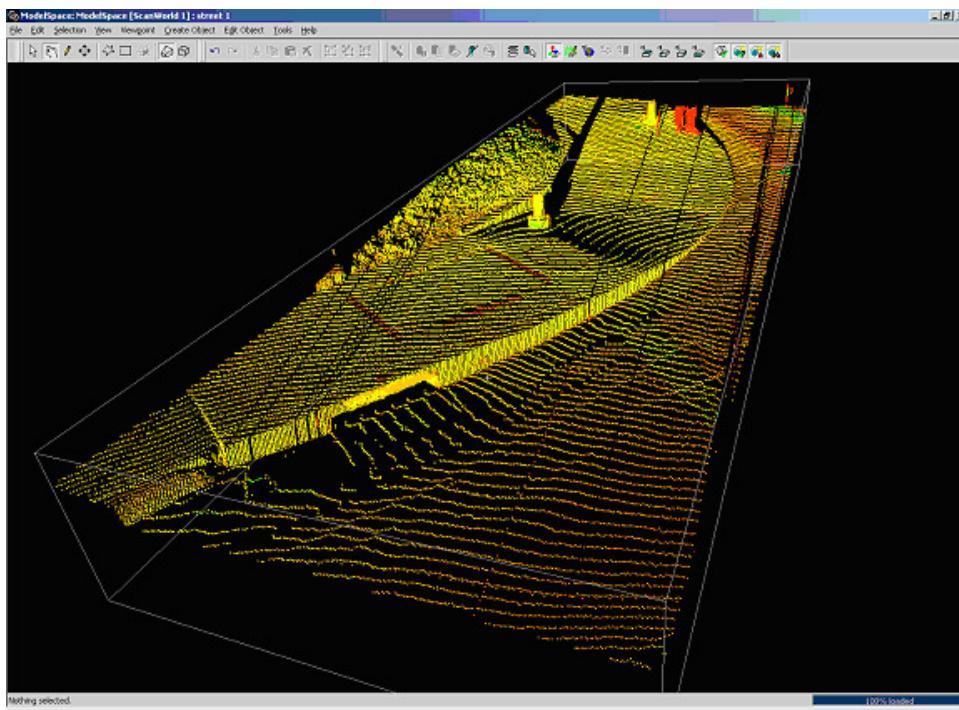
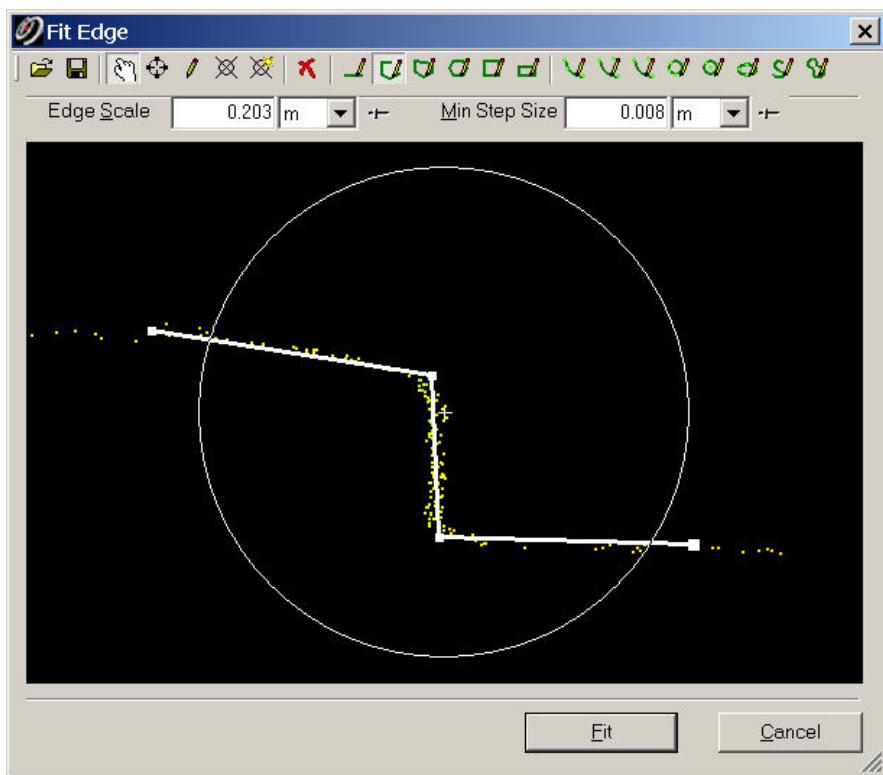
The following is an outline of a typical work-flow:

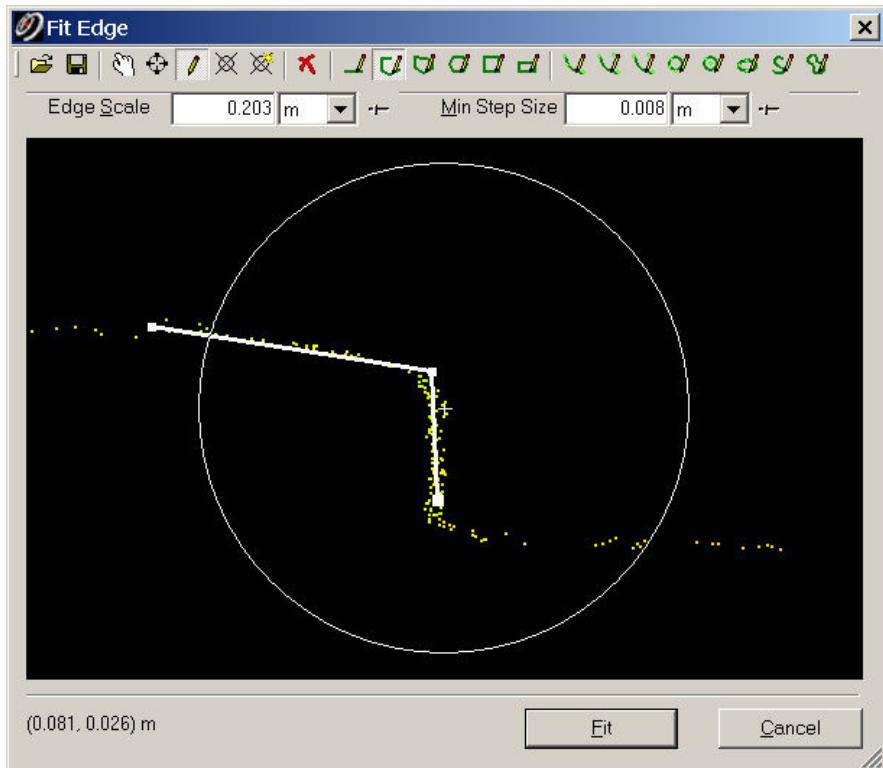
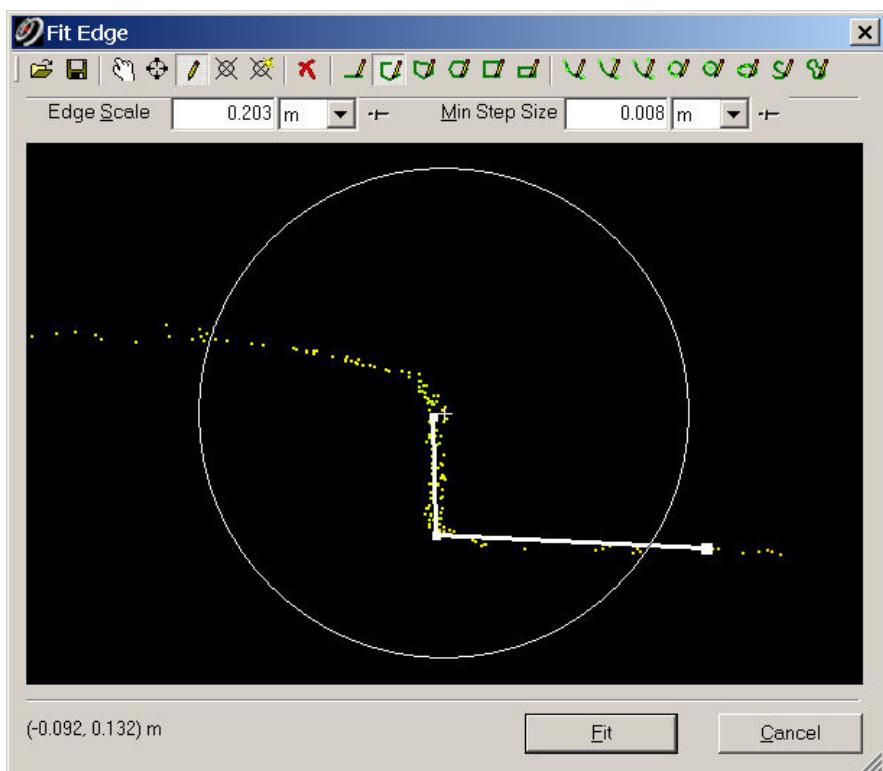
- All edge fitting is done via the **Fit Edge** dialog, which is in the **Create Object** submenu of the **ModelSpace** window.
- 1. Select one or more point clouds to which you want to fit an edge.
- 2. From the **ModelSpace** window, open the **Fit Edge** dialog, and then manipulate the viewpoint so that the cross-section of points is apparent.
- 3. Draw a 2D template using the selected point cloud as a reference.
- 4. Add nodal points to the 2D template.
 - The nodal points indicate the position at which polylines are created and swept across the point cloud based on the fit.
 - Hold the Alt key and click on a nodal point to delete it.
- 5. Fit the results to the selected point cloud to create the swept polylines.

Fit Edges Using Simple Templates

The following example of finding the edges of a street curb illustrates how complex structures may be effectively modeled in simple pieces.

Figure 1 shows a scan of a street corner within a limit box. (See [Figure 1](#)). In order to model the upper and lower edges of the curb, one approach is to use the Create Object - Fit Edge command to draw a polyline describing the cross-section of the curb (See [Figure 2](#)). However, because the curb changes height at the wheel chair access ramp and at the drain, this template only describes the points for a short distance. To avoid this problem, split the curb template split into two parts: the upper edge and the lower edge (See [Figure 3](#) and [Figure 4](#)). By using these two templates separately, we are able to model the top edge over the drain and into the wheelchair access ramp.

Fit Edges Illustration 1**Fit Edges Illustration 2**

Fit Edges Illustration 3**Fit Edges Illustration 4**

Color Mapping

In *Cyclone*, the default color mapping scheme works as follows: each cloud is assigned a range of intensities based on the highest and lowest intensity values of the sampled points. These high and low values are mapped to the highest and lowest possible colors in the given scheme (hue-based, grayscale, etc.) producing the largest range of colors, which aids visualization. This is done for each individual point cloud. A consequence is that two points of identical intensity value that appear in different clouds are often mapped and colored differently.

- To view a cloud/mesh's color map, select the point cloud/mesh and point to **Edit Object | Appearance | Apply Color Map** and select the type of color map.

-or-

Select the point cloud/mesh and select the desired color map from **View As** tab of the **View Properties** dialog. See the *View Object As..* entry in the *Commands* chapter.

- For definitions of the available types of color maps, see the [Edit Color Map](#) entry in the *Commands* chapter.
- To edit the color map for the selected cloud/mesh, select **Edit Color Map...** from the **Appearance** submenu of the **Edit Object** menu.

Global Color Map settings allow users to designate the color mapping for all clouds simultaneously, based not on the original or current range of its source scan but on the actual intensity value of each of its sampled points, so that all points with the same intensity are drawn with the same color. When Global Color Mapping is ON, it overrides any individual color map parameters or point coloring for individual point clouds and meshes.

- To toggle Global Color Mapping ON/OFF, select [Global Color Map](#) from the **Appearance** submenu of the **Edit Object** menu.
- To edit the Global Color Map, select [Edit Global Color Map...](#) from the **Appearance** submenu of the **Edit Object** menu.

Object Preferences

The **Object Preferences** dialog provides a central location where object modeling defaults may be configured. The dialog is available from the **Navigator** window (to set user default and session-level preferences) and ModelSpace Viewers (to set user default, session-level, and ModelSpace View-specific preferences).

Object Information

Cyclone displays information for any selected object within a ModelSpace when the **Object Info** command is selected from the **Info** submenu. The type of information displayed depends on the type of object selected.

- When more than one object is selected, information is displayed for each object in its own **Object Info** dialog. These dialogs can be moved and closed individually.
- When the **Automatically Update** box is checked in the **Object Info** dialog, the object information changes when the object changes. If the box is unchecked, the contents of the

Object Info dialog for that object do not change. Disabling the automatic update can be useful when performing a “before and after” comparison of values.

Working with the ModelSpace

ModelSpace Navigation

Changing the Viewpoint

Changing the Viewpoint

A viewpoint is a viewing position or camera angle used to simplify ModelSpace navigation. *Cyclone* provides controls in **View** mode that allow you to manually zoom in/out, and rotate or pan around a focal point using a mouse. The viewpoint can also be zoomed in/out, moved, and rotated around the focal point by preset increments using the commands in the **Zoom**, **Move Viewpoint**, and **Turn Viewpoint** submenus (located in the **Viewpoint** menu). The focal point is a certain distance in front of the center of the viewpoint; It can be reset in **Seek** mode and moved by panning the viewpoint.

Cyclone uses Level of Detail (LOD) reduction, which allows for faster loading of point clouds and smoother interaction and movement within a view when you are panning, rotating, or zooming.

- If your display has automatic LOD reduction enabled for point clouds, you can temporarily disable the LOD reduction by holding down the left **SHIFT** key while panning, rotating, or zooming.

Manual Viewpoint Controls

- The ability to pan, zoom, or rotate can be disabled temporarily via the **View Lock** submenu commands. This can be useful when you need to refrain from rotating the viewpoint, for example.

ROTATE

To rotate the viewpoint around the focal point, drag (press and hold the left mouse button while moving the mouse) while in **View** mode.

PAN

Right-drag (press and hold the *right* mouse button while moving the mouse) while in **View** mode.

ZOOM

Press and hold both mouse buttons and move the mouse down/up to zoom in/out.

Advanced Manual Viewpoint Controls

ROTATE FOCAL POINT

To rotate the focal point around the viewpoint, hold **CTRL** and drag while in **View** mode.

ROLL

To roll the viewpoint around the current forward direction (similar to an airplane barrel roll), hold **CTRL** and right-drag while in **View** mode. **Viewpoint | Keep Viewpoint Upright** must be disabled to use this control.

ZOOM FOCAL POINT AND VIEWPOINT

To move both the focal point and the viewpoint forward/backward, hold **CTRL** and both mouse buttons while dragging up/down while in **View** mode.

To reset the focal point:

1. From the **Edit** menu, point to **Modes**, and then select **Seek Mode**. The cursor changes to the **Seek Mode** cross-hair.
2. Pick the point that you want as your new focal point. The viewpoint is adjusted and the focal point is set.

Standard Viewpoints

Your viewpoint can be switched at any time to one of several standard viewpoints using the **Standard Viewpoints** submenu commands. Standard viewpoints include: **top**, **bottom**, **left**, **right**, **front**, **back**, **isometric**, and **right isometric**. See the corresponding commands in the *Commands* chapter.

Aligning the Viewpoint to Objects

The viewpoint can be aligned to a reference object in the ModelSpace with the **Align to Selection** command. This command moves the viewpoint the shortest possible distance in order to be parallel with the axis of a selected object.

- For a selected point cloud, the viewpoint is aligned with the position of the scanner that scanned the point cloud (if available).

Zoom Control

In addition to manual zoom controls (described above), *Cyclone* provides commands to zoom the viewpoint in or out by preset increments (from 1.5 to 2 times larger or smaller), or to zoom to fill the view with a fence or selection.

Saving and Restoring Viewpoints

The ability to save and restore several viewpoints can be helpful in the navigation of larger ModelSpaces. User-defined viewpoints can be saved, deleted, and restored using the **User Viewpoints** dialog. You can save viewpoints that are useful for modeling or provide a specific viewpoint for later restoration.

Maintaining Up Direction

Cyclone lets you pan, rotate, and zoom the viewpoint in 3D. However, you may prefer not to rotate the view with all three degrees of freedom (roll, yaw, and pitch) turned ON. For example, it is not always obvious which way is "up". The **Keep Viewpoint Upright** command removes the "roll" component of the viewer in order to keep the "up" axis oriented upward in the viewer while rotating the viewpoint. That is, you can rotate left and right, and up and down, but you can't view a scene upside-down.

- A ModelSpace's "up" and "down" are initially defined by the current coordinate system. By default, "up" follows the same direction as the positive Y axis.
- A ModelSpace's "up" can be set via the [Set Up Direction](#) or the [Set from Points...](#) command.

ModelSpace Views

A ModelSpace View is a collection of settings applied to a ModelSpace, including the initial viewing position and settings for graphical viewing parameters that override the object's default parameters. Multiple ModelSpace Views can be saved and restored for a given ModelSpace to provide ready access to particular aspects of a ModelSpace, or to provide a custom modeling environment for a modeler in a large, multi-person project.

All changes made to objects within the ModelSpace are immediately reflected in all of the ModelSpace's Views. Multiple users may share a single view; one user's changes to the shared View are visible to the other users of that View.

- To save ModelSpace Views, use the **Save View As** dialog.

Note: The View must be saved at least once before changes to the View are made persistent and before the View is available to other users.

- To restore saved ModelSpace Views, use the **Switch to Views** dialog or open them from the **Navigator** window.

Orthographic/Perspective Views

A ModelSpace can be toggled between orthographic and perspective projection by clicking the **Orthographic** or Perspective icons, or by selecting the **Orthographic/Perspective** command in the **Viewpoint** menu.

- In *orthographic* projection, parallel lines remain parallel, though perceived angles may change. It may be difficult to determine relative depth, but this view is often preferred by architects and engineers in top-down or side views. The real world, however, is viewed in perspective.
- In *perspective* projection, parallel lines may converge and relative depth may be easier to visualize.

Coordinate System

When a ModelSpace is contained by a ScanWorld, the default coordinate system is set to that of the ScanWorld. After changing the active coordinate system in the ModelSpace View, the default coordinate system can be restored by selecting the Default coordinate system in the **User Coordinate Systems** dialog (see the *Save/Edit Coordinate Systems* entry in the *Commands* chapter). When working with larger and more complex ModelSpaces, it can be helpful to define and save multiple coordinate systems for navigating and analyzing the ModelSpace. Coordinate systems are stored with the ModelSpace View.

- The current coordinate system for a ModelSpace View can be copied into its parent ScanWorld. This is done via the **Set ScanWorld Coordinate System** command. After a ScanWorld's coordinate system is reset, any subordinate ModelSpace Views are created using the new coordinate system.

Editing the Coordinate System

Individual coordinate system axes can be set to the axis of a reference object using the **Set Using One Axis** submenu commands, while the remaining two axes are arbitrarily assigned, perpendicular to the one axis. Using the **Set Using Two Axes** submenu commands, all three coordinate system axes can be set at once to the axes of two reference objects. With the **Using Two Axes** submenu commands, the first reference axis is used exactly, and the second is used to find an axis perpendicular to the first axis as close to the second reference axis as possible. The third axis is set perpendicular to the first two and ensures that a right-handed coordinate system is created.

Additionally, the coordinate system's origin can be set to a picked point via the **Set Origin** command in the **Coordinate System** submenu. The **Set Origin** procedure does not affect the orientation of the axes.

- The coordinate system can also be edited via the **Set from Points** command, using pick points. This command allows you to set any or all of the following: reference point (coordinate system origin), azimuth (horizontal orientation), and "up" direction (vertical orientation).

Layers

As ModelSpaces become larger and more complex, using layers can improve modeling efficiency. Layers are used to organize objects into logical groups that can be hidden, made selectable/unselectable, and colored uniformly. Unlike "grouping" (see the *Group* entry in the *Commands* chapter), which causes several objects to act as a single object, objects in layers maintain their individual properties. One advantage to using layers is that the visibility of individual layers can be toggled ON/OFF, so certain layers can be hidden while others remain visible. For example, it is possible to have all of the "pipes" in a ModelSpace on one layer. Turning OFF all other layers would then make it easier to focus on modeling or viewing only the pipes. An added benefit to working on one layer at a time is that it may improve system performance, because *Cyclone* does not need to render objects or point clouds on hidden layers. It can also be useful to keep a layer visible but unselectable in order to use that layer's objects as a visual reference without being able to alter them while modeling.

Note: Layer properties (e.g., visibility and color settings) are set at the ModelSpace View level, and are not shared by other users working in the same ModelSpace. This enables multiple users to use Layer settings to personalize their Views.

Basic Layer Functions

Layer functions are controlled from the **Layers** tab of the **View Properties** dialog. From this dialog, layers can be created, merged, renamed, deleted, and designated as the *current* layer (the layer to which new objects belong by default). Additionally, the visibility and selectability of layers can be toggled ON/OFF. Finally, selected objects in the ModelSpace can be assigned to a layer, and all of an existing layer's visible objects can be selected/deselected.

To assign selected objects in the ModelSpace to a specific layer:

1. Select the desired objects, and then select the layer you want in the **Layers** tab of the **View Properties** dialog.
2. Click the **Assign** button. The objects are assigned to the selected layer.

To select or deselect all objects in a layer:

1. From the **Layers** tab of the **View Properties** dialog, select the relevant layer.
2. Click the **Select/Deselect** button. All of the layer's visible objects are selected/deselected in the ModelSpace. If the layer or object type is unselectable or invisible, then selecting the layer has no effect.

Assigning Colors to Layers

Each layer is assigned a default color that can be edited. A layer's objects can be displayed using either the layer's color or the objects' individual colors.

- To display a layer's objects using the layer's color, select the box next to the layer's color box. When this box is not selected, objects maintain their individual color and material properties.

To edit a layer's color:

1. Click the color box next to the layer for which you want to change the color. The **Color Picker** appears.
2. Make the desired adjustments to the layer's color.
3. Click **OK** to close the **View Properties** dialog. The layer's color is updated.

Assigning an Object Type to a Single Layer

In some cases, modeling efficiency can be improved by assigning all objects of a particular type to a single layer.

Lighting

Surfaces in *Cyclone* are filled in and shaded with color and material, which are, in turn, affected by light sources in the scene.

The default light sources in a scene are the head light, which is tied to the viewpoint, and the ambient lighting. Additional light sources (point lights and spot lights) may also be inserted in 3D.

- Non-surface shapes such as point clouds and thin lines do not respond to lighting. The only material property that affects their appearance is the diffuse material property.

Default Light Sources

Ambient Light

This is the amount of “natural” light in the scene; it has no position or direction, and affects all surfaces. The ambient light interacts with the ambient material property of an object. The ambient light is a dim white light by default.

- To adjust ambient light color and intensity, use the **Light Settings** dialog. (See the *Edit Environmental Lights* entry in the *Commands* chapter.)

Head Light

This is a light source with a position and direction matching those of the viewpoint. Wherever the view is pointing, so is the head light. The head light interacts with the diffuse material property of objects for natural shading effects and with the specular material property of objects for highlights. The head light is a bright white light by default.

- To adjust head light color, intensity, and ON/OFF state, use the **Light Settings** dialog. (See the *Edit Environmental Lights* entry in the *Commands* chapter.)
- To select the head light type (directional or point source), go to the **ModelSpace** tab in the **Edit Preferences** dialog.

3D Light Sources

Point Light

A point light source emits light in all directions from a 3D point. By default, the light is bright white and does not dim with distance. A point light’s color, attenuation, position, size, and ON/OFF state can be edited within the **Light Properties** dialog. (See the *Edit Point/Spot Light* entry in the *Commands* chapter.)

- To insert a point light: From the **Tools** menu, point to **Lighting**, and then select **Insert Point Light**. The point light is inserted at the current pick point. If there is no pick point, the point light is inserted at the current viewpoint.

Spot Light

A spot light emits light within a cone from a 3D point. The centerline of the cone determines the direction of the spot light source, and the tip determines the origin. The angle of the cone

determines the spread of the spot light. By default, the light is bright white and does not dim with distance; the cut-off of the light at the edge of the cone is immediate.

- To insert a spot light: From the **Tools** menu, point to **Lighting**, and then select **Insert Spot Light**. If there is a pick point on an object, the object provides the location and (in most cases) orientation for your spot light. If there is no pick point, the spot light is inserted at the current viewpoint.

Fog

The fog setting adds a sense of depth to a ModelSpace by making more distant objects appear darker.

- To toggle fog ON/OFF, select **Fog** from the **Lighting** submenu of the **Tools** menu.
- Having Fog ON while segmenting clouds may make distant points difficult to see and can lead to modeling and measurement errors.

Limit Box Manager

Limit boxes allow you to restrict the scope of objects and points loaded, improving performance and facilitating comprehension of point clouds. The **Cyclone Limit Box Manager** (accessed from the [Add/Edit Limit Boxes](#) command) allows you to define and edit multiple limit boxes for each ModelSpace, select from among your defined limit boxes, and import and export limit boxes for use in other projects.

- Limit Box commands are found in the **View** menu in the **ModelSpace** window.

Drawing

In addition to creating best-fit objects and inserting objects, *Cyclone* provides interactive tools for drawing 2D objects in a 3D space. This allows you to create shapes that are otherwise unavailable in *Cyclone*, and to create shapes where fitting or object insertion are impractical.

- There may be many Reference Planes in a given ModelSpace View; however, all drawing is done on the *active* Reference Plane. The active Reference Plane does not need to be visible to perform 2D drawing.

Supported Shapes

Line, Polyline, Square, Rectangle, Polygon, Isometric Polygon, Text, Arc, Circle, Ellipse, Cubic Spline, Cubic Spline Loop

Drawing 2D Objects

In general, 2D objects are drawn by selecting the type of object you want to draw from the **Tools | Drawing** submenu, which puts *Cyclone* in **Drawing** mode, and then plotting a sequence of vertices in the active Reference Plane in the ModelSpace. A drawing is completed by pointing to **Drawing** within the **Tools** menu and selecting **Create Drawing**. Slight variations in the way that different shapes are drawn are documented in the individual command entries for each supported shape in the *Commands* chapter of this manual.

Plotting Vertices

To place a vertex in the plane, you can either click or drag the cursor while in **Drawing** mode. If you click, one vertex is created. If you drag the cursor before releasing, the new vertex tracks the cursor. When you release the mouse button, the vertex is inserted at the current position.

- For polylines and cubic splines, hold the **ALT** key and drag the mouse to insert a stream of vertices along the path of the cursor.
- To align a vertex along a grid axis of the Reference Plane from the previous vertex, first enable **Align Vertices to Axes**, then drag the cursor in the direction you want. The second vertex snaps along the closest Reference Plane axis from the first vertex.

Editing Vertices

At any time before a drawing object is complete, vertices can be moved and deleted. When deleting vertices, the resulting shape depends on the order in which they were plotted. For example, an arc requires three vertices - the first two define its endpoints, and the third defines the point through which the curve of the arc passes. If you have plotted all three, and then delete the first vertex plotted, the remaining two become the first and second endpoints. Likewise, if you have plotted all three and then delete the second, the first and third then become the first and second endpoints respectively. If the shape you are drawing already has enough vertices (e.g., a line segment with two vertices), plotting a new vertex replaces the last vertex placed.

- To translate the current drawing on the reference plane, hold the **CTRL** key and drag the mouse.
- To delete a vertex, hold the **ALT** key and click on the vertex without moving the mouse.
- To move a vertex, drag it to the desired location. Depending on the shape being drawn, the other vertices may be moved accordingly. For example, moving one vertex of a square which already has four sides drawn, moves the other vertices as necessary in order to maintain the integrity of the square.
- To cancel a drawing in progress, select another drawing command or select **Cancel Drawing** from the **Drawing** submenu.
- To remove a shape that you have just created, select **Undo** from the **Edit** menu, or select the object, and then select **Delete** from the **Edit** menu.
- When **Snap to Grid** is ON, vertices jump to the next increment on the Reference Plane's grid as you plot them (press and hold **ALT** to snap at angular intervals). When OFF, vertices are

plotted/moved independently of grid settings.

Inserting Vertices

For polygons and polylines being drawn, a new vertex can be inserted by holding the **ALT** key and clicking on an existing line segment within the active drawing.

Converting Drawings

A drawing may be converted into another shape (if appropriate). For example, a line may be converted into a polyline, into an arc (with the first two vertices defined), etc. An arc may be converted into a circle or ellipse, but not into a line unless only one or two of its vertices have been plotted. To convert a drawing into another type, select the new type of drawing. If conversion from the previous type of drawing into the new type of drawing is supported, the **Convert** button is available.

Multiple Drawings

More than one drawing can be in progress. To begin drawing another shape, select another drawing command (of the same or different type). If you have plotted at least one vertex, the **Keep** button is available.

Only one drawing can be active. To resume drawing another drawing in progress, move the cursor over the existing drawing (the cursor changes shape) and click. This activates the other drawing. All drawings in progress may be created or cancelled at the same time.

Pipe Modeling...

Pipe Modeling is used to insert piping components in a model, without best-fitting to a point cloud. Placement of the components is typically performed using the point cloud as a reference. Components can be inserted in sequence to create a run of connected piping components. The components can be assigned information, such as piping specification, line ID, insulation thickness and Symbol Keys. This information can be exported and mapped to various plant design software applications.

- Pipe Modeling is accessed through the **Pipe Modeling** command. See the [*Pipe Modeling*](#) entry in the *Commands* chapter.

A typical workflow is as follows:

1. Select an existing object (e.g., from region growing, fitting, or insertion) to which a piping component is to be added, or start with nothing selected to start a new sequence.
2. Using the **Pipe Modeling** dialog, insert the connected piping components with supplemental piping-specific information in *Cyclone*.
3. Export the ModelSpace to PCF.
4. Using the **I-Convert** (from Alias Ltd) converter, process the PCF file into a format compatible with the target piping design package.

- **I-Convert** currently supports PDS, PDMS, and AutoPlant.

Piping Mode...

Piping Mode is used to add piping component information to a model in the form of annotations. The component information includes piping specification, line ID, insulation thickness and Symbol Keys. This information can be exported and mapped to various plant design software applications.

- Piping Mode is accessed through the **Piping Mode** command. See the [Piping Mode](#) entry in the *Commands* chapter.

A typical workflow is as follows:

1. Model objects in *Cyclone*.
 2. Using the **Piping Mode** dialog, add piping-specific information. Piping-specific information may also be added *during* the modeling process.
 3. Export ModelSpace to PCF.
 4. Using the **I-Convert** (from Alias Ltd) converter, process the PCF file into a format compatible with the target piping design package.
- **I-Convert** currently supports PDS, PDMS, and AutoPlant.

Animation

Cyclone's animation tool can be used to generate a series of frames along an editable path through a ModelSpace for a fly-through animation. The ModelSpace should remain static (individual objects cannot be edited in any way) throughout the animation. However, multiple animations can be created and combined to produce this effect.

The Animation Process

Creating an animation can be accomplished in a variety of ways. The following series of steps represents an outline of a typical project.

- All animation commands and dialogs are found in the **ModelSpace** window within the **Tools | Animation** submenu.
1. In the **ModelSpace** window, define an animation path by inserting Camera objects along the path in which you wish to create an animation (**Create Object | Insert | Camera**). The Camera objects serve as keyframes.
 2. [Create a path](#) by selecting the Camera objects and then selecting the [Create Path](#) or [Create Path \(Loop\)](#) command. A spline or spline loop connecting the Camera objects is created.
 3. Set the path via the [Set Path](#) command, which associates the keyframe cameras with the spline and allows the animation tools to use the spline as the animation path. Once the path is

- set you may begin the editing process.
4. Open the [Animation Editor](#) dialog, and [edit the animation](#) by setting keyframes and specifying the number of frames between keyframes. In the **ModelSpace Viewer**, edit the animation by adjusting the spline shape and camera angles.
 5. Preview the current animation via the **Animate** button in the [Animation Settings](#) dialog (**Tools | Animation | Animate...**). For previewing, the **Output** setting should be **In Window Only**.
 6. When finished editing, [create animation frames](#) by changing the **Output** setting to **To File (prefix)** in the **Animation Settings** dialog and then clicking the **Animate** button. The specified sequence of frames is saved as numbered files in the specified folder and can be used in your animation software to create the final animation.

Creating a Path

The first step in the animation process is creating a path along which the animation moves.

To create an animation path:

1. Define the path.
 - Insert a series of Camera objects at intervals along the path you want the camera to follow in the final animation. The Camera objects serve as key frames.
2. Create the path.
 - Multi-select the Camera objects in the same sequence that you want the camera to move, and then select [Create Path](#) or [Create Path \(Loop\)](#) from the **Animation** menu. A spline curve or closed spline loop connecting the Camera objects is created.
 - Alternatively, a path can be created via *Cyclone*'s drawing tools by drawing a cubic spline or cubic spline loop (instead of inserting Camera objects). However, for predictable results, this method requires the camera to be oriented towards a single focal point throughout the animation. (The **Keep Current Focal Point** box should be selected in the [Animation Settings](#) dialog.) Using the camera objects method, the focal point and viewpoint orientation are defined by each inserted Camera object.
3. Set the path.
 - With the spline and cameras selected, select **Set Path** from the **Animation** menu. A green spline with tangent manipulation handles appears connecting the Camera objects. Once the path is set, you may begin editing the animation.

Note: The path is not saved with the database, so each time a ModelSpace viewer is opened the path must be reset via the [Set Path](#) command in order to continue editing or generating frames.
4. Edit the path.
 - Use the handles connected to each point in the spline or spline loop to edit the shape of the curves in each section of the spline.
 - Use the camera translation and rotation handles to edit the viewpoint at each keyframe.
 - To clear the current path, select [Clear Path](#).

Editing the Animation

The second step in creating an animation is defining the animation characteristics via the [Animation Editor](#) dialog.

To edit the animation:

1. From the **Animation** submenu, select **Animation Editor**. The [**Animation Editor**](#) dialog appears.
- Set keyframes, specify the number of frames between keyframes, adjust other settings and then click **OK**. For details on **Animation Editor** dialog settings, see the [**Animation Editor**](#) command.
2. Preview the animation with the current settings.
 - To preview the animation without the Camera objects or animation path, use the **Set Visibility** command to turn visibility OFF for those object types.
 - From the **Animation** menu, select **Animate**. The [**Animation Settings**](#) dialog appears. Select **In Window Only**, and then click **Animate**. The animation is displayed in the current ModelSpace window.

-or-

- From the **Animation** menu, select **Animation Editor**. The **Animation Editor** dialog appears. Press the arrow icon by **Frame** or **Parametric Time**, and then press and hold the **ENTER** key. The animation is displayed in the current ModelSpace window.

Generating the Frames

The final step is generating the actual frames for the animation via the [**Animation...**](#) command.

1. From the **Animation** submenu, select **Animation....** The [**Animation Settings**](#) dialog appears.
2. Set output options and then click **OK**. Individual frames are saved to a series of files with a user-defined prefix, available for use with your animation software.

Databases

Databases

Cyclone databases can reside on your local hard drive or on a remote server, which can be a network server or another networked workstation. The advantages to having a database stored on a networked server are many:

- ◆ Multiple users can easily access the database and work concurrently.
- ◆ Database integrity can be protected by limiting access to the host server.
- ◆ Storing a database on a remote server puts the bulk of memory load on the server, rather than on an individual's workstation

Cyclone provides tools for remote *Cyclone* database administration, and options for maximizing database storage and performance.

Unshared Database Server

Databases added to the “(unshared)” server can only be opened by the local client, but the performance of disk-based operations (e.g., loading points) can be significantly improved.

Remote Administration

Databases can be stored on a networked server and administered remotely from a user workstation using the **Configure Database** dialog. This is a secure feature that requires the database administrator to log in by entering a password. Once the login has been authenticated, the administrator can not only add and remove databases from the remote server, but can also destroy and compact databases on the remote server.

- Access to databases by other users can be restricted by toggling the visibility of selected databases in the **Configure Database** dialog.

To set up a server (or networked workstation) for remote administration:

Note: This feature is applicable within *Cyclone* only, and does not permit you to perform functions beyond those defined in this section.

1. From the server console where the database resides, run the password utility.
2. From the Windows **Start** menu, point to **Programs | Leica HDS Cyclone 5.8**, and then click **Set Server Password**. The **Set Cyclone Server Password** dialog appears.
3. Set the password.
 - To set the password, type the desired password and click **OK**. At the confirmation prompt, type the password again and click **OK**.
 - To remove any existing password, leave the field blank and click **OK**.

To log on to a remote server:

1. From the **Navigator** window, open the **Configure Databases** dialog.
2. Select the **Admin Login...** button. A **Login** dialog appears.
3. Enter the password as configured on the remote server. The **Configure Databases** dialog is redisplayed with all buttons active.
- See the *Database* entry in the *Commands* chapter for more detailed information on the use of the buttons in the **Configure Databases** dialog.

Configure Databases Dialog Options

ADD

The **Add** button is used to add a database to the selected server in the **Navigator** window.

Note: A valid database file is one that is a *Cyclone* IMP file.

- This function is available to all users when administering their local machine; it is available only to authorized users administering a remote server.

REMOVE

The **Remove** button is used to remove a database from the **Navigator** window.

Note: Activating this button does not remove or delete files; it only removes the database object from the **Navigator** window. Use the **Add** button to reinstate a database that has been removed.

- This function is available to all users when administering their local machine; it is available only to authorized users administering a remote server.

DESTROY

The **Destroy** button is used to remove a database from the Navigator window and delete the database file from the computer or server where the database resides.

Note: Destroying a database is irreversible. Use this function with caution.

- This function is available to all users when administering their local machine; it is available only to authorized users administering a remote server.

COMPACT

The **Compact** button rewrites the selected database from the ground up to a second file, minimizing slack space in the new database file without increasing database size.

Note: It is recommended that you create a backup copy of the database when prompted to do so.

Note: Compacting a database can be a time-consuming process depending on the size of the database.

Note: Make sure you have enough free disk space to perform this operation (at least twice the size of the database being compacted).

OPTIMIZE

The **Optimize** button optimizes the selected databases (see the *Optimize* entry in the *Commands* chapter).

Optimizing a Database

With the degree of changes in this version of *Cyclone*, older databases must be optimized before they may be used. Optimizing a database makes sure that the data in the database is consistent; new objects (e.g., ControlSpaces) may be created as needed.

Note: Optimizing a database can be a time-consuming process depending on number of objects to be processed.

Virtual Surveyor

Cyclone's **Virtual Surveyor** tool can be used to generate traditional survey output from a scanned model in *Cyclone*. Virtual Surveyor allows you to pick points in a ModelSpace, assign feature codes, and export data in custom text formats that can be imported into other software programs.

The Virtual Surveyor Process

Creating survey data can be accomplished in a variety of ways. The following series of steps represents a typical project. Your specific projects may differ from the following general guidelines:

- All Virtual Surveyor commands and dialogs are accessed in the **ModelSpace** window from the **Tools | Virtual Surveyor** command.
- 1. Open the **Select Virtual Surveyor Input/Output** dialog, and [create a new file \(or open an existing file\) for data input/output](#). The **Virtual Surveyor** dialog appears.
- 2. From the **Virtual Surveyor** dialog, [set feature data](#).
- Feature data include prefix, starting number, feature code, and any notes that you wish to attach to the feature. The prefix and number are combined into a unique Point Number for each feature.
- Use the polyline creation feature to define linework (as a polyline or polygon) as you pick each feature.
- 3. [Pick a point for a candidate feature](#). The current feature data are attached to the candidate feature and displayed in parentheses.
- You may edit and/or lock coordinates for candidate features using the **X**, **Y**, and **Z** fields.
- 4. Multi-pick (continue picking while holding the **SHIFT** key) to add additional features with the current feature settings.
- Features are sequentially numbered in the order in which they were picked.
- 5. Confirm candidate features.
- Confirmed features are displayed with an asterisk following the feature data.
- 6. Continue picking and adding sequential sets of features with the appropriate feature attributes.
- 7. [Export all features and data](#) to the text file.
- Export formats can be customized to match the input format rules of your preferred survey data processing software.
 - Click **Create Vertices** to create one vertex for each feature. (Candidate features are ignored.)
 - Click **Export LandXML** to export features directly into the LandXML format. (Candidate features are ignored.)
 - Click **Export Leica System 1200** to export features as Leica System 1200 files. (Candidate features are ignored.) See **Export Leica System 1200** in the Command Reference.

Select a File for Data Input/Output

The first step in the Virtual Surveyor process is selecting a file for data input and output. You can create a new file, you can append or overwrite an existing file, or you can skip selection of a file and continue with Virtual Surveyor (you will be prompted for the file to save later).

To create a new file:

1. From the **Tools** menu in the **ModelSpace** window, select **Virtual Surveyor**. The **Select Virtual Surveyor Input/Output** dialog appears.
2. Navigate to the desired folder, enter a name in the **File name:** field, and click **Save**. The **Virtual Surveyor** dialog appears.

To overwrite an existing file:

- Overwriting a file discards all existing data in the file.
1. From the **Tools** submenu in the **ModelSpace** window, select **Virtual Surveyor**. The **Select Virtual Surveyor Input/Output** dialog appears.
 2. Navigate to the desired file and click **Save**. A confirmation prompt appears.
 3. Click **Overwrite**. A second confirmation prompt appears.
 4. Click **Overwrite**. All data in the current file is deleted.
 5. The **Virtual Surveyor** dialog appears.

To append an existing file:

- Appending a file opens a file with current data intact.
1. From the **Tools** submenu in the **ModelSpace** window, select **Virtual Surveyor**. The **Select Virtual Surveyor Input/Output** dialog appears.
 2. Navigate to the desired file and click **Save**. A confirmation prompt appears.
 3. Click **Yes**. A second prompt appears.
 4. Click **Append**. The **Import: ASCII File Format** dialog appears.
 5. Choose the appropriate import settings, and then click **Import**. The file is imported and the **Virtual Surveyor** dialog appears.
- See the *Import* entry in the *Commands* chapter of the *Cyclone User's Manual* for details on import settings.
 - Existing features and attached data may be displayed in the **ModelSpace**.

Set Feature Data

The next step in the Virtual Surveyor process is setting feature data for the next feature you add. Feature data include prefix, starting number, feature code, and any custom notes that you wish to attach to the feature.

To set feature data:

1. After selecting a file for data input/output, the **Virtual Surveyor** dialog appears.
2. Enter the desired feature data.

- The current feature data displayed in the dialog is applied to the next feature that you add as well as all subsequent features until the feature data is changed.
 - To customize attributes, push the **Customize Columns** button, or right-click the **Attribute** field and select **Customize Text Fields**. The **Customize Virtual Surveyor Attributes** dialog appears. This dialog can be used to create, delete and reorder attribute fields. When finished, click **OK**. The **Virtual Surveyor** dialog reappears.
3. You are ready to begin adding features.

Pick and Add Features

Once feature data have been set, the next steps are to pick and edit (candidate) features and then add or confirm the ones you want to keep.

To add features:

1. In **Pick** mode, pick a point you want to designate as a feature. The feature is considered a “candidate” until confirmed.
- The current feature data are attached to the candidate feature. The point number for a candidate feature is displayed in parentheses.
- Until confirmed, candidate features are removed with the next pick, unless the next pick is made while holding the **SHIFT** key.
- The **Auto-Pick** function (see the *Auto-Pick Points* entry in the *Commands* chapter) can be useful when adding features. It allows you to specify a significant point (i.e., the highest or lowest point on a cylinder) that is automatically picked when that object type is selected. When using auto-pick, the specified point is always the only picked point regardless how the object was selected (i.e., selected by fence, or picked at any other point on the object). For example, when fitting a cylinder, the cylinder is automatically selected and its top-most point may be auto-picked.
2. Multi-pick (continue picking while holding the **SHIFT** key or while in **Multi-Pick** mode) to pick additional features with the current feature settings.
 - Features are sequentially numbered in the order in which they were picked.
 - To use a different prefix or starting number for the next candidate feature, change the appropriate field and pick a new feature. Previously picked candidate features are not affected.
3. Edit candidate features.
 - To edit the coordinate data for the current candidate features, click the arrow icon next to the **X**, **Y**, or **Z** field and enter the desired data. All current candidate features are adjusted accordingly.
 - To lock the coordinate data for successive picks, click the lock icon next to the desired coordinate. A locked coordinate does not change in response to a new pick.
 - To change the feature code or note for all current candidate features, enter a new code or note in the appropriate field. The data for all candidate features are updated.

- To delete one of a series of current candidate features, press and hold **SHIFT** and re-pick the candidate feature you want to delete. The candidate feature is deleted. Note that other candidate features in the series are *not* renumbered, which may result in gaps in the series. However, you may fill in the gap, by entering the number (of the feature that you deleted) in the **Number** field and picking a new feature.
4. Click the **Add Features from Picks** button to add (or confirm) candidate features.
 - Confirmed features are displayed with an asterisk following the point number.
 - To revert added features to candidate features, click the **Undo** button.
 - Candidate features can be added as they are picked by clicking the **Auto-Add Features** icon.
 5. Continue picking and adding sequential sets of features with the necessary feature settings until all features are added.

To edit added features:

- The only features that may be deleted via Virtual Surveyor are the current candidate features.
1. In the **Virtual Surveyor** dialog, select the prefix and number of the feature to be edited (in the **Prefix** and **Number** fields).
 2. Edit the feature.
 - Edited features are displayed with an asterisk following the point number.
 - To edit the coordinate data for the current feature, click the arrow icon  next to the **X**, **Y**, or **Z** field and enter the desired data.
 - To change the feature code or notes for the current feature, enter the new code or notes in the appropriate field.
 - To back out of an edit without applying it, press **ESC**.

Create Polyline

You can simultaneously define a polyline or polygon as you pick features.

To create a polyline or polygon:

1. In the Create Lines area of the Virtual Surveyor dialog, click Polyline or Polygon to select the type of linework to be created.
2. Push the **Start** button in the **Create Lines** area.
 - The button text changes to **Cancel**. Push this button again to discard the linework before the features are committed.
 - To use a different prefix or starting number for the next candidate feature, change the appropriate field and pick a new feature. Previously picked candidate features are not affected.
3. Edit the code tags, which will be prepended to the feature code of each of the corresponding features.
 - The Begin tag is prepended to the first feature in the linework. The End tag is prepended to the last feature in the linework. The Continue tag is prepended to the remaining features

within the linework.

4. Pick on points to define the features that comprise the linework. Note the graphical preview of the linework.
5. Click the **Add Features from Picks** button to add the features as well as the linework. The linework is added as a polyline or polygon to the ModelSpace.

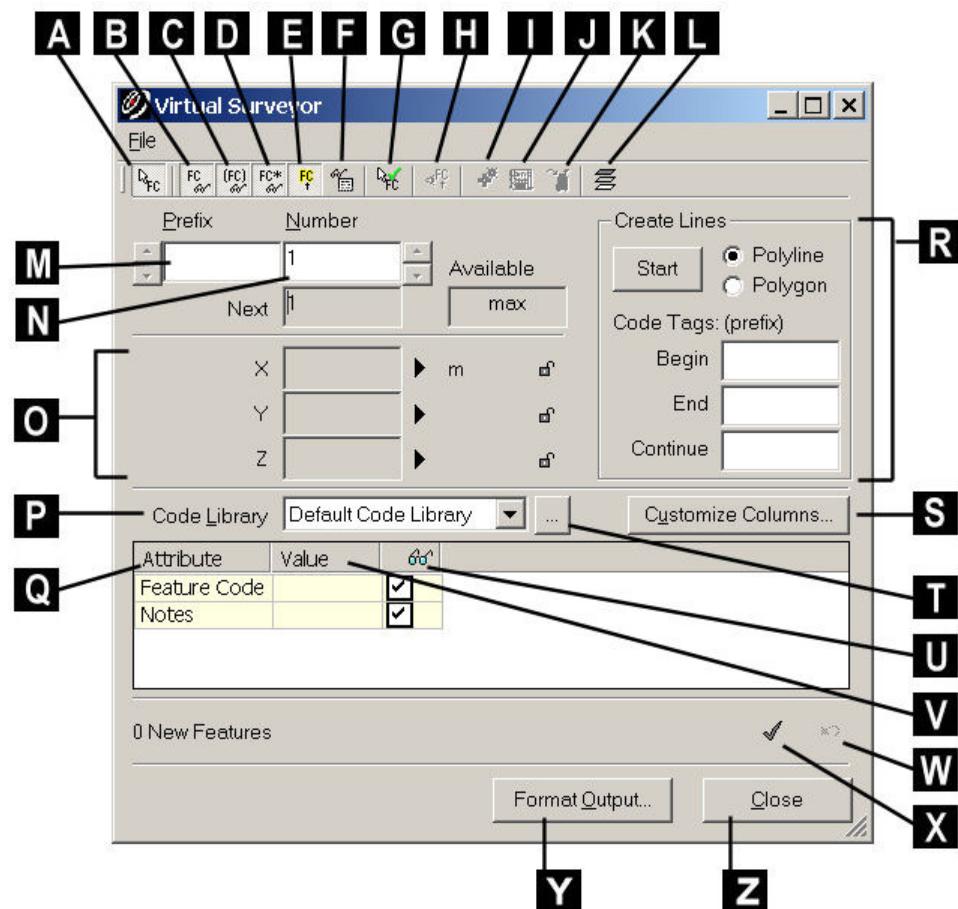
Export Data

The final step in the Virtual Surveyor process is exporting your data.

To export data:

1. When finished adding features, click **Format Output** in the **Virtual Surveyor** dialog. The **Export: ASCII File Format** dialog appears.
2. Choose the appropriate export settings, and then click **Save**. The file is exported to the selected file, and the **Virtual Surveyor** dialog reappears.
- See **Custom ASCII Export** section of the *Export* entry in the *Commands* chapter for details on import settings.

Virtual Surveyor Dialog



- A.** Click to add a candidate feature with each pick.

- B. Show features in the ModelSpace viewer.
- C. Show candidate features in the ModelSpace viewer.
- D. Show unsaved features in the ModelSpace viewer.
- E. Highlight the current feature in the ModelSpace viewer.
- F. Show features options, which affect how features are displayed.
- G. Select to add a feature with each pick.
- H. Change the viewpoint to view the current feature.
- I. Adds one vertex to the ModelSpace for each feature.
- J. Exports the features directly to LandXML.
- K. Exports the features directly to the Leica System 1200 format.
- L. Invokes the **Layers** command.
- M. The prefix of the current feature.
- N. The point number that will be assigned to the candidate feature created from the next pick.
- O. Displays coordinate data for the current feature. Click the arrow icon to edit the coordinate data. Click the lock icon to keep that value constant.
- P. Select a feature code library.
- Q. List of values for the available attributes.
- R. The **Create Lines** area. Select the creation of a polyline or polygon, supply the **Code Tags** that will be prepended to each corresponding feature, then click the **Start** button.
- S. Click to customize the attributes displayed.
- T. Invokes the **Customize Lookup Lists** command. See the [Customize Lookup Lists](#) entry in the Commands chapter for information on managing the libraries.
- U. Visibility of each attribute in the graphical window.
- V. Click to select or enter a value for the desired attribute.
- W. Undo last add features.
- X. Commit the candidate features.
- Y. Click to export current added features.
- Z. Click to close the **Virtual Surveyor**.

Sections Manager

The **Sections Manager** tool can be used to generate and manage sections of point clouds from a selected alignment.

The Sections Process

Creating and editing sections can be accomplished in a variety of ways. The following series of steps represents a typical project. Your specific projects may differ from the following general guidelines:

- All Sections commands and dialogs are accessed in the **ModelSpace** window from the **Tools** | **Alignment and Section** submenu.
 1. [Create and select an alignment](#).
 2. Open the **Sections Manager** dialog and [create sections](#) on the selected alignment using the **Create Sections** dialog.

3. [Use the Sections Manager tools](#) to navigate through the sections, modify section parameters, delete sections, collect the points that make up a section, and create section lines.

Create an Alignment

The first step in the process of creating sections is creating the alignment object you want to serve as the reference for your sections.

To create an alignment:

1. Select the line, polyline, or arc you want to use as an alignment.
 - The object used to create an alignment can be imported from your CAD software via the COE exchange format (see the *Cyclone Object Exchange* chapter). The object may also be created in a variety of ways including using *Cyclone*'s Drawing tools and creating the object via the **From Pick Points** submenu commands (see the *From Pick Points* entry in the *Commands* chapter).
 - When a single object is selected, the object must be picked near the end-point at which the starting station is assigned.
 - When multiple objects are selected, the objects must be geometrically connected in a single unbroken sequence. The object picked last provides the end-point at which the starting station is assigned.
 - Alignments may be horizontal, skewed, or a mixture of both, but may not be vertical.
2. From the **Tools** menu, point to **Alignment and Section**, and then select **Create Alignment**. The **Create Alignment** dialog appears.
3. Enter a **Starting Station Number** and click **OK**. The alignment object is created.
 - Station numbers increase with the linear distance "travelled" along the alignment, from the starting station.

Create Sections

The next step is to create sections from your alignment object.

To create sections:

1. Select the desired alignment object.
2. From the **Tools** menu, point to **Alignment and Section**, and then select **Create Sections**. The [Create Sections dialog](#) appears.
 - The **Create Sections** command can also be accessed from the **Sections** menu in the **Sections Manager** dialog.
3. Choose the desired alignment settings, and then click **OK**. The sections are created and the [Sections Manager dialog](#) appears.

Managing and Editing Sections

The final step is managing and editing sections from the **Sections Manager** dialog.

1. For existing sections, select the alignment, and then point to **Tools | Alignment and Section** and select **Sections Manager**. The [Sections Manager dialog](#) appears.
2. Use the **Sections Manager** dialog to create, delete, and edit sections. When finished click **Close**.
- To edit multiple sections simultaneously (see below), activate a single section, and then multi-select the desired sections from the sections list and make your edits. Changes are applied to all selected sections.

Create Sections Dialog Options

INITIAL STATION

Specify the distance along the alignment at which you want to place the first section.

END STATION

Specify the distance along the alignment at which you want to place the last section.

SPACING

Specify the interval between sections.

INCLUDE TRANSITION POINTS

Select this option to also create a section at each transition point (the point at which two of the original curves or arcs connect) that falls within the creation interval.

SECTION EXTENTS

All **Section Extents** settings are relative to the direction (from the start station towards the end station) of the alignment.

CREATE LINES

Select this option to create lines based on a temporary TIN mesh, which is created from the points within each section. More controllable results may be obtained by first creating the sections, and then adjusting sections and creating lines afterward via the **Create Lines at Section** in the **Sections Manager** dialog.

COLLECT POINTS

Select this option to introduce merged clouds from the points that fall within the boundaries of each section. This function can also be performed after sections have been created via the **Collect Points in Section** in the **Sections Manager** dialog.

Sections Manager Dialog Commands

- The **Section** menu contains commands to create, edit, and navigate through sections.

CREATE SECTIONS

This command opens the **Create Sections** dialog. See the *Create Sections* section for details.

CREATE LINES AT SECTION

This command creates lines for the selected section(s) based on a TIN mesh temporarily created from the points within the section's extents.

COLLECT POINTS IN SECTION

This command segments the points that fall within the extents of the selected section(s).

DELETE SECTION

This command deletes the selected selection.

ACTIVATE SECTION

This command activates the selected section and displays the section within a limit box (using the section's six dimensions). The viewpoint, a temporary Reference Plane, and a temporary limit box are aligned to the current section. Additionally, the datum and coordinate system origin and axes are temporarily aligned to the section. The section extents can be interactively edited by using the handles on the limit box.

DEACTIVATE SECTION

This command deactivates the active section and returns to the ModelSpace View.

ACTIVATE NEXT/PREVIOUS SECTION

These commands are used to activate the section following or preceding the currently activated section.

ZOOM/VIEW ALL

The Zoom/View All command is an ON/OFF toggle to determine whether to adjust the viewpoint so that the entire Section is visible in the current window.

FRONT VIEWPOINT

This command is an ON/OFF toggle to determine whether to adjust the viewpoint to Front view relative to the current UCS.

KEEP VIEWPOINT

This command is an ON/OFF toggle to determine whether to keep the current viewpoint when scrolling through the Sections in the Sections Manager.

- The **Alignment** menu contains commands to edit and navigate through alignments.

DELETE ALIGNMENT

This command deletes the selected alignment and all associated sections from the **Sections Manager**. The original alignment is not removed from the ModelSpace.

RESTORE ALIGNMENT INTO MODELSpace

This command is used to restore an alignment that may have been deleted from the ModelSpace.

EXPAND ALIGNMENT

This command displays the list of all sections associated with the selected alignment.

COLLAPSE ALIGNMENT

This command collapses the list of sections associated with the selected alignment.

To incorporate a polyline as a breakline into the temporary TIN mesh

1. Before using the **Create Lines** command, select the polyline objects(s).
2. The TIN mesh is extended to the polyline. The polyline is incorporated into the mesh; the polyline vertices are added into the mesh, and mesh triangles are redrawn as appropriate to accommodate the breakline.

Texture Mapping

Texture mapping is drawing objects using colors from an image. The colors are assigned based on the *mapping* defined for the image (the *texture* in this context). *Cyclone* supports defining and applying texture maps to point clouds and meshes.

The Texture Mapping Process

The basic process involves:

1. Adding the image from the database.
2. Matching points in 3D with pixels in the image.
3. Computing the optimal mapping.
4. Displaying the texture map.
5. Managing texture maps.

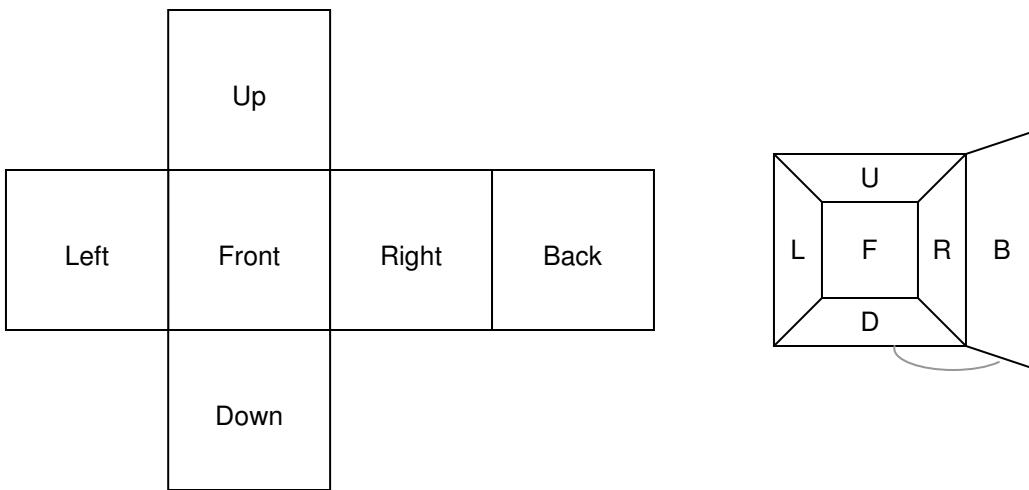
Most of these procedures are accessed via the **Texture Map Browser** command. Invoke the **Texture Map Browser** command to open the dialog.

Adding the Image

1. Select one point cloud or mesh. Most of the commands in **Texture Map Browser** support only one selected object at a time.
2. In the **Texture Map Browser**, click the **Add** toolbar button.
3. Browse to and select an Image using the **Select Image** dialog.
 - For convenience, images can be imported into the database from this dialog. Push the **Import** button, then select one or more image files.
4. In the **Texture Map Type** dialog, indicate the image's type. Most images will be "Perspective". "Orthographic" images include posters, wallpaper, or other flat images.
5. A texture map is added to the object with that Image. The **Texture Editor** dialogs appear.

Adding a Cube-Map

A cube-map is a special set of images that correspond to the six faces of a cube, with the camera placed at the center of the cube. The six images are: Up, Down, Front, Back, Left, Right. A cube-map in Cyclone consists of one to six of these cube images. Because of the nature and arrangement of the images, cube-maps provide a complete and convenient mapping of the full field-of-view. The images are arranged in this manner:



1. Select one point cloud or mesh.
2. In the **Texture Map Browser**, click the **Add (Cube-Map)** toolbar button.
3. Browse to and select a Project that contains cube-map images using the **Select Project with Cube-Map Images** dialog.
 - The cube-map images must follow a naming convention: There must be at most one image from each side of the cube in the Project, and each image must have a name ending in “_u”, “_d”, “_f”, “_b”, “_l”, or “_r”, for Up, Down, Front, Back, Left, and Right, respectively.
 - For convenience, images can be imported into the database from this dialog. Push the **Import** button, then select one or more image files.
4. A cube-map texture map is added to the object with that Image. The **Texture Editor** dialogs appear.

Editing and Computing a Texture Map

Texture maps are defined by correspondences (or constraints) between points in 3D with pixels on the image. Orthographic images need at least three constraints. Perspective images need at least six constraints, plus one for confidence, for a minimum of seven constraints. The **Texture Editor** dialogs provide access to constraint creation and management.

1. Select one point cloud or mesh.
2. In the **Texture Map Browser**, select a texture map and click the **Edit** toolbar button, or double-click on the texture map. The two **Texture Editor** dialogs appear.
 - The two **Texture Editor** dialogs are linked. Click the **Close** button in the **Texture Editor** to dismiss both.
3. Identify and pick points in the 3D scene that correspond to pixels in the image, then click the **Add** toolbar button in the **Texture Editor**. Pick features that are easy to identify (e.g., the corner of a room or sign) rather than more vague features. Constraints with higher correspondences result in better texture mapping.
 - When adding multiple picks, be sure to pick the 3D point and the corresponding pixel in the same order.
4. To replace one or both of the picks in an existing constraint, re-pick one or both 3D point and/or image pixel, then click the **Replace** toolbar button.

5. With enough enabled constraints, the **Compute** toolbar button becomes enabled. Push the **Compute** toolbar button to compute the texture mapping parameters.
- If the **Suggest Improvements** option in the **Texture Map Editor Options** dialog is set to “Yes”, after the computation is finished (success or failure), *Cyclone* searches for a subset of constraints that results in a lower overall error, if available. The process may be cancelled at any time. If any improvements were found, *Cyclone* displays the list of up to ten improvements.
- To visually verify the texture mapping results, select the texture map in the **Texture Map Browser** and click the **Align** toolbar button. The 3D viewpoint should change to closely match the image.

Displaying a Texture Map

Once a texture mapping has been computed, the texture map is considered “valid” and can be used to color the point cloud or mesh.

1. Enable texture mapping in the ModelSpace View by toggling ON the **View | Global Texture Map** command.
2. Select one point cloud or mesh.
3. In the **Texture Map Browser**, select the texture map and toggle its **Use** checkbox ON.
4. Use **Edit Color Map** or **Edit Global Color Map** to select the **Image Texture Map** mode. Alternately, select the **Edit Object | Appearance | Apply Color Map | Image Texture Map** command.
5. When a single cloud or mesh has multiple enabled texture maps, the texture maps may overlap. Where two texture maps overlap, the texture map that appears lower in the **Texture Map Browser**’s list will be covered by the other texture map. To change the order of the texture maps, click the **Raise** and **Lower** toolbar buttons in the **Texture Map Browser**.

Managing Texture Maps

Copy/Paste

Texture maps can be copy/pasted between objects. Creating a texture map for one object and then copying it to other objects can be an efficient way of propagating a texture map among different scans taken from the same position.

1. Select one or more point cloud or mesh with at least one texture map.
2. In the **Texture Map Browser**, select one or more texture map and click the **Copy** toolbar button. The texture maps to be copied do not need to be enabled or valid.
3. Select one or more point cloud or mesh.
 - Texture maps may be copy/pasted to the same object. This can be useful for keeping different texture mappings.
 - If multiple objects are selected, the **Texture Map Browser**’s list will show only the texture maps shared in common by the selected objects.
4. Click the **Paste** toolbar button. The texture maps are copied to the selected objects.

Replacing the Image

The image of a texture map can be replaced by another image. The replacement image may have different dimensions from the current image, but they must have the same proportions.

Uses of image replacement include: to have different images taken from the same position (so they can use the same texture mapping) but at different times (e.g., a day-time image versus a night-time image); to create the texture mapping using a lower-resolution “preview” image and then replacing the image with a higher-resolution version of the image.

1. Select one point cloud or mesh with at least one texture map.
 2. In the **Texture Map Browser**, select a texture map and click the **Replace** toolbar button. Follow the same process for adding a new image. The image is replaced in the texture map.
- When replacing the images for a cube-map, images are replaced by the corresponding Images in the selected Project, and any new cube-images are added to the cube-map. Any images without a corresponding Image in the selected Project are unaffected.

Saving the Texture Map

Loading texture maps for point clouds incurs some additional cost in the computer’s resources. Once the texture mapping is satisfactory/, the colors displayed by the **Colors from Scanner** color map mode can be replaced by the colors from texture mapping, and then texture mapping can be disabled.

1. Select one point cloud with a valid texture map.
 2. In the **Texture Map Browser**, select a valid texture map and click the **Save** toolbar button.
- Saving does not work for point clouds copied by reference from another database.
 - 3. Confirm the saving, then restart *Cyclone* when prompted.
- The saved texture map appears in the **Texture Map Browser** (saved texture maps always appear at the bottom of the list), but it can no longer be edited or enabled as a texture map. However, the saved texture map can be copy/pasted as a new texture map that can then be edited and activated (e.g., for updating the texture mapping and then re-saving it).

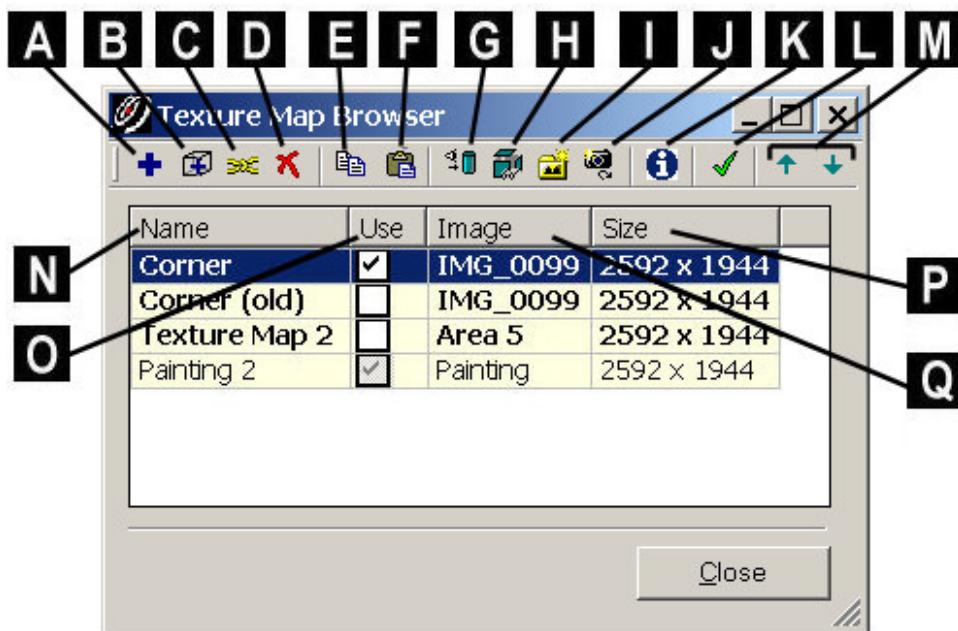
Texture Mapping Hints and Tips

- **Source images:** For best results, use images that are taken from positions and orientations close to the scanner position and orientation. This minimizes the effects of parallax, where the image contains elements that were not visible from the scanner.
- **When to create texture maps:** Create and save the texture mapping before any other modeling.
- **Computation #1:** Pick points that are well-distributed in space for better results. Avoid picking only constraints that are clustered together or points on a single plane.
- **Computation #2:** When creating constraints in a cube-map, try to pick at least one point in each image.
- **Computation #3:** Disabling or deleting the constraint with the highest error may not lead to a lower overall error.
- **Computation #4:** A lower overall error is not necessarily a better texture mapping. Always be sure to check the graphical results visually.
- **Computation #5:** When starting out, increase the **Texture Map: Tolerance** preferences. The actual values can depend on the resolution of the image as well as the quality desired.
- **Graphics Performance #1:** Disable any texture maps that are not used in the point cloud or mesh. For example, if a point cloud is segmented such that only one of two textures is relevant in the segmented cloud, turn off or delete the unused texture map. For cube-maps,

disable individual images via the **Enable/Disable Cube-Map Components** toolbar button in the **Texture Map Browser** dialog.

- **Graphics performance #2:** Meshes perform actual texture mapping, where the interior of each triangle can have multiple colors. Some graphics cards may not robustly support large texture maps. Try selecting a software-only OpenGL Mode for *Cyclone*.

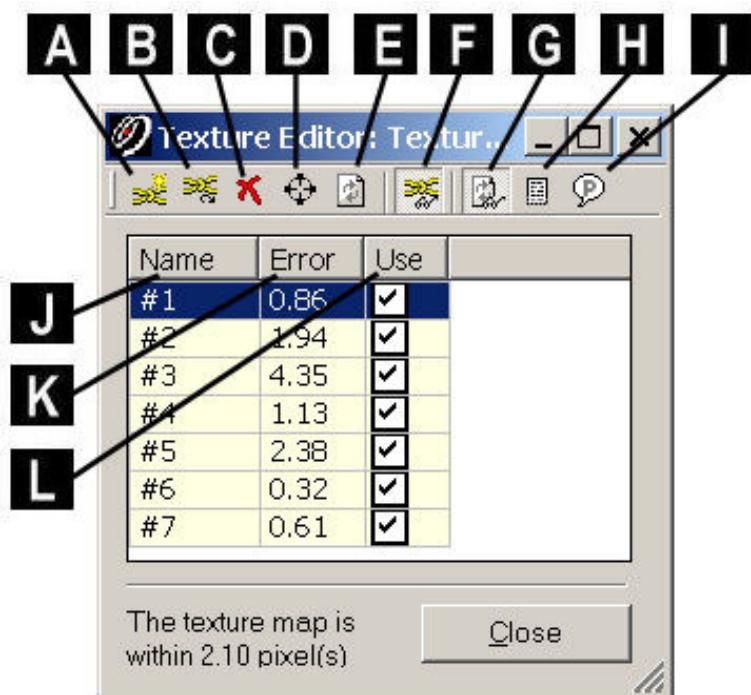
Texture Map Browser Dialog



- A. Click to select and add an Image in the database.
- B. Click to select and add a Project in the database containing cube-map Image(s).
- C. Click to edit the selected texture map in the **Texture Editor**.
- D. Click to delete the selected texture map(s).
- E. Click to copy the selected texture map(s) to the clipboard.
- F. Click to paste the texture map(s) on the clipboard to the selected object. If multiple point clouds and meshes are selected, the list area will be blank, but you will still be able to paste the copied texture maps to the selected objects. This is a convenient way of propagating texture maps.
- G. Click to align the viewpoint to the selected texture map.
- H. Click to enable/disable individual images within the selected cube-map.
- I. Click to create a Multi-Image from the selected cube-map texture maps. The Multi-Image can be used as a background when publishing a TruView data set.
- J. Click to replace the image for the selected texture map with another image in the database. The replacement image can have different dimensions, but it must have the same proportions as the current image.
- K. Click to display information about the selected texture map.

- L. Click to save the selected texture map(s) to the selected point cloud. This replaces the colors displayed by Colors from Scanner for each point in the point cloud. Points not in the current cloud are not (re)colored.
- M. Click to move the selected texture map higher or lower in the list. Texture maps higher in the list can be covered by texture maps lower in the list wherever they overlap.
- N. The name of the texture map. Click to rename it.
- O. The visibility of the texture map. If the texture map is visible and is valid, the object's color map is **Image Texture Map**, and **Global Texture Map** is enabled, then the object is drawn with colors from the enabled texture map.
- P. The name of the image used for the texture map.
- Q. The size of the image used for the texture map.

Texture Editor Dialog



- A. Click to add one constraint for each pick in the ModelSpace viewer and in the **Texture Editor (Image)** dialog, in sequence.
- B. Click to replace the selected constraint's 3D pick and/or image pick, depending on the existing pick(s).
- C. Click to delete the selected constraint(s).
- D. Click to focus the viewpoint on the selected constraint. For a cube-map, the corresponding image is loaded into the **Texture Editor (Image)** dialog.
- E. Click to compute the texture mapping parameters based on the current constraints.
- F. Click to toggle the graphical display of the constraints.
- G. Click to toggle the automatic refreshing of the texture map graphics in the ModelSpace viewer whenever the texture mapping changes. Toggle this OFF if you are making several computations, to avoid the overhead of refreshing the graphics each time.

- H. Click to access the **Texture Map Editor Options** dialog.
- I. Click to access the **Modeling** tab of the **Edit Preferences** dialog. The **Texture Map: Tolerance** preferences are available in the **Modeling** tab.
- J. The name of the constraint. Click to rename it.
- K. The difference (in pixels) between the location of the constraint's pixel vs where its 3D pick lands on the image for the texture mapping parameters.
- L. Click to toggle the use of the selected constraint(s) during the texture mapping computation. Constraints that are disabled do not contribute to the computation.

Texture Editor (Image) Dialog

The **Texture Editor (Image)** dialog accompanies the **Texture Editor** dialog.

1. Click in the image to zoom and pan to the pixel of interest, then release to select a pixel.
2. Right-click on the image for a context menu.
 - **Single-Pick:** Select this mode to have at most one pick at a time in the image.
 - **Multi-Pick:** Select this mode to have multiple picks at a time in the image. It can be more efficient to make multiple picks and then add them in a single click.
 - **Remove Last Pick:** This command discards the last pick made on the image.
 - **Clear Pick/Clear All Picks:** This command discards the picks made on the image.
 - **Set Zoom...:** This command invokes the **Zoom Factor** dialog to let the user enter the image magnification.
 - **Back, Down, Front, Left, Right, Up:** For a cube-map, select the image to activate in the image dialog.
 - **Cancel:** This command dismisses the context menu.

Texture Map Editor Options Dialog

The **Texture Map Editor Options** dialog provides access to several settings used by the **Texture Editor** dialogs. The settings are saved.

- **Constraint Color (Enabled):** Select the color used to display the enabled constraints.
- **Constraint Color (Disabled):** Select the color used to display the disabled constraints.
- **Callout Length:** Enter the length (in pixels) of the line between the graphical constraint text and the picked point.
- **Callout Angle:** Enter the angle of the line from the picked point to the graphical constraint text.
- **Font:** Enter the name of the font to use for the graphical constraint text.
- **Font Size:** Enter the size (in pixels) of the font to use for the graphical constraint text.
- **Suggest Improvements:** This setting determines if *Cyclone* attempts to find a set of constraints (from the enabled constraints) that results in a lower overall error. Disabled constraints are not used.
- **Image Render Mode:** Select “Software-only” if your graphics card has trouble drawing the image in the **Texture Editor (Image)** dialog.

Cyclone Object Exchange (COE)

COE for AutoCAD

Introduction

COEIN, an AutoCAD 2000 and higher ObjectARX application, was developed to enable the translation of objects from *Cyclone* to AutoCAD and vice versa. When used to import objects into AutoCAD, COEIN generates 3D representations of objects imported from a *Cyclone* COE file. Users can then interact with these objects using the full range of AutoCAD's modeling functions. COEIN also provides the means for exporting to a COE file from AutoCAD, with *Cyclone* as the destination application. Using COE to export AutoCAD modeled/edited objects allows you take advantage of AutoCAD's advanced modeling functionality, yet maintain a link to the original COE file created in *Cyclone*. Drawings from AutoCAD may also be imported into *Cyclone* for various uses, including interference checking, and visualization.

This document describes the main features of the COE application, along with any limitations. A troubleshooting section is included to help resolve common problems.

Requirements

- To use the COE import/export feature, you must be running AutoCAD 2000 or higher. For information on system requirements, consult the AutoCAD documentation.

Installation

Installation is done via the *Cyclone* installer. It is a one-time process that consists of loading the COEIN.ARX application, making the following commands available:

COEIN

This command is used to import COE files into AutoCAD.

COEOUT

This command is used to export drawing contents to a COE file.

COEXPLODE

This command is used to explode blocks, and is similar to the AutoCAD **Explode** command.

- The *Cyclone* installer adds a COE menu and toolbars to the default AutoCAD user profile. If you are using a different user profile, you can add the COE menu by loading the menu file CYCLONECOE.MNC using AutoCAD menu customization.

Usage

The COEIN feature allows you to use both *Cyclone* and AutoCAD to maximize your modeling capabilities. Not only can you translate objects from *Cyclone* to AutoCAD, but you can make changes to those objects and even model new objects, which can then be translated back to *Cyclone* and incorporated into the source database.

Translating Objects from *Cyclone* to AutoCAD

Objects modeled in a *Cyclone* environment can be exported to a COE file using the COE export feature. The COE import feature can then be used to import COE files into any active AutoCAD drawing. COE translates objects from the *Cyclone* environment by generating equivalent, or near equivalent, objects in the destination environment. The following sections describe the process for exporting objects via COE from *Cyclone*, and importing those objects via COE into AutoCAD.

Note: This ARX application is designed primarily to translate modeled 3D objects between applications. While it is possible to translate points via COE, we recommend using *CloudWorx for AutoCAD* when you need to bring *Cyclone* point clouds into AutoCAD.

Creating a COE File in *Cyclone*

To export modeled objects to a COE file from *Cyclone*:

1. Open the desired ModelSpace, and use *Cyclone*'s 3D modeling tools to create equipment, piping, vessels, or other objects.
- If you want to isolate selected objects for export to the COE file, multi-select the desired objects, then select **File | Launch | Copy Selection to New ModelSpace**.
2. When you are ready to export, click **File | Export....** The **Export to File** dialog appears.
3. Identify a location for the file, name the file to be created, then select “**COE format (*.coe)**” from the **Save as type** field.
4. Click **OK** to create the COE file. All objects within the *Cyclone* ModelSpace are translated to COE format. This step may take some time, depending on the complexity of the ModelSpace being translated and your computer's capabilities.

BRINGING A COE FILE INTO AUTOCAD

Once a drawing has been exported to COE format from *Cyclone*, it can be imported into AutoCAD using COEIN.

To import a COE file into AutoCAD:

1. From the menu bar, select ***Cyclone* | COE | Import COE Objects**, or type **COEIN** at the command line. The **Import Options** dialog appears.
2. Find and select the COE file to be imported into AutoCAD by clicking the **Browse** button below the **File Name** field.
3. Select whether to import objects as ACIS solids or as Blocks.

IMPORT OBJECTS AS ACIS SOLIDS

Select this option to bring in modeled objects as 3D ACIS solids.

IMPORT OBJECTS AS BLOCKS

Select this option to bring in modeled objects as 3D blocks. Use of this option results in the best compression rate as the objects are brought into the AutoCAD drawing. In addition, the blocks can be exploded once in AutoCAD.

Note: All Xdata added to the blocks by COEIN will remain intact once a block has been exploded.

4. Check **Import Pointsets** if point clouds are contained in the COE file.
 - This option is not recommended for large point clouds. Instead, we recommend using Leica Geosystems HDS's *CloudWorx* application for bringing Leica Geosystems HDS point clouds into AutoCAD.
5. Check **Create Log file to** generate a <drawing name>.LOG file (recommended). This log file can help you identify problems and potential solutions if the COEIN process is aborted.
 - If **Use Default File Path** is checked, the log file is saved to the default folder (the same folder as the current drawing). You can specify a destination for the log file by deselecting this option, and typing in or browsing to select the desired folder location for the log file.
6. Select the units used in the current drawing from the pull-down list in the **Import Drawing Units** field, or select "User defined" and enter a user-defined multiplier in the **Scale Factor** field to multiply all measurements, including coordinates, by this number.
 - By default, all COE objects are brought into AutoCAD from *Cyclone* using meters and radians to define size and position. You need to define the units to which COEIN will convert for your drawing by selecting the *current* drawing units from the pull-down list.
7. Click the **OK** button. The COE file is loaded into the current drawing. The complexity of the COE file, as well as your computer's capability, determines the speed with which the objects are brought in and displayed. You may click **Cancel** at any time during the process to abort the process.
 - If an error message is displayed at any time during the import, refer to the *Troubleshooting* section in this appendix.

Working with Imported Objects in AutoCAD

Once objects have been loaded into an AutoCAD drawing from a COE file, standard AutoCAD commands and functionality are fully available to view, edit, and manipulate the objects as needed. You can also model new objects for later export back to *Cyclone*.

In addition to AutoCAD's powerful capabilities, COEIN provides you with additional options which are discussed in the following sections.

EXPLODING OBJECTS

You can use the **COEEplode (coeExplode)** command or AutoCAD's **Explode** command to explode objects imported as blocks.

Note: All Xdata associated with the blocks by COEIN remains intact even when a block has been exploded by either command.

The major difference between the **coeExplode** command and the AutoCAD **Explode** command is that **coeExplode** actually deletes the exploded block and redraws it as an Acis Solid. Otherwise, the two commands behave similarly.

Note: The **coeExplode** command requires the presence of the original COE file that created the object.

FULL LAYER TRANSLATION AND FUNCTIONALITY

When you import a COE file, you can elect to import layer settings as defined in *Cyclone* into AutoCAD, along with any objects contained in the file. *Cyclone*-defined layers are easily and cleanly translated; COE not only uses the same naming protocol defined in *Cyclone*, but it also translates layer settings and object assignment.

Once in place, imported layers function as any other AutoCAD layer, and can be used to make your modeling session more productive.

It is important to note that AutoCAD does not accept certain characters in a layer name (< > / \ \ : ? * | , = ') If the import finds a layer name in a COE file with one of these characters in it, it removes the invalid character and replaces it with an underscore “_”. The changed layer name is recorded in the *.log file and you are informed via a popup message at the end of the import that some layer names had invalid characters.

Limitations

NO POINT CLOUDS

We do not recommend using COEIN to bring large point clouds into AutoCAD. If a Leica Geosystems HDS point cloud contains large quantities of points, a lot of memory is used, and an attempt to import a COE file containing a point cloud may lock up your system. Instead, we recommend that you use Leica Geosystems HDS's *CloudWorx* application to bring *Cyclone* point clouds into AutoCAD.

Translating Objects from AutoCAD to *Cyclone*

Objects modeled within the AutoCAD environment can be exported to a COE file using the COEOUT command for subsequent import into *Cyclone*. There are two primary reasons for translating objects from AutoCAD to *Cyclone*:

Simple export: Objects modeled in AutoCAD can be exported to a new COE file. When this file is imported into *Cyclone*, a new ModelSpace is created and populated with the AutoCAD-modeled objects contained in the COE file.

Data integration: Objects modeled in *Cyclone* and imported into AutoCAD via a COE file can be modified in AutoCAD and then exported back into *Cyclone*, all the while maintaining much of the original data (e.g., annotations) from the original COE file from *Cyclone*. To do this, CoeOut uses the original COE file as a reference when creating the COE file to be translated to *Cyclone*.

Creating a COE file in AutoCAD

1. Do your modeling as desired in AutoCAD.
2. From the menu bar, select ***Cyclone* | COE | Export COE Objects**, or type **cofout** at the command line. The **Export Options** dialog appears.
3. Enter a new file name and path in the **Export File Name** field, or use the **Browse** button to find and select a path.
4. Select an export process option from one of the two options listed:

EXPORT OBJECTS WITHOUT ORIGINAL COE REFERENCE FILE

Select this option if you are creating a new COE file and do not want any reference or connection made to the original COE file you brought into AutoCAD.

EXPORT OBJECTS USING ORIGINAL COE FILE AS REFERENCE

Select this option if you want to integrate data with the original COE file brought into AutoCAD. When this new COE file is created, references are made to the original COE file imported into AutoCAD, enabling COE to maintain as much original object information/data as possible when the object is brought back into *Cyclone*.

5. Define which objects to export by selecting one of the following options:

ALL OBJECTS

Select this option to export all objects on all layers that are contained within the current drawing.

VISIBLE OBJECTS

Select this option to export only those objects that belong to layers that are turned on (viewable) in the current view. Be sure the correct view is displayed before selecting this option.

SELECTED OBJECTS

This option is active only if there are objects currently selected in the drawing. If selected, only those objects currently selected in the drawing will be exported.

6. Click **OK** to begin exporting the selected objects to the COE FILE. The complexity of the objects, as well as your computer's capability, determine the speed with which the objects are exported.

Note: If an error message appears during the exportation process, refer to the [Troubleshooting](#) section in this appendix for information.

Bringing a COE File into *Cyclone*

COE files are brought into *Cyclone* via *Cyclone*'s Navigator window, where a new ModelSpace is created for the imported file. Most objects, as well as layer functionality, can be brought into *Cyclone* from AutoCAD. Complex objects (e.g., cubes with rounded corners, objects with notches, etc.) may not be translated as objects, but may instead be translated as a collection of lines and arcs.

To bring a COE file into *Cyclone*:

1. Open the *Cyclone Navigator* window and select the database or project into which you want to import the COE file.
2. Right click the Database/ModelSpace name and select **Import**, or select **File | Import** from the **Navigator** window. The **Import from File** dialog appears.
3. Locate the COE file to be imported and initiate the process. A new ModelSpace is created in the selected project.

Limitations

SHAPES

If there is an unsupported shape, it will be brought into *Cyclone* as a wireframe object, but ONLY if you check the "Unsupported objects" options in the COEOUT Options dialog in MicroStation; otherwise, the unsupported objects are not translated. Instead, a message is displayed informing you of that the objects have not been translated, and asking if you want to select the objects not translated.

MODELSPLANES

You cannot bring COE objects directly into an existing ModelSpace; however, you CAN multi-select and copy objects from the ModelSpace created from the COE import, and paste them to the desired ModelSpace. Alternatively, you can use the Merge from ModelSpace command to merge ModelSpace Views. See the *Merge from ModelSpace* entry in the *Commands* chapter for more information.

ANNOTATIONS

With the exception of tags, annotations and measurements made in AutoCAD are not translated to *Cyclone*.

Troubleshooting

SOME OBJECTS ARE NOT TRANSLATED OR ARE PARTIALLY TRANSLATED

It may be that the non-translated object is a type of complex 3D solid object that is not presently supported by the COE import/export tool. Try exploding the object before exporting to the COE file.

COEIN/COEOUT PROCESS ABORTS PREMATURELY

If you check the Create Log File option, open the Log file to find the reason for the premature abort. Knowing the reason may help you determine the appropriate resolution.

If you selected “Use Default File Path”, you will find the Log file in the folder containing the active AutoCAD drawing. It has the same name as the drawing, but with a *.LOG extension. Otherwise, the Log file will be in the folder you specified during the import/export.

COE FILE WILL NOT LOAD IN AUTOCAD; SYSTEM LOCKS UP

Does the COE file you are trying to import contain points? If so, try this:

1. Return to *Cyclone* and multi-select only the objects (this is easy if you use the [View Properties](#) dialog to turn off the “selectable” option for points).
2. Select all objects and copy them into a new ModelSpace by selecting File | Launch | Copy Selection to New ModelSpace.
3. Create a new COE file from this ModelSpace, and then import this new file into AutoCAD.

COEIN.ARX IS NOT LOADED WHEN AUTOCAD IS REBOOTED

Add the folder containing the file COEIN.ARX to the Support File Search Path in the AutoCAD Options dialog.

Note: For instructions on adding a folder to the Support File Search Path, refer to your AutoCAD documentation.

YOU ARE UNABLE TO LOAD COEIN.ARX FILE

Are the COEIN.ARX and CyraCoeOut.DLL files located in the same folder? Search for both files, and if they aren't both in the same folder, copy the CyraCoeOut.DLL file to the folder containing the COEIN.ARX folder, and try loading COEIN.ARX again.

You Are Unable to Import a COE file into *Cyclone*

COE files can only be imported into *Cyclone* via the Navigator window. If you were trying to import via a ModelSpace View, the COE option is not available. Retry your import from the *Cyclone* Navigator.

If you are in the *Cyclone* Navigator and you are not seeing the *.COE option in the Files of Type field, you cannot import a COE file into the selected Navigator level. COE files can be imported only at a Database or Project level.

Commands

Import Objects (coein)

Window: AutoCAD
Menu: *Cyclone* | COE
Action: Opens dialog
Usage: The Import Objects (coein) command loads the Import Options dialog, which is used to set parameters for importing COE files from *Cyclone* into AutoCAD.
Note: This command can be activated by selecting it from the ***Cyclone COE*** menu, by typing **coein** at the command line, or by clicking the **Import Objects** icon in the toolbar.

Explode Blocks (coeXplode)

Window: AutoCAD
Menu: *Cyclone* | COE
Action: Executes command
Usage: The Explode Blocks (coeXplode) command is used to explode objects imported as blocks from *Cyclone*.

- The **coeXplode** command is similar to the AutoCAD **Explode** command; however, the **coeXplode** command actually *deletes* the exploded block and redraws it as an Acis Solid.

Note: The **coeXplode** command requires the presence of the original COE file from which the object was imported.

Export Objects (coeout)

Window: AutoCAD
Menu: *Cyclone* | COE
Action: Opens dialog
Usage: The Export Objects (coeout) command opens the **Export Options** dialog, which is used to set parameters for exporting COE files from AutoCAD to *Cyclone*.

Note: This command can be activated by selecting it from the **Cyclone | COE** menu, by typing **coeout** at the command line, or by clicking the **Export Objects** icon in the toolbar.

COE for MicroStation

Introduction

The *Cyclone* Object Exchange (COE) file format is a neutral file format created for the specific purpose of interchanging objects between *Cyclone* and major CAD systems, including MicroStation.

COE files are accessible via two MicroStation command plug-ins: COEIN and COEOUT. This chapter addresses how to use these plug-ins in MicroStation, as well as how to create new COE files and read imported COE files in *Cyclone*. A troubleshooting section is also included to help you address some of the more common problems that may occur when working with COE files.

Requirements

To use the COE import/export feature, you must be running MicroStation SE or later.

Note: Although COEOUT functions in MicroStation SE, the translation of complex surface/solid type objects into *Cyclone* objects (e.g., extrusions or revolutions of complex shapes or complex strings) is more likely to succeed when using MicroStation/J or later.

The following files are required to run COEIN and COEOUT:

COEIN.MA

This MDL application is used to import a COE file into a DGN file.

COEOUT.MA

This MDL application is used to export a DGN file (or selected parts) to a COE file.

CYRACOE.DLL

This DLL file resides in the system directory.

COE.INI

This INI file is used to configure preference and mapping between *Cyclone* layers and MicroStation levels.

Usage

The COE feature allows you to use both *Cyclone* and MicroStation together to maximize your modeling capabilities. Not only can you translate objects from *Cyclone* to MicroStation, but you can also make changes to those objects and even model new objects, which can then be exported back to *Cyclone* and incorporated into the source database.

Translating Objects from *Cyclone* to MicroStation

Objects modeled in a *Cyclone* environment can be exported to MicroStation using the COEIN MDL application. COEIN can be used to import COE files into any active MicroStation DGN file that has been initialized using a 3D seed file. COEIN translates objects from the *Cyclone* environment by calculating equivalent, or near equivalent objects, in the destination environment. Some objects, however, may not be translated directly because a direct equivalent doesn't exist in MicroStation. In such cases, a group of shapes representing the unknown object may be used. For more information on how COE objects from *Cyclone* are translated into MicroStation entities, see the section entitled *How Objects Are Translated* in this appendix.

The following sections describe the processes both for exporting objects from *Cyclone*, as well as importing COE objects into MicroStation. Limitations to the COE functionality are also discussed, along with a troubleshooting section.

Note: This MDL application is designed primarily to translate modeled 3D objects between supported applications. While it is possible to translate point clouds via COE, we recommend using the Leica Geosystems HDS's CloudWorx application when you need to bring *Cyclone* point clouds into MicroStation.

Creating a COE File in *Cyclone*

To export modeled objects to a COE file from *Cyclone*:

1. Open the desired ModelSpace, and use *Cyclone*'s 3D modeling tools to create equipment, piping, vessels, or other objects.
- If you want to isolate selected objects for export to the COE file, multi-select the desired objects, then select **File |Launch | Copy Selection to New ModelSpace**.
2. When you are ready to export, click **File | Export....** The **Export to File** dialog appears.
3. Identify a location for the file, name the file to be created, then select "**COE format (*.coe)**" from the **Save as type** field.
4. Click **OK** to create the COE file. All objects within the *Cyclone* ModelSpace are translated to COE format. This step may take some time, depending on the complexity of the ModelSpace being translated.

Note: When you import the COE file into MicroStation, you have the option of importing point clouds. However, be aware that bringing point clouds into MicroStation using COE can be a memory-intensive process, both when importing, and when attempting to work with the imported points. In addition, because points consume so much memory, COE will only bring in the first 500,000 points. For these reasons, we recommend using the *CloudWorx* application to view *Cyclone* point clouds in MicroStation.

Bringing a COE File into MicroStation

Once a ModelSpace has been exported to COE format from *Cyclone*, it can be imported into MicroStation using the COEIN MDL application.

To import a COE file into MicroStation:

1. Open a new or existing MicroStation drawing created from a 3D seed file.
2. If you do not currently have a command line interface available, select **Key-in** from the **Utility** menu to display the **Key-in** dialog.
3. Enter the command **MDL LOAD COEIN** at the command line.
4. From the file dialog, select the COE file to be imported. The **COEIN Options** dialog appears.
5. Select your desired options, which are described as follows.
6. Choose your desired options and click **OK** to continue (or **Cancel** to abort). The **Select World Coordinate System** dialog appears.
 - Notice that any user coordinate systems (UCS) you mapped while in *Cyclone* are listed here; if you wish to use one of these *Cyclone*-defined coordinate systems as your world coordinate system (WCS) in this DGN file, select the desired coordinate system from the list, then click **=>WCS** to map the selected user coordinate system to the world coordinate system, making them one and the same. All objects are aligned accordingly.
 - If you do not wish to use a UCS in this DGN file, click the **Default** button to map the COE data to the current WCS in MicroStation.

Note: Whether selected or not, all user coordinate systems contained within the COE file are imported into the DGN file as “auxiliary coordinate systems (ACS).”

- Once the coordinate system has been set, the COE file will be loaded into the current DGN file. The complexity of the COE file, as well as your computer’s capability, determine the speed with which the objects are brought in and displayed.
- If an error message or a standard output window is displayed indicating a system fault, refer to the *Troubleshooting* section in this document.

Mapping COE Layers to MicroStation Levels

In *Cyclone*, you have access to an unlimited number of layers for use in organizing your drawings. When you export objects to a COE file, these layers are saved to the COE file as well, and when the COE file is brought into MicroStation, these layers are mapped to MicroStation “levels.” However, MicroStation only supports 63 levels, which are named by default numerically (1 to 63), with level 62 reserved by COEIN for auxiliary objects and piping/extrusion centerlines, and level 63 reserved as the “default.”

If you used numbering to identify your *Cyclone* layers, COEIN can intelligently map the defined layers to corresponding MicroStation levels (e.g., COE layer 22 is mapped to MicroStation level 22). Layers with names that do not conform to the numbering scheme are mapped incrementally to available MicroStation levels (e.g., “Walls” layer is mapped to the next available MicroStation level; for example, level 12).

If the number of COE layers exceeds the number of available MicroStation levels, COEIN assigns excess COE layers to MicroStation level 63 (default), unless you have configured the mapping table to do otherwise.

Note: When mapping layers to levels, COEIN gives first priority to instructions given in the mapping table.

To customize the mapping table:

Note: You should edit the mapping table **before** running COEIN.

1. Use a file browser to locate the COE.INI file; open this file by double-clicking. Scroll down to find the mapping table, which is defined in the **[CXFIN LEVEL MAP]** section.

Warning! DO NOT modify the contents of the **[GENERAL]** section!

2. Enter your customized mapping directives by typing the name of the COE layer, followed by a “=”, followed by the MicroStation level number. See the example below.

```
[CXFIN LEVEL MAP]
```

```
Default=63
```

```
Piping=2
```

```
Walls=4
```

- You may enter as many lines as necessary to map all the levels. You can map multiple layers to the same MicroStation level if desired
 - If you don't want to use the mapping table, you can comment the section name out, or delete the entire **[CXFIN LEVEL MAP]** section from the coe.ini file.
3. Once you have set the layer-level mapping as desired, save the COE.INI file. Run COEIN as described in this document.

Limitations

SUPPORTED EQUIPMENT

Not all objects modeled in *Cyclone* are available in MicroStation. Where there is not an exact object match, the objects are represented by geometrical shapes (or groups of shapes) approximating the translated object. For more information on how COE objects from *Cyclone* are translated into MicroStation entities, see the section entitled *How Objects Are Translated* in this document.

POINT CLOUDS

MicroStation uses a zero-length line to represent a point, and COEIN is capable of translating points from *Cyclone* (via COE) into such points. However, point clouds brought into MicroStation in this manner are limited to the first 500,000 points, as performance and DGN file size are impacted significantly. Unless you have a compelling reason, we recommend that you not use COEIN to import points into MicroStation.

Note: An alternative method for bringing *Cyclone* point clouds into MicroStation is to use Leica Geosystems HDS's CloudWorx application.

Translating Objects from MicroStation to *Cyclone*

Objects modeled within the MicroStation environment can be exported using the COEOUT command to a COE file for subsequent import into *Cyclone*. There are two primary reasons for translating objects from MicroStation to *Cyclone*:

Simple export: Objects modeled in MicroStation can be exported to a new COE file. When this file is imported into *Cyclone*, a new ModelSpace is created and populated with the MicroStation-modeled objects contained in the COE file.

Data integration: Objects modeled in *Cyclone* and imported into MicroStation via a COE file can be modified in MicroStation, and then exported back into *Cyclone*, all the while maintaining much of the original data (e.g., annotations) from the original COE file from *Cyclone*. To do this, COEOUT uses the original COE file as a reference when creating the COE file to be translated to *Cyclone*.

Note: Although COEOUT functions in both MicroStation/J and Micro-Station SE, the translation of complex surface/solid type objects into *Cyclone* objects (e.g., extrusions or revolutions of complex shapes or complex strings) is more likely to succeed when using MicroStation/J.

MicroStation-modeled objects are translated by mapping to equivalent or near equivalent objects within *Cyclone*. However, some objects may not have equivalent or near equivalent objects in the destination environment; in these cases, the objects are represented by geometrical shapes (or groups of shapes) approximating the translated object. For more information on how COE objects from *Cyclone* are translated into MicroStation entities, see the section entitled *How Objects Are Translated* in this appendix.

The following sections describe the process for exporting objects from MicroStation, as well as importing COE objects into *Cyclone*. Limitations to the functionality are included as well, along with a troubleshooting section.

Note: This MDL application is designed primarily to translate modeled 3D objects between *Cyclone* and MicroStation. While points can be translated from *Cyclone* to MicroStation, both your DGN file size and your system performance will be greatly impacted. As an alternative, we recommend using Leica Geosystems HDS's CloudWorx application to view point clouds in MicroStation.

Creating a COE file from MicroStation

Note: You must be in a 3D environment (i.e., an active DGN file initialized using a 3D seed file) to run this command.

To perform an export to a COE file:

1. Do your modeling as desired in MicroStation, using a 3D seed DGN file.
 - If you plan to export the entire DGN file contents, make sure all objects are deselected before proceeding. If you plan to export only objects that are visible (e.g., objects in levels that are turned on and viewable), make sure you are in the correct view before proceeding. If you plan

to export only selected objects, make sure those selections are made now, taking care to select only what you intend to export.

- If you do not currently have a command line interface available, select **Key-in** from the **Utility** menu to display the **Key-in** dialog.
2. Enter the command **MDL LOAD COEOUT** at the command line.
 3. In the file dialog, enter a unique file name in the **Files** field, select the destination folder, and click **OK**. The **COEOUT Options** dialog appears.
 4. Choose your desired options and click **OK** to continue (or **Cancel** to abort). The selected objects are exported to the COE FILE.
- The complexity of the DGN file, as well as your computer's capability, will determine the speed with which the objects are exported.
 - If an error message is displayed indicating a system fault, refer to the *Troubleshooting* section in this appendix.

Mapping MicroStation Levels to COE Layers

MicroStation supports as many as 63 levels, with level 63 reserved as the "default" layer. When you export a DGN FILE to a COE FILE, these levels are exported as well, and COEOUT intelligently maps each level containing objects to a *Cyclone* layer, naming the *Cyclone* layer based on the level number (or level name if you used a level name file *.LVL).

However, as with COEIN, you also have the option of customizing the level/layer mapping process by editing the mapping table in the COE.INI file.

To customize the mapping table:

Note: You should edit the mapping table **before** running COEOUT.

1. Use a file browser to locate the COE.INI file; open this file by double-clicking. Scroll down to find the mapping table, which is defined in the **[CXFOUT LEVEL MAP]** section.

Warning! DO NOT modify the contents of the **[GENERAL]** section!

2. Enter your customized mapping directives by typing the number (or name) of the MicroStation level, followed by a "=", followed by the *Cyclone* layer name. See the example below.

```
[cxfout level map]
```

```
63=Default
```

```
2=Piping
```

```
4=Walls
```

- You may enter as many lines as necessary to map all the levels. You can map multiple levels to the same *Cyclone* layer.
- If you don't want to use the mapping table, you can comment the section name out, or delete the entire **[CXFOUT LEVEL MAP]** section from the COE.INI file.

3. Once you have set the layer-level mapping as desired, save the COE.INI file. Run COEOUT as described in this document.

Importing a COE File into *Cyclone*

To import a COE file into *Cyclone*:

1. Open the ***Cyclone Navigator*** and select the database or project into which you want to import the COE file.
2. Right click the database or project name and select **Import**, or select **File | Import** from the ***Navigator*** window. The **Import from File** dialog appears.
3. Locate the COE file to be imported and initiate the process. A new ModelSpace is created in the selected database or project.

Limitations

SHAPES

If there is an unsupported shape, it will be brought into *Cyclone* as a wireframe object, but ONLY if you check the “Unsupported objects” options in the COEOUT Options dialog in MicroStation; otherwise, the unsupported objects are not translated. Instead, a message is displayed informing you of that the objects have not been translated, and asking if you want to select the objects not translated.

MODELSPLANES

You cannot bring COE objects directly into an existing ModelSpace; however, you CAN multi-select and copy objects from the ModelSpace created from the COE import, and paste them to the desired ModelSpace. Alternatively, you can use the Merge from ModelSpace command to merge ModelSpace Views. See the *Merge from ModelSpace* entry in the *Commands* chapter for more information.

ANNOTATIONS

With the exception of tags, annotations and measurements made in MicroStation are not translated to *Cyclone*.

Troubleshooting

COEIN/COEOUT Does Not Finish And an Error Is Displayed

If you checked the Create LOG option in the COEIN/COEOUT Options dialog (recommended), open the LOG file to check the reason. You will find the LOG file in the same folder as the active DGN file; it has the same file name as the DGN file, but with a *.LOG extension.

SOME OBJECTS ARE NOT TRANSLATED TO THE DESTINATION ENVIRONMENT

If you are translating objects from *Cyclone* to MicroStation, it may be that the objects are not supported by COE, and they may have not have been translated. If you checked the Create LOG option in the COEIN/COEOUT Options dialog (recommended), you can check the LOG file to confirm that this is the cause.

If you are translating objects from MicroStation to *Cyclone*, double the check the following:

- Did you select the correct **Export Objects** option (All, Visible, Selected) in the COEOUT Options dialog?
- If you intended to export the entire DGN file contents, did you deselect all objects before proceeding?
- If you intended to export only visible objects, did you have the correct levels turned on?
- If you intended to export only selected objects, were all the desired objects selected?
- Did you check the **Unsupported Objects** (Smartsolid use wire, Unknown surface use wire) options in the **COEOUT Options** dialog?

THE APPEARANCE OF OBJECTS (COLOR) IS NOT AS DEFINED IN THE SOURCE ENVIRONMENT

Objects translated via COE files are managed in layers. Each layer has its own appearance override (assigned color), which is imported as “level symbology” in MicroStation.

OBJECT PRECISION IS INADEQUATE

Check the resolution parameters in the Design File Settings dialog (Settings | Design File...). If the resolution parameters are set too low, you will have decreased drawing precision, though you will have more design space. Adjust these parameters to achieve the desired results.

COE CONTENTS DO NOT FIT INTO THE DGN DESIGN SPACE

Check the resolution parameters in the Design File Settings dialog (Settings | Design File..., then select the Working Units category). If the resolution parameters are set too high or the DGN units are set improperly (e.g., DGN units are set to **mm** for a model that has a large area more appropriately measured in meters), you will have a decreased design space, though you will have more object precision. Adjust these parameters to achieve the desired results.

How Objects Are Translated

In most translations between *Cyclone* and MicroStation, there are elements in one system that have no exact equivalent in the other. The following tables show how COEIN and COEOUT translate objects.

From Cyclone to MicroStation

Cyclone	MicroStation Object Type
Vertex	03:Line 17:Text (<i>when object contains annotation "Text" or "Note"</i>)
Line Segment	03:Line
Circular Arc	15:Ellipse (when curve is closed) 16:Arc
Elliptical Arc	15:Ellipse (when curve is closed) 16:Arc
Polyline	04:Line String
Steel Shapes	04:Line String User Data Linkage: "StdShape"
Spline Curve	27:B-spline curve
Patch	06:Shape
Extruded Patch	18:Surface (when it is not fully capped) 19:Solid (when both side flat capped) User Data Linkage:"PlanarPolylineExtrusion"
Box	See PlanarPolylineExtrusion User Data Linkage: "RectangularBox"
Steel Section	See PlanarPolylineExtrusion User Data Linkage: "Steel" User Data Linkage:{ section type,d,bf,tf,tw}
Cylinder	02:Cell Header (Cone+Caps when using cells) 23:Cone (for cone)

18:Surface(for semi-elliptic, hemispheric caps)
 15:Ellipse (for flat caps)

Cone *See Cylinder*

Flange *See Cylinder*
 User Data Linkage: "Flange"

Reducer *See Cylinder*
 User Data Linkage: "Reducer"

Sphere 18:Surface
 User Data Linkage: "Sphere"

Torus 02:Cell Header (Solid+Caps when using cells)
 19:Solid (for torus)
 18:Surface (for semi-elliptic, hemispheric caps)
 15:Ellipse (for flat caps)
 User Data Linkage: "Torus"

Elbow, Reducing Elbow 02:Cell Header (Solid+Caps when using cells)
 19:Solid or 18:Surface (for torus)
 18:Surface (for semi-elliptic, hemispheric caps)
 15:Ellipse (for flat caps)
 User Data Linkage: "Torus"

Valve 02:Cell Header (Cone+Line+Ellipse when
 using cells)
 23:Cone (for valve admission and exhaustion)
 03:Line (for stem)
 15:Ellipse (for handle wheel)
 User Data Linkage to 1st Cone: Valve
 parameters

Group 02:Cell Header

Contours	02: Cell Header (with LineString and Text) User Data Linkage: "ContourObj"
Mesh	02: Cell Header (w/Shapes, limit to 800 triangles) User Data Linkage: "TriMesh" (V3.1 feature)
ExclusionVolume	02:Cell Header User Data Linkage: "ExclusionVolume", User Data Linkage: { size.x, size.y, size.z}
Layer	Level
Point Cloud	03:Line (individual zero-length lines)
Annotation (attached to object)	37:Tag
User Coordinate Systems	05: level 3 object Aux_coordinate

From MicroStation to *Cyclone*

Using COEOUT to translate objects to *Cyclone* is not as straightforward as the translation from *Cyclone* to MicroStation, largely because MicroStation's advanced CAD capabilities allow users to create any number of unique objects which do not exist in *Cyclone*, and thus, cannot be translated by COEOUT.

However, COEOUT can translate most standard objects to *Cyclone*. In addition, objects that were originally translated from *Cyclone* using COEIN are fully reversible whether the original COE file is referenced or not. If the original COE file is referenced, all non-geometric parameters and *Cyclone* reserved parameters are kept intact.

The following table shows the translation protocol that COE uses when using COEOUT to translate objects back to *Cyclone* from MicroStation. The “Mapped” column indicates whether the listed item can be mapped, and to what extent by using the following codes:

Y = fully mapped

P = mapping is possible but some complex objects are not supported

F = implementation may change in a future release

MicroStation Entity	Mapped Cyclone Object
02: Cell Header	P COE tries to fit these into the following Leica HDS objects: - Valve (only with Leica HDS label) Cylinder with caps (only with Leica HDS label) Flange with caps (only with Leica HDS label) Cone with caps (only with Leica HDS label) Reducer with caps (only with Leica HDS label) Elbow , Reducing Elbow with caps (only with Leica HDS label) Group (other non-name Cell Headers) Mesh (only with Leica HDS label) Contours (only with Leica HDS label) Not supported: Smart Geometry SMSLD, SMSURF Not supported (other named cell headers)
03: Line	Y - Vertex (when line length is zero) - Line Segment
04: Line String	Y - Polyline for non-planar Polyline - Planar Polyline (only with Leica HDS label "StdShape")
05: Group Data	- User Coordinate Systems: Lvl 3 obj. Aux_Coordinate
06: Shape	Y - Patch
07: Text Node	F - Group of Vertices (with Annotation Text)
12: Complex Line String	F - Group of Polylines and Circular Arcs
14: Complex Shape	F - Group of Polylines and Circular Arcs
15: Ellipse	Y - Circular Arc for circle - Elliptical Arc for general ellipse

16: Arc	Y	- Circular Arc for circular arc - Elliptical Arc for general elliptic arc
17: Text	Y	- Vertex (with Annotation "Text")
18: Surface (Complex)	P	COE tries to convert these to these Leica HDS objects: <ul style="list-style-type: none">- Extruded Patch (always capped)- Steel Section (only with Leica HDS label)- Sphere- Torus- Elbow, Mitered Elbow, Reducing Elbow- Cone, Cylinder, Reducer- Revolved complex shapes as group of connected shapes- Cones/Cylinders, Extruded complex shape as an Extruded Patch
19: Solid (Complex)	P	COE tries to fit these into the following Leica HDS objects: <ul style="list-style-type: none">- Extruded Patch (always capped)- Steel Section (only with Leica HDS label)- Sphere- Torus- Capped Elbow or Mitered Elbow or Reducing Elbow- Capped Cone, Cylinder, Reducer- Revolved complex shapes as a group of connected shapes- Cones/Cylinders, Extruded complex shapes use Extruded Patch
23: Cone	Y	- Cylinder (capped or uncapped) - Cone (capped or uncapped) - Flange (only with Leica HDS label) (capped or uncapped) - Reducer (capped or uncapped)

27: B-spline Curve	P	Spline Curve
37: Tag	P	- Find associated element and add as an annotation
Levels	Y	<ul style="list-style-type: none">- Layers- Reads mapping tables from COE..INI- Current level mapped to Default Layer- Other levels (with objects) to named external layers

OpenGL Modes

OpenGL Modes Introduction

The **OpenGL Modes** applet (installed with *Cyclone*) displays the available graphics modes supported by the graphics system of your computer. It is intended for use with *Cyclone*, to help the user choose an alternate graphics mode. Each graphics mode has its own particular characteristics. The relevant characteristics are described below.

Sometimes, the graphics system does not fully support a particular mode, or *Cyclone* may have odd interactions with a given mode. In the case that *Cyclone* chooses one of those problematic modes, the user has the option of reconfiguring *Cyclone* to use another graphics mode.

Note: A particular graphics card/drive combination may have additional settings that affect *Cyclone*'s performance.

To set a new graphics mode:

1. Launch the **OpenGL Modes** applet.
 - To launch the **OpenGL Modes** applet: From the Windows **Start** menu, point to **Programs | Leica HDS Cyclone 5.8 | Utilities** and then select **OpenGL Modes**. The **Supported OpenGL Modes** dialog appears.
2. Select a new graphics mode.
 - The default graphics mode chosen by *Cyclone* has an asterisk next to it, and the user-selected graphics mode is enclosed in brackets. To reset the mode to the user default, click the **User Default** button. This gives *Cyclone* the option of choosing the mode to use.
 - To select a new graphics mode, click a number in the **Mode** column, and then click the **Set Mode** button. The new mode is written to your registry.
 - The **Check Screen Saver** checkbox controls *Cyclone*'s redrawing while a screen saver is active. When enabled, *Cyclone* does not redraw when the screen saver is active - this can reduce load on the CPU over time (e.g., when scanning). If *Cyclone* does not render properly, turn this option **OFF** and then restart *Cyclone*.
3. Click **Exit**, and then restart *Cyclone* to switch to the new mode.
 - If you change your graphics card or graphics driver, please be sure to test whether or not you still need that graphics mode, or if in fact that mode still exists with the desired characteristics.

Graphics mode characteristics

Please note that some graphics modes may seem identical as presented by this program, when in fact, there are differences not listed.

MODE

The number of the graphics mode.

RGB

This setting determines the number of color bits used in the graphics mode when rendering objects in color. At least 15 bits is recommended for good color differentiation.

2 bits = black and white (not recommended)

4 bits = 16 colors (not recommended)

8 bits = 256 colors

15 bits = 32,768 colors

16 bits = 65,536 colors

24 bits = 16,777,216 colors

32 bits = 4,294,967,296 colors

The difference between 24 bits and 32 bits for a modern display monitor is insignificant, but can have a positive or negative effect on memory layout and memory usage.

ZBUF

This setting determines the number of bits used for the depth buffer, or "Z-buffer", which is a commonly used way of rendering in 3D so that objects closer to the viewer appear "on top" of objects further away. At least 16 bits is recommended for avoiding some rendering artifacts. 24 bits is better, etc.. On some cards, the Z-buffer depth is limited by the number of RGB bits, so you may need to change the display settings to access a deeper Z-buffer.

DBLBUF

This setting determines whether or not the graphics mode supports double-buffering, which lets the system draw sequential frames without flickering.

Y = sequential frames will not flicker (recommended)

(blank) = sequential frames may obviously flicker

SW

This setting determines whether or not the graphics mode supports software rendering, where the main CPU in the computer does all of the rendering, which can be more correct than hardware rendering (depending on the graphics driver), but at the expense of rendering performance. If none of the other graphics modes work correctly, then try one of these.

Y = rendering is done entirely in software

(blank) = some rendering tasks may be done by the graphics hardware

HW

This setting determines whether or not the graphics mode supports hardware acceleration, where the graphics card does some or all rendering tasks, which can be faster than software rendering.

Y = supports rendering entirely in the graphics hardware

(blank) = some rendering tasks may be done by the graphics hardware

OPENGL

This setting determines whether or not the graphics mode supports the OpenGL API. This must be set to **Y**.

Command Reference

2D Drawing Mode

Window: ModelSpace
Menu: Edit | Modes
Action: Executes command
Usage: The **2D Drawing Mode** command enters **Drawing** mode and sets the drawing tool to draw the shape selected in the **Drawing** submenu in the **Tools** menu.

For 2D drawing procedures, see the [Drawing](#) entry in the *Modeling* chapter.

2D Extraction

Window: ModelSpace
Menu: Tools
Action: Displays submenu
Usage: Pointing to **2D Extraction** displays submenu commands used to create 2D drawings from the contents of a rectangular data fence.

The following commands are available from the **2D Extraction** submenu:

[From Fence Contents...](#), [From Selection in Fence...](#)

- For **2D Extraction** submenu command and dialog details, see the individual command entries.

About Cyclone

Window: All Windows
Menu: Help
Action: Opens dialog
Usage: The **About Cyclone** dialog displays a splash screen containing the version number of the *Cyclone* software.

- To exit the **About Cyclone** dialog, click **OK**.

Above Ref Plane (Mesh)

Window: ModelSpace

Menu: Tools | Measure | Mesh Volume

Action: Opens dialog

Usage: The **Above Ref Plane** command is used to measure the “cut” volumes of a selected mesh relative to the active Reference Plane.

- To measure the “fill” volume of a mesh, see the [*Below Ref Plane \(Mesh\)*](#) command for more information.

To measure the “cut” volume of a mesh down to the active Reference Plane:

1. Select the mesh whose “cut” volume you want to measure.
 - The cloud must be viewed as a mesh. (See the [*View Object As*](#) entry in this chapter.)
2. From the **Tools** menu, point to **Measure | Mesh Volume**, and then select **Above Ref Plane....** The **Sampling Step** dialog appears.
3. Enter a value for the interval by which the Reference Plane is divided into a grid for measurement.
 - A smaller interval may yield more accurate results, but may take longer to compute.
4. Click **OK**. The measurement results are displayed in cubic units. The measurement result is also displayed in the Output Box, if it is enabled.

Acquire Targets

Window: Scan Control

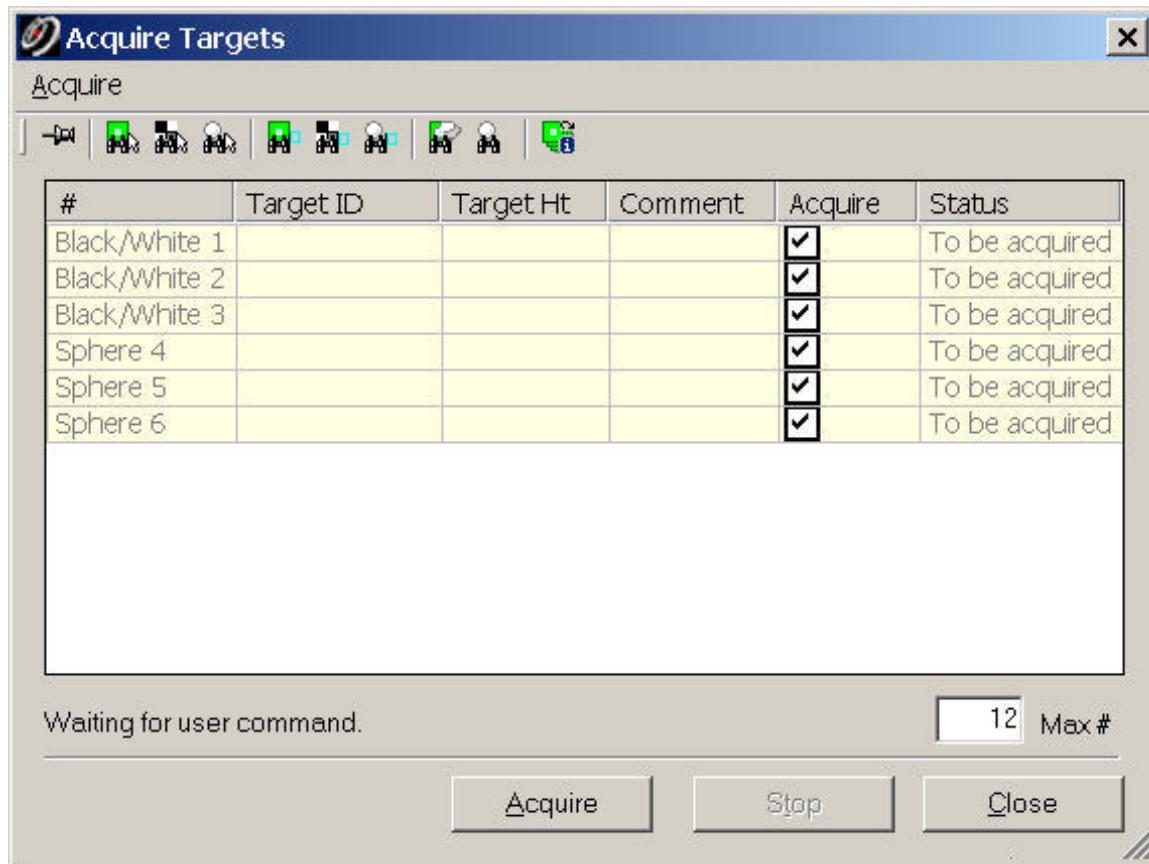
Menu: Scanner Control

Action: Opens dialog

Usage: The **Acquire Targets** command displays a dialog that is used to acquire HDS Targets, 6" diameter spherical targets or black/white targets near user-controlled picks in the current point cloud, as well as to acquire HDS Targets or 6" diameter spherical targets from selected point clouds.

- Once acquired, HDS targets and sphere targets are automatically added to the ControlSpace.

Acquire Targets Dialog



- Acquire** Selects one of the following target acquisition options: **HDS Targets from Pick Points**, **Spheres from Pick Points**, **Black/White Targets from Pick Points**, **HDS Target from Fence**, **Black/White Target from Fence**, **Sphere from Fence**, **HDS Targets from Selected Cloud**, **Spheres from Selected Cloud**, or **Recheck Targets**.
- ☛ Click the **Keep on Top** icon to display this dialog on top of other open windows.
 - ☛ Click the **Acquire HDS Targets from Picks** icon to find potential HDS target candidates near user-picked points.
 - ☛ Click the **Acquire Black/White Targets from Picks** icon to find potential black/white target candidates near user-picked points.
 - ☛ Click the **Acquire Spheres from Picks** icon to find potential sphere target candidates near user-picked points.
 - ☛ Click the **Acquire HDS Target from Fence** icon to find a HDS target within the user-defined fence. The fence should be as small as possible (no larger than the actual target), centered on the target.
 - ☛ Click the **Acquire Black/White Target from Fence** icon to find a black/white target within the user-defined fence. The fence should be as small as possible (no larger than the actual target), centered on the target.
 - ☛ Click the **Acquire Sphere from Fence** icon to find a sphere target within the user-defined fence. The fence should be as small as possible (no larger than the actual target), centered on the target.

-  Click the **Search HDS Targets from Selected Cloud** icon to find potential HDS Target candidates in a selected point cloud. You are only required to select a point cloud before activating this command; you are not required to pick near potential targets. *Cyclone* locates and lists up to a maximum number of potential HDS Target targets in the point cloud (with the maximum number defined in the **Max #** field in this dialog).
-  Click the **Search Spheres from Selected Cloud** icon to find potential sphere target candidates in a selected point cloud. You are only required to select a point cloud before activating this command; you are not required to pick near potential targets. *Cyclone* locates and lists up to a maximum number of potential sphere targets in the point cloud (with the maximum number defined in the **Max #** field in this dialog).
- Note:** The number of target candidates listed when you invoke either **Search HDS Targets from Selected Cloud** or **Search Spheres from Selected Cloud**, is independent of the number of potential targets already listed in this dialog.
-  Click the **Recheck Targets** icon to automatically re-acquire the checked targets and display a report of the deviations from the previous results. In the dialog that appears, specify the **Recheck Tolerance** (the maximum distance that the target position is allowed to have changed) and **Target Prefix** (prepended to the names of the re-acquired targets). This can be used to detect if the scanner or targets have unknowingly changed location over some period of time. The rechecked targets are placed on the special **Target QA/QC** layer for reference.
- # The **Target Number (#)** column displays target candidates that have been identified either by the user or by *Cyclone* (using one of the icon commands).
- Target ID** Type a unique registration label for each potential target.
- Target Ht** Type the height for each potential target.
- Comment** Type notes or commentary about the potential target. When this field contains data for a target that is subsequently acquired, the data becomes a text annotation.
- Acquire** Select this check box to mark the target for attempted acquisition. If cleared, *Cyclone* ignores the potential target during target acquisition.
- Status** Displays the ongoing status of target acquisition for each target, ending with the final result of the attempted acquisition.
- Max #** Type the maximum number of target candidates to be found by *Cyclone* when either the **Search HDS Targets from Selected Cloud** or **Search Spheres from Selected Cloud** commands are selected.
- Acquire** Begins the acquisition process of selected targets.
- Stop** Stops the acquisition process of selected targets.

To acquire targets from picks:

- The process of acquiring targets is usually done after an initial scan of the scene, which provides a general reference as to the location of targets in the scene.
1. From the **Project** menu in the **Scan Control** window, select **Open ModelSpace Viewer** (or click the **Open Viewer** button on the right side of the **Scan Control** window). A new ModelSpace viewer for the incoming scan data opens.
In order to make targets stand out from the scene, make sure color mapping is turned **ON**. (See the *View Object As* entry in this chapter).
 2. In the ModelSpace window, multi-select the target locations. Each pick should be within two

inches of the actual location. If you have both HDS Targets and spheres in your scan, select all the potential HDS Targets first and proceed through step 4, then select all the sphere targets and repeat steps 2 - 4.

3. From the **Scanner Control** menu in the **Scan Control** window, select **Acquire Targets**. The **Acquire Targets** dialog appears.
4. From the **Acquire Targets** menu in the dialog, select **HDS Targets from Pick Points** to mark potential HDS Targets in the ModelSpace viewer, and list them in the **Acquire Targets** dialog; select **Black/White Targets from Pick Points** to mark potential black/white targets in the ModelSpace viewer, and list them in the **Acquire Targets** dialog; select **Spheres from Pick Points** to mark potential sphere targets in the ModelSpace viewer, and list them in the **Acquire Targets** dialog.
5. View each potential target to determine which ones you want *Cyclone* to attempt to acquire.
 - To center the ModelSpace viewpoint on a potential target, select the potential target from the list in the **Acquire Targets** dialog.
 - To rotate the ModelSpace viewpoint around a potential target, select the potential target from the list in the **Acquire Targets** dialog, and then press the **LEFT/RIGHT ARROW** keys.
 - To move up/down the list of potential targets, press the **UP/DOWN ARROW** keys.
6. Add an identifier or comment to potential targets to be acquired.
 - To add an identifier or comment: select the target, click the **ID** or **Comment** field, type an identifier or comment, and press **ENTER**.
 - Identifiers function as registration labels and can be used for automatic registration.
 - Comments function as text annotations in the ModelSpace once a target is acquired.
7. Once you have viewed potential targets, select the targets to be acquired by checking the box in the fourth column for each target to be acquired.
 - To toggle the check box for a selected target, press the **SPACEBAR** key.
8. Click **Acquire**. The target acquisition process begins; progress and results are displayed in the **Status** column.
 - During the target acquisition process, *Cyclone* performs a coarse and then a fine scan on each potential target, and then fits a vertex to the target. If the coarse scan fails to find sufficient data to begin a fine scan, the **Status** column in the **Acquire Targets** dialog displays *Failed at Coarse*. If the fine scan fails to find sufficient data to place a target, the **Status** column in the **Acquire Targets** dialog displays *Failed at Fine*.
 - Coarse and fine scans appear in the ModelSpace viewer. Coarse scans are deleted upon successful acquisition.
 - Fine scans can be viewed by double-clicking the scan item in the **Navigator** window. Fine scans are saved in the HDS Target Scans folder of the parent ScanWorld.
 - To adjust the resolution of the fine scan, see the use the **Scan Control** tab of the **Edit Preferences** dialog.

To acquire a target from a fence:

- The process of acquiring targets is usually done after an initial scan or image acquisition of the scene, which provides a general reference as to the location of targets in the scene.
1. In the Scan Control window, visually identify a potential target in the point cloud or image.
 2. For each potential target, draw the smallest fence that completely contains the target. The larger the fence is, the more likely it is that the target will be missed in the coarse scan.
 3. From the **Scanner Control** menu in the **Scan Control** window, select **Acquire Targets**. The

Acquire Targets dialog appears.

4. From the **Acquire Targets** menu in the dialog, select **HDS Target from Fence** to list the potential HDS Target in the **Acquire Targets** dialog; select **Black/White Target from Fence** to list the potential black/white target in the **Acquire Targets** dialog; select **Sphere from Pick Fence** to list the potential sphere target in the **Acquire Targets** dialog.
5. View each potential target to determine which ones you want *Cyclone* to attempt to acquire.
 - To center the ModelSpace viewpoint on a potential target, select the potential target from the list in the **Acquire Targets** dialog.
 - To rotate the ModelSpace viewpoint around a potential target, select the potential target from the list in the **Acquire Targets** dialog, and then press the **LEFT/RIGHT ARROW** keys.
 - To move up/down the list of potential targets and spheres, press the **UP/DOWN ARROW** keys.
6. Add an identifier or comment to potential targets to be acquired.
 - To add an identifier or comment: select the target, click the **ID** or **Comment** field, type an identifier or comment, and press **ENTER**.
 - Identifiers function as registration labels and can be used for automatic registration.
 - Comments function as text annotations in the ModelSpace once a target is acquired.
7. Once you have viewed potential targets, select the targets to be acquired by checking the box in the fourth column for each target to be acquired.
 - To toggle the check box for a selected target, press the **SPACEBAR** key.
8. Click **Acquire**. The target acquisition process begins; progress and results are displayed in the **Status** column.
 - During the target acquisition process, *Cyclone* performs a coarse and then a fine scan on each potential target, and then fits a vertex to the target. If the coarse scan fails to find sufficient data to begin a fine scan, the **Status** column in the **Acquire Targets** dialog displays *Failed at Coarse*. If the fine scan fails to find sufficient data to place a target, the **Status** column in the **Acquire Targets** dialog displays *Failed at Fine*.
 - Coarse and fine scans appear in the ModelSpace viewer. Coarse scans are deleted upon successful acquisition.
 - Fine scans can be viewed by double-clicking the scan item in the **Navigator** window. Fine scans are saved in the HDS Target Scans folder of the parent ScanWorld.
 - To adjust the resolution of the fine scan, see the use the **Scan Control** tab of the **Edit Preferences** dialog.

To acquire targets from selected clouds:

- The process of acquiring targets is usually done after an initial scan of the scene, which provides a general reference as to the location of targets in the scene.
1. From the **Project** menu in the **Scan Control** window, select **Open ModelSpace Viewer** (or click the **Open Viewer** button on the right side of the **Scan Control** window). A new ModelSpace viewer for the incoming scan data opens.
 2. In the ModelSpace viewer, select the point cloud from which you want to acquire potential targets.
 3. From the **Scanner Control** menu in the **Scan Control** window, select **Acquire Targets**. The **Acquire Targets** dialog is displayed.
 4. From the **Acquire** menu in the dialog, select **HDS Targets from Selected Cloud** to find potential HDS Targets in the selected point cloud and list them in the **Acquire Targets** dialog;

select **Spheres from Selected Cloud** to find potential sphere targets in the selected point cloud, and list them in the **Acquire Targets** dialog. *Cyclone* searches the point cloud for characteristics indicative of HDS tie-points or 6" sphere targets (depending on which command is selected), marks up to the maximum number of potential targets (user-defined), and lists them in the dialog.

5. View each potential target to determine which ones you want *Cyclone* to attempt to acquire.
 - To center the ModelSpace viewpoint on a potential target, select the potential target from the list in the **Acquire Targets** dialog.
 - To rotate the ModelSpace viewpoint around a potential target, select the potential target from the list in the **Acquire Targets** dialog, and then press the **LEFT/RIGHT ARROW** keys.
 - To move up/down the list of potential targets, press the **UP/DOWN ARROW** keys.
6. Add an identifier or comment to potential targets to be acquired.
 - To add an identifier or comment: select the target, click the **ID** or **Comment** field, type an identifier or comment, and press **ENTER**.
 - Identifiers function as registration labels and can be used for automatic registration.
 - Comments function as text annotations in the ModelSpace once a target is acquired.
7. Once you have viewed potential targets, select the targets to be acquired by checking the box in the fourth column for each target to be acquired.
 - To toggle the check box for a selected target, press the **SPACEBAR** key.
8. Click **Acquire**. The target acquisition process begins; progress and results are displayed in the **Status** column.
 - During the target acquisition process, *Cyclone* performs a coarse and then a fine scan on each potential target, and then fits a vertex to the target. If the coarse scan fails to find sufficient data to begin a fine scan, the **Status** column in the **Acquire Targets** dialog displays *Failed at Coarse*. If the fine scan fails to find sufficient data to place a target, the **Status** column in the **Acquire Targets** dialog displays *Failed at Fine*.
 - Coarse and fine scans appear in the ModelSpace viewer. Coarse scans are deleted upon successful acquisition.
 - Fine scans can be viewed by double-clicking the scan item in the Navigator window. Fine scans are saved in the HDS Target Scans folder of the parent ScanWorld.
 - To adjust the resolution of the fine scan, see the use the **Scan Control** tab of the **Edit Preferences** dialog.

Activate (Limit Box Manager)

Window: Limit Box Manager

Menu: Limit Box

Action: Executes command

Usage: The **Activate** command activates the selected limit box.

Activate Next (Limit Box Manager)

Window: Limit Box Manager
Menu: Limit Box
Action: Executes command
Usage: The **Activate Next** command activates the limit box in the list position below the currently active limit box.

Activate Previous (Limit Box Manager)

Window: Limit Box Manager
Menu: Limit Box
Action: Executes command
Usage: The **Activate Previous** command activates the limit box in the list position above the currently active limit box.

Add Both Flat Caps

Window: ModelSpace
Menu: Edit Object | End Caps
Action: Executes command
Usage: The **Add Both Flat Caps** command creates flat end caps on both ends of the selected object.

To add caps to both ends of the selected object:

1. Select the object that you want to cap.
2. From the **Edit Object** menu, point to **End Caps**, and then select **Add Both Flat Caps**. The caps appear.
 - To add a cap to only one end, see the *Add Flat Cap Closest to Pick* entry in this chapter.

Add Both Semi-Elliptical Heads

Window: ModelSpace
Menu: Edit Object | End Caps
Action: Executes command
Usage: The **Add Both Semi-Elliptical Heads** command creates semi-elliptical heads on both ends of the selected object.

To add semi-elliptical heads to both ends of the selected object:

1. Select the object that you want to cap.
- From the **Edit Object** menu, point to **End Caps**, and then select **Add Both Semi-Elliptical**

Heads. The caps appear.

- To add a semi-elliptical head to only one end, see the *Add Semi-Elliptical Head Closest to Pick* entry in this chapter.
- To remove caps, see the *Remove Cap Closest to Picks* entry or the *Remove Both Caps* entry in this chapter.

Add Cloud Constraint

Window: **Registration**

Menu: **Cloud Constraint**

Action: Executes command

Usage: The **Add Cloud Constraint** command creates a constraint between two ScanWorlds from two selected point clouds or meshes.

- For more information on the cloud registration process, see the [Cloud Registration](#) entry in the *Registration* chapter.
- You can create Cloud Constraints using **Add Cloud Constraint** without picking initial pick hints for the two following cases:
 1. If you have registered your ScanWorlds with targets, you are able to add cloud constraints to tighten the registration in areas where that have overlapping scan clouds but targets are not available.
 2. If you have set the ScanWorld Default Coordinate system to a shared or global coordinate system for both ScanWorlds.

To add a cloud constraint:

1. From the **ModelSpaces** tab of the **Registration** window, load both of the ModelSpaces that contain the objects you want to constrain.
- To load a ModelSpace: In the upper pane, select the ModelSpace you want to view, and then go to the **ScanWorld** menu and select **View ModelSpace**. The first ModelSpace that you open appears in the Constraint viewer on the bottom left side of the **Registration** window. The next appears on the right, and then each subsequently loaded ModelSpace alternates between loading in the left and right Constraint viewers.
2. Pick two or more corresponding points on meshes or point clouds in each viewer. Picks on objects other than meshes/clouds are ignored.
- When picking only two points, the picks must be in the same order for both ScanWorlds. We recommend three or more picks since two picks are often inadequate for the **Optimize Cloud Alignment** command to work reliably.
- The picking order does not matter for three or more picks, except where the point layout has some symmetry and the order of the picks is used to reduce this geometric ambiguity.
3. From the **Cloud Constraint** menu, select **Add Cloud Constraint**. A new Cloud Constraint is added to the Constraint List, labeled as a cloud/mesh constraint, and automatically named. Picked clouds/meshes are copied to the appropriate ControlSpace as needed.
- To view the objects involved in a selected constraint, see the *Show Constraint* entry in this chapter. Bad pick points (those that are further than the **Max Search Distance** preference in the **Edit Preferences** dialog) are displayed in blue.

- If the algorithm fails to match at least two points or if the corresponding points are more than the **Max Search Distance** preference after alignment, then a warning dialog appears.

To add a cloud constraint without initial pick hints:

- You can create Cloud Constraints using Add Cloud Constraint without picking initial pick hints for the two following cases:
 - From the **ScanWorlds' Constraints** tab of the **Registration** window, select both of the ScanWorlds that contain the clouds you want to constrain.
 - From the **Cloud Constraint** menu, select **Add Cloud Constraint**. The **Add Cloud Constraint** dialog appears.
 - Select the appropriate initial alignment method, and then click **OK**. A new Cloud Constraint is added to the Constraint List, labeled as a cloud/mesh constraint, and automatically named.

Add Constraint

Window: **Registration**

Menu: **Constraint**

Action: Executes command

Usage: The **Add Constraint** command creates a constraint between two ScanWorlds from two selected objects.

- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.
- Adding a constraint defines two objects as having similar size and shape and being located in the overlapping area of the ScanWorlds that you are registering.

To manually add a constraint:

- From the **ModelSpaces** tab of the **Registration** window, load both of the ModelSpaces that contain the objects you want to constrain.
- To view a ModelSpace: In the upper pane, select the ModelSpace you want to view, and then go to the **ScanWorld** menu and select **View ModelSpace**. The ControlSpace appears in one of the two Constraint viewer windows. The first ModelSpace that you open appears in the subwindow on the bottom left side of the **Registration** window. The next appears on the right, and then each subsequently loaded ModelSpace alternates between loading in the left and right windows.
- In each Constraint viewer, select the object you want to constrain.
 - In order to manually add a constraint, both objects must first be copied to their respective ControlSpace.
- From the **Constraint** menu, select **Add Constraint**. The constraint is added and labeled as a manual constraint and given the lowest available number (e.g., the first manually added constraint is named “Manual 1”).
- To view the objects involved in a selected constraint, see the *Show Constraint* entry in this chapter

Note: A manual constraint is not added if a constraint between the two objects already exists in the Registration.

Add/Edit Annotations...

Window: ModelSpace, Navigator, Image Viewer

Menu: ModelSpace: Tools | Annotations

Navigator: Edit

Image Viewer: Annotation

Action: Opens dialog

Usage: The **Annotations** dialog is used to add, edit, and delete annotations for selected objects.

- Annotations can be attached to any selectable object, and can consist of text, numerals, or file attachments. In addition, multiple annotations can be created for an object. Annotation visibility can be adjusted globally or specifically as described below:

- To enable the visibility of all annotations marked as “visible” in the **Image Viewer** or **ModelSpace** windows, select [**Show Annotations**](#) from the **Annotations** submenu (**Annotation** menu in the **Image Viewer** window).
- To toggle the visibility of an individual annotation, select the object with which the annotation is associated, then open the **Annotations** dialog. Select/deselect the box in the **Visibility** column for the desired annotation. The **Show Annotations** option in the viewer must be enabled before any annotation is drawn.
- The **Annotations** dialog contains two tabs: the **Annotations** tab, and the **Custom** tab. The **Annotation** tab lists all annotations for the current selection, including annotations created in the **Custom** tab, feature code annotations, and registration labels.
- See the [**Add/Edit Custom Annotations**](#) command for information on creating and editing custom annotations. Refer to the following text for directions on adding, editing, and deleting annotations using the **Annotations** tab of the **Annotations** dialog.

Note: The procedures below refer to the menus and sub-menus in the **ModelSpace** window for accessing the **Annotations** dialog. If you are in the **Navigator** or **Image Viewer** windows, substitute the appropriate menus.

To create a new annotation:

Note: In addition to the procedure described below, you can create new annotations from the **Custom** tab when you are in a ModelSpace View. For information on creating custom annotations, see the *Add/Edit Custom Annotations* entry in this chapter.

1. Select the object to which you want to attach an annotation.
- When creating an annotation in the **Image Viewer** window, select an existing vertex, or pick a spot on the image.
- When you want to annotate an area within the ModelSpace View rather than a specific object, insert an info marker, and then attach the annotation to the info marker.
2. From the **Tools** menu, point to **Annotations**, then **Add/Edit Annotations....** The **Annotations** dialog appears.
3. Click the **New Annotation** icon. The **New Annotation** dialog appears.
4. Enter the key, enter the value, and select the desired type of annotation according to the field information provided below:
 - The **Key** field is used to identify the annotation (e.g., pipe size) and cannot be used twice for the same object. However, the same annotation key can be used by multiple objects. For information on showing/hiding annotation keys, see the *Show Annotation Keys* entry in this chapter.
 - The **Value** field is the main information of the annotation. (For example, if the annotation key is "pipe size," you can enter the actual pipe size in this field.) You may also attach a file of any type (including executable files) as an annotation value, either by typing in the file name and location, or by clicking the **Browse** button and selecting the desired file from the browser. If you choose to attach a file, you must select **External Document** for the **Type** field to enable additional functionality.

Note: File and program names/paths that include spaces and/or non-standard characters (e.g., commas, semi-colons) must be enclosed by quotation marks.

- The **Type** field identifies the type of data entered in the **Value** field. The types available for annotation values are **Text**, **Integer**, **Decimal**, and **External Document**.
- 5. Click **OK** to assign the annotation and return to the **Annotations** dialog.

To edit annotations:

1. Select the object for which you want to change the annotations.
2. From the **Tools** menu, point to **Annotations**, and then select **Add/Edit Annotations....** The **Annotations** dialog appears.
3. Make the desired changes to the **Value**, **Type**, or **Visibility** field; the changes are applied immediately.

Note: The **Key** field cannot be modified.

To attach a file as an annotation:

1. Select the object to which you want to attach a file.
2. From the **Tools** menu, point to **Annotations**, then select **Add/Edit Annotations....** The **Annotations** dialog appears.
3. Create an annotation and set the **Type** field to **External Document**.
4. Click the **Browse File** icon. The **Open** dialog appears.
5. Find and select the desired file and click **Open** to return to the Annotations dialog. The file that you selected is displayed in the **Value** field.
- You can attach a text document, or specify a command line executable.

To open an external document:

1. Select the object containing the annotated file attachment to be opened.
2. From the **Tools** menu, select **Annotations**, then select **Add/Edit Annotations....** The **Annotations** dialog appears.
3. Highlight the key containing the file you want to open.
4. Click the **Open a File Annotation** icon.
- If the annotation file is an executable file, use the **Launch an Executable File** icon , rather than the **Open a File** icon.

To launch an executable file attachment:

1. Select the object or point containing the executable file attachment to be launched.
2. From the **Tools** menu, point to **Annotations**, then select **Add/Edit Annotations....** The **Annotations** dialog appears.
3. Highlight the key containing the executable you want to launch.
4. Click the **Launch an Executable File** icon to launch the program.

To edit an annotation's properties:

1. Select the object(s) for which the annotation properties are to be edited.

Hint: Multi-selecting like objects (e.g., pipes) and setting their annotations properties to identical values is a good visual tool for quickly distinguishing groups of annotations.

2. The **Tools** menu, point to **Annotations**, then select **Add/Edit Annotations....** The **Annotations** dialog appears.
3. Highlight the annotation key name of the desired annotation.
4. Click the **Annotation Properties** icon. The **Annotation Properties** dialog appears, containing the following settings:

TEXT COLOR

Use the color picker to select the color used to display the annotation text and callout line.

FONT SIZE

Enter a point value for the font size used to display the annotation text.

LEADER LINE LENGTH

Enter the length (in pixels) of the line connecting the annotation text to the source object.

LEADER LINE ANGLE

Enter the angle of the callout line.

5. Edit the desired settings and then click **OK**. Your changes are made.

To delete an annotation:

1. Select the object(s) (in the **Image Viewer** window, select a vertex) containing the annotation(s) to be deleted.
2. From the **Tools** menu, point to **Annotations**, then select **Add/Edit Annotations....** The

Annotations dialog appears.

3. Highlight the annotation key name of the annotation to be deleted, and then click the **Delete** icon. The annotation is deleted.

Add/Edit Camera Images...

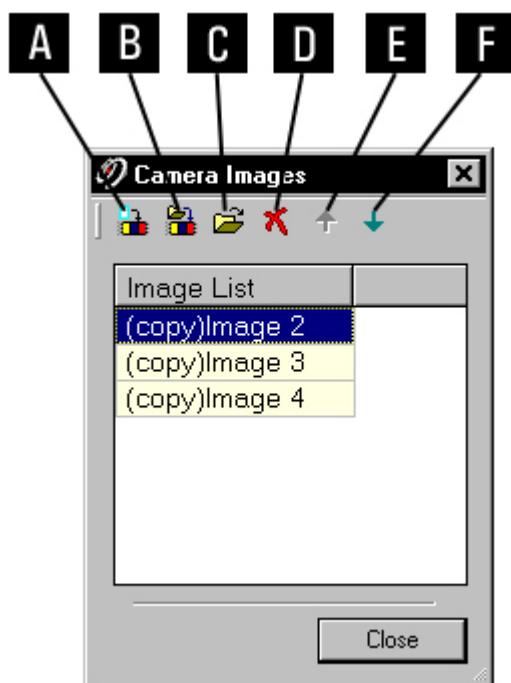
Window: ModelSpace

Menu: Tools | Annotations

Action: Opens dialog

Usage: The **Camera Images** dialog is used to add, delete, and view images associated with a camera object.

Camera Images Dialog



- A. Add image.
- B. Add image file.
- C. Open image.
- D. Delete image.
- E. Move the selected image one position higher in the list.
- F. Move the selected image one position lower in the list.

To open an existing image:

1. From the **Camera Images** dialog, select the image you want to view from the **Image List**.
2. Click the **Open Image** icon.. The image is opened in an **Image Viewer** window.

To delete an existing image:

1. From the **Camera Images** dialog, select the image you want to delete from the **Image List**.
2. Click the **Delete Image** icon. The image is deleted from the **Image List**.

To add a camera image from within *Cyclone*:

1. From the **Tools** menu, point to **Annotations** and then select **Add/Edit Camera Images**. The **Camera Images** dialog appears.
2. Click the **Add Image** icon. The **Select Images** dialog appears displaying a database browser in the left pane.
3. Select the image you want, and then click the **Add** icon between the panes. The image appears in the right pane.
 - To add more images, repeat this step.
 - To remove images, select the image in the right pane and click the left arrow icon between the panes.
4. Click **OK**. The added image appears in the **Image List**.

Note: If the image being added is from a different database, the image is prefixed by "(Copy)" to indicate that it has been copied into the local directory.

5. Click **Close**.

To add a camera image from a file:

1. From the **Tools** menu, point to **Annotations** and then select **Add/Edit Camera Images**. The **Camera Images** dialog appears.
2. Click the **Add Image File** icon. The **Import File** dialog appears.
3. Navigate to the image file you want to import.
4. Select **Open**. The **Camera Images** dialog reappears, and the imported file appears in the **Image List**.
5. Click **Close**.

Add/Edit Colors...

Window: ModelSpace

Menu: Edit Object | Appearance

Action: Opens dialog

Usage: The **Color Palette** dialog is used to save and restore color samples.

- The **Color Palette** can operate independently of the [Color and Material Editor](#). If the **Color and Material Editor** is closed, **Color Palette** operations interact directly with the ModelSpace.

To apply a saved color from the Color Palette directly to selected object(s) in the ModelSpace:

1. In the current ModelSpace, select the object(s) whose color you want to change.
2. Close the **Color and Material Editor** dialog if it is open.
3. From the **Color Palette**, select the color that you want to apply.
4. From the **Color Palette** shortcut menu, select **Apply Color** (or double-click the color you want from the **Color Palette**). The color (the diffuse and ambient properties) of the selected object is changed.

To apply a saved color from the Color Palette to the current color swatch in the Material Editor:

1. From the **Color Palette**, select the color you want to apply.
2. From the **Color Palette** shortcut menu, select **Apply Color** (or double-click the color you want from the **Color Palette**). The current color swatch changes to match the selected color in the **Color Palette**.

To save a color to the Color Palette from a selected object in the ModelSpace:

1. Open the **Color Palette**.
2. Close the **Color and Material Editor** dialog if it is open.
3. Select the object whose color you want to save.
4. From the **Color Palette** shortcut menu, select **Add Color**. The current color is inserted into the **Color Palette**.

To save a color to the Color Palette from the Color and Material Editor dialog:

1. Create/place the color you want in the **Current Color** swatch.
2. From the **Color Palette** shortcut menu, select **Add Color**. The current color is inserted into the **Color Palette**.

To replace a current saved color in the Color Palette:

1. In the **Color Palette**, select the color you want to replace.
2. Create/place the color you want in the **Current Color** swatch, or, if the **Color and Material Editor** is closed, select the object whose color you want in the current ModelSpace.
3. From the **Color Palette** shortcut menu, select **Replace Color**.

To delete a color from the Color Palette:

1. Select the color that you want to delete.
2. From the **Color Palette** shortcut menu, select **Delete Color**.

Add/Edit Custom Annotations...

Window: ModelSpace

Menu: Tools | Annotations

Action: Opens dialog

Usage: The **Custom** tab in the **Annotations** dialog is used to add, edit, and delete annotations for selected objects using a customizable framework.

- The **Annotations** dialog contains two tabs: the **Custom** tab, and the **Annotations** tab. See the [Add/Edit Annotations](#) command for information on creating and editing annotations using the **Annotations** tab.
- Though the **Annotations** and **Custom** tabs contain much of the same data, the annotations are displayed and accessed differently. In the **Annotations** tab, each annotation is listed as a row; each row contains a unique annotation key, a value, a type, and a visibility toggle. If there are ten annotations for one object, they are all listed. In contrast, the **Custom** tab contains only one entry (row) for each object selected, no matter how many annotations exist for that object. Any number of columns may be defined by the user, in addition to the default **Name** and **Position** columns, which can't be deleted (though they can be hidden). Each column is equivalent to an annotation in the **Annotations** tab, and each is headed by an annotation key. In the cells below the annotation key column headers, you can either enter new values, or select from the pull-down list for each key, which lists the most recently entered values for re-use. Each *new* value entered is added to the pull-down list.

Note: The lookup lists used to populate these cells can be edited via the [Customize Lookup Lists...](#) command in the Navigator **Edit** menu.

To add a custom annotation key (column) to the Custom tab:

1. Select the object(s) to which you want to attach a custom annotation.
2. From the **Tools** menu, point to **Annotations**, then select **Add/Edit Custom Annotations...**. The **Annotations** dialog appears with the **Custom** tab displayed.
3. Click the **Add Custom Annotation Keys** icon. The **Customized Annotation Keys** dialog appears.
4. Click the **Create a new key** icon.
5. Enter a unique name in the **Keys** column. The same key name cannot be used more than once.
6. Click in the **Type** column and select a value type from the pull-down list: **External Document**, **Decimal**, **Integer**, or **Text**. This cell is identical in functionality to the **Type** field in the **Annotation** tab.
7. Click in the **Lookup List** column. Leave this cell blank if you plan to define a new list, or select a pre-defined list from the pull-down.

Note: For information on editing lookup lists, see the *Customize Lookup Lists* entry in this chapter.

8. Use the **Arrow** icons to move keys up or down the list as desired. The order from top to bottom in which keys are listed in this window dictates the order in which columns are displayed from left to right in the **Custom** tab of the **Annotations** dialog.
- Use the **Visibility** toggle option to show/hide this or any other key in the **Custom** tab.
9. Click on **Close** to return to the **Custom** tab of the **Annotations** dialog. The new column is displayed.
- Column displays can be resized as desired by clicking and dragging the divider lines separating the column headers. Additionally, columns can be moved by picking up column headers and dragging them to a different location in the table.
- Columns can be hidden by dragging the key off the header. Hiding a column does not affect existing annotations.

Note: To edit existing annotations, see the *Add/Edit Annotations* entry in this chapter for more information.

To add a custom annotation to an object:

1. Select the object(s) being annotated.
2. From the **Tools** menu, select **Annotations**, then **Add/Edit Custom Annotations....** The **Annotations** dialog appears with the **Custom** tab displayed.
3. Create a new annotation key.
4. Enter a value for each selected object under the newly created annotation key (or pick a value from the list if a lookup list has been defined). The custom annotation is created.
- If you move the **Annotations** tab, you will see the newly created annotation.

Note: To edit or delete existing annotations, see the *Add/Edit Annotations* entry in this chapter for more information.

To populate and save a new lookup list:

Note: These instructions are suitable for creating brief lookup lists. You may prefer to use the **Customize Lookup Lists...** command in the Navigator **Edit** menu to create more extensive lists.

1. Select the object(s) being annotated.
2. From the **Tools** menu, select **Annotations**, then **Add/Edit Custom Annotations....** The **Annotations** dialog appears with the **Custom** tab displayed.
3. Click in the cell in the object's row and in the column with the key name for which a list is to be created, and type a list entry followed by **ENTER**. Repeat this step for each list entry.
- Notice that, each time you click in the cell, a list of previously entered selections appears, with the most recent entry at the top of the list. Also notice that, as you type a new value, the entry is auto-completed for you; continue typing, press **BACKSPACE** to ignore the auto-match, or press **ENTER** to accept the auto-match
4. When you have entered all the desired lookup list options, click the **Add Custom Annotation Keys** icon to display the **Customized Annotation Keys** dialog.
5. Highlight the key containing the list you just entered, and click the **Save this key's lookup list** icon. The **Save Lookup List** dialog is displayed.
6. Enter a new name for the lookup list, or leave the default name in place, as desired. Click **OK**

to save the list and return to the **Customized Annotation Keys** dialog. The newly saved list will now appear as a pull-down option in the **Lookup List** column.

- See the *Customize Lookup Lists* command entry for information on editing and deleting lookup lists.

To delete a custom annotation key:

1. Select the object(s) containing the annotation key to be deleted.
2. From the **Tools** menu, point to **Annotations**, then select **Add/Edit Custom Annotations...**. The **Annotations** dialog appears with the **Custom** tab displayed.
3. Click the **Add Custom Annotation Keys** icon to display the **Customized Annotation Keys** dialog.
4. Highlight the key to be removed, and click the **Delete** icon. The annotation key is removed from this dialog, and the corresponding column in the **Custom** tab in the **Annotations** dialog is removed as well.

Note: Deleting a custom annotation key as described above *does not* remove annotations from objects. If you wish to remove actual annotations, use the **Delete** icon in the **Annotations** tab.

To copy Custom Annotation values to the Windows clipboard:

1. Select the object(s) containing the desired custom annotation.
2. From the **Tools** menu, point to **Annotations**, then select **Add/Edit Custom Annotations...**. The **Annotations** dialog appears with the **Custom** tab displayed.
3. Multi-select the entries to copy, or select nothing to copy all entries.
4. Click the **Copy Custom Annotations to Clipboard** icon. The selected contents of the custom tab are copied as tab-delimited text to the Windows clipboard, from which it may be pasted into another application (e.g., Notepad).

Add/Edit Cutplanes...

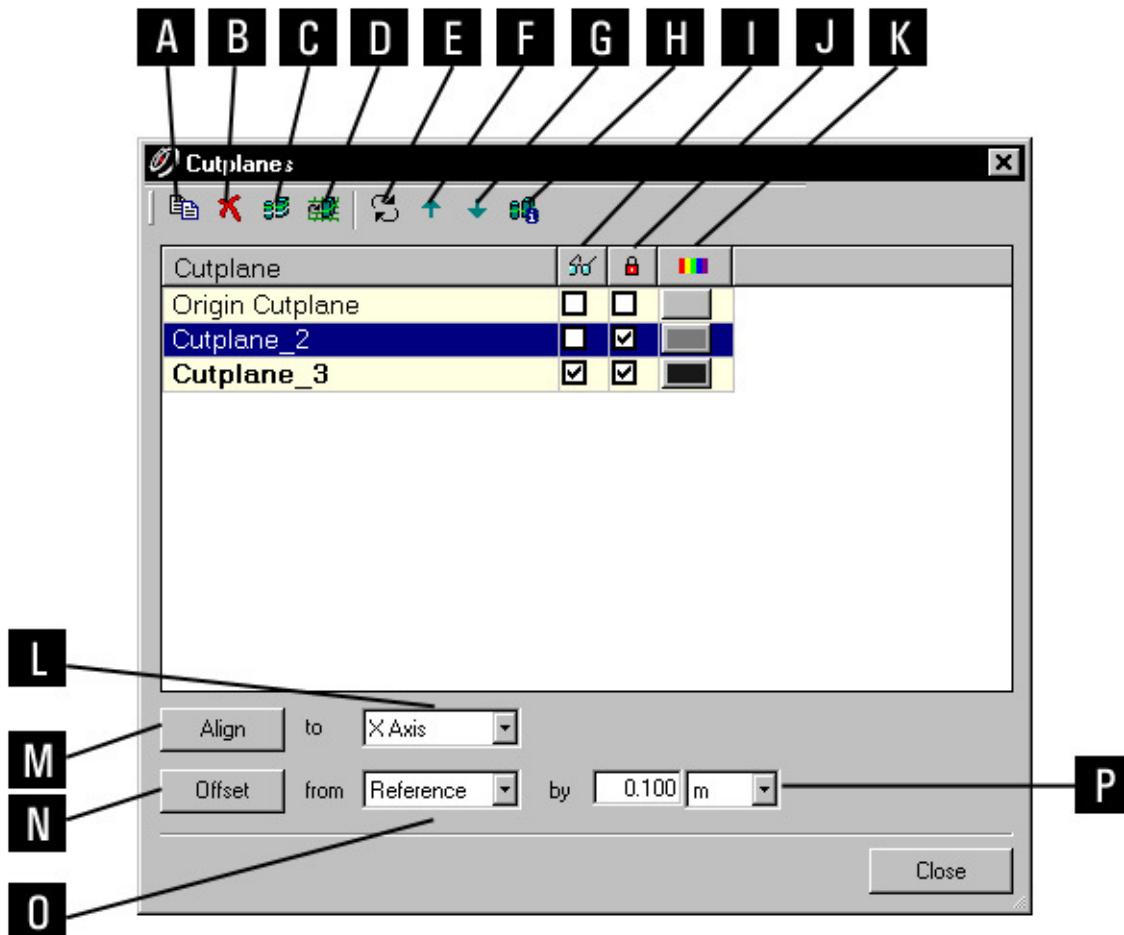
Window: ModelSpace

Menu: Tools | Cutplane

Action: Opens dialog

Usage: The **Cutplanes** dialog is used to edit cutplanes and cutplane attributes.

Cutplanes Dialog



- A. Create a copy of the selected Cutplane.
- B. Delete the selected Cutplane.
- C. Set the selected Cutplane as the active Cutplane, which is used in all Cutplane operations.
- D. Add a Reference Plane on the selected Cutplane. The new Reference Plane appears in the Reference Planes dialog.
- E. Flip the selected Cutplane normal in order to toggle the visibility of geometry above/below the default Cutplane.
- F. Move the selected Cutplane one position higher in the list.
- G. Move the selected Cutplane one position lower in the list.
- H. View origin and normal coordinates for the selected Cutplane.

- I. Toggle the visibility of the Cutplane.
- J. Lock the Cutplane in place.
- K. Click to use the color picker to set material properties for the Cutplane.
- L. Select the axis to which you want to align the selected Cutplane.
- M. To align the selected Cutplane, select an axis from the Align drop-down menu, and then click Align.
- N. Execute the Cutplane offset with the current settings.
- O. Choose Reference or Self from the Offset drop-down list, depending on whether you want to use the Reference Plane or the Cutplane itself as the offset reference point.
- P. This setting determines the increment used for the Raise Active Cutplane and Lower Active Cutplane commands in the **Cutplanes** submenu.

To offset the selected cutplane:

1. From the **Tools** menu, point to **Cutplane**, and then select **Add/Edit Cutplanes....** The **Cutplanes** dialog appears.
2. Choose **Reference** or **Self** from the **Offset** drop-down list, depending on whether you want to use the Reference Plane or the Cutplane itself as the reference point.
3. Enter a value in the **by** box, and then click **Offset**. The selected Cutplane is moved, and the increment for raising and lowering Cutplanes is set.

Add/Edit Feature Code...

Window: ModelSpace

Menu: Tools | Annotations

Action: Opens dialog

Usage: The **Add/Edit Feature Code** dialog is used to add and edit Feature Code annotations for a selected object.

- Specific to surveying, feature codes are used to identify specific reference objects. For example, "toc" is a feature code used to represent "top of curb." Measurements taken with the top of curb as a reference would be annotated with the feature code "toc."
- *Cyclone* Feature Codes maintains their "feature code" identity when exported.

To add a feature code annotation:

1. Select the object to which you want to attach a feature code annotation.
 - It is not possible to add a feature code annotation to an Info Marker, Camera, or similar

- “reference-only” object.
2. From the **Tools** menu, point to **Annotations**, then select **Add/Edit Feature Code....** The **Add/Edit Feature Code** dialog appears.
 3. Select a pre-defined Features List, and then assign a Feature Code to the selected object from the Feature list. Alternatively, you can enter a new Feature Code, which is added to the selected Feature List.
 4. Click **OK** to attach the feature code annotation to the selected object(s).

Note: Feature code annotations are listed in the **Annotations** dialog in addition to any other annotations which may have been added. If you wish to edit feature code annotations, do so from the **Annotations** dialog.

To edit a feature code annotation:

1. Select the object for which the feature code annotation is to be edited.
2. From the **Tools** menu, point to **Annotations**, then select **Add/Edit Feature Code....** The **Add/Edit Feature Code** dialog appears.
3. You may change the Feature List, and/or the Feature Code selected from the Feature List as desired. Alternatively, you may enter a new Feature Code, which is then added to the selected Feature List.

Note: The annotation key **Feature Code** may not be edited.

4. Click **OK** to attach the updated feature code annotation to the selected object(s).

To delete a feature code annotation:

1. Select the object(s) with the feature code annotation to be deleted.
2. From the **Tools** menu, point to **Annotations**, then select **Add/Edit Annotations....** The **Annotations** dialog appears.
3. Select the feature code annotation from the list in the **Annotations** tab, and click the **Delete** icon in the toolbar. The feature code annotation is removed.

Add/Edit Limit Boxes

Window: **ModelSpace**

Menu: **View | Add/Edit Limit Boxes**

Action: Opens dialog

Usage: The **Limit Box Manager** dialog is used to add, delete and edit the properties of limit boxes in the current ModelSpace View.

- Limit boxes are unique to the ModelSpace View in which they are defined and are not shared with other ModelSpace Views.
- For information on specific **Limit Box Manager** commands, see the individual command entries.

Limit Box Manager Dialog

Limit boxes are unique to the ModelSpace in which they are defined and are not shared with other views.

Name: Displays the name of the Limit Box.

X, Y, Z: Type topic text here.

W, H, D: Displays limit box dimensions.

Align: Indicates whether or not the limit box is aligned with the current user coordinate system axes.

New: Creates a new limit box that is a copy of the currently selected limit box. The new limit box is automatically identified as a copy of the original limit box.

Delete: Deletes the selected limit box.

Set Sets the selected Limit Box as the active Limit Box.

Current:

Info: Displays the parameters of the selected limit box.

Enable Select this check box to enable the current limit box.

Limitbox:

Show Select this check box to display limit boxes or clear this check box to hide limit boxes.

Limit

Box:

Show All: Select this check box to display all limit boxes or clear this check box to display only activated limit boxes. When selected (and all limit boxes are displayed), the active limit box is highlighted with thick lines and the selected limit box is displayed with dashed lines.

Add/Edit Materials...

Window: **ModelSpace**

Menu: **Edit Object | Appearance**

Action: Opens dialog

Usage: The **Material Palette** is used to save and restore material samples.

The **Material Palette** functions identically to the **Color Palette** (See the [Add/Edit Colors command](#).) except that it handles materials (combinations of color properties) rather than colors.

Add/Edit Reference Planes...

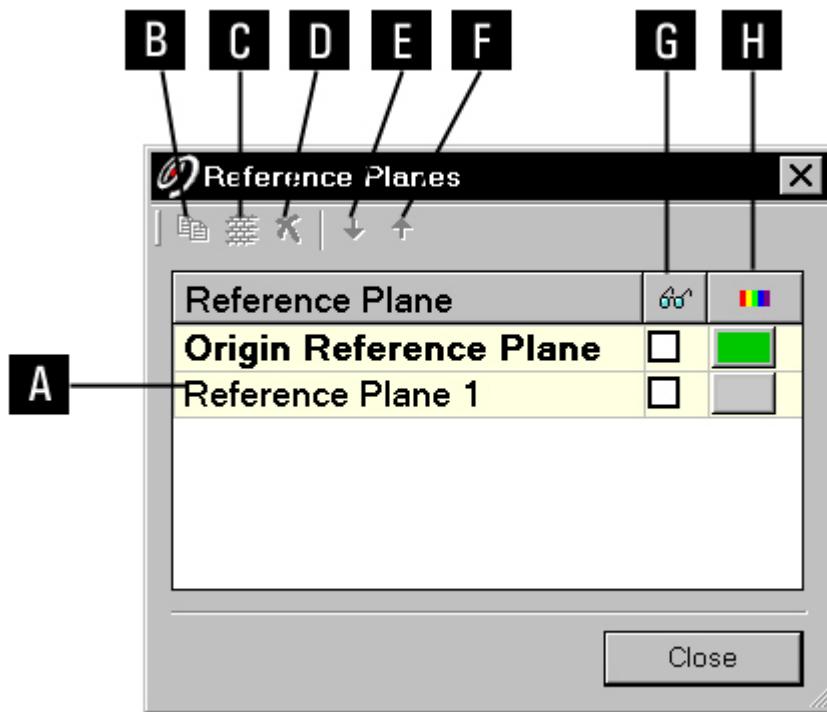
Window: **ModelSpace**

Menu: **Tools | Reference Plane**

Action: Opens dialog

Usage: The **Reference Planes** dialog is used to add, edit, and delete Reference Planes.

Reference Planes Dialog



- A. List of Reference Planes in current ModelSpace.
- B. Create a copy of the selected Reference Plane.
- C. Set the selected Reference Plane as the active Reference Plane, which is used in all Reference Plane operations.
- D. Delete the selected Reference Plane.
- E. Move the selected Reference Plane one position lower in the list.
- F. Move the selected Reference Plane one position higher in the list.
- G. Toggle the visibility of the Reference Plane.
- H. Click to use the color picker to adjust the color of the Reference Plane.
 - To rename a Reference Plane: click the name of the Reference Plane to be changed, click it again after a short pause to select the current name, type a new name, and then press **ENTER**.

Add/Edit Registration Label

Window: Registration

Menu: Constraint

Action: Opens dialog

Usage: The **Registration Label** dialog is used to add or edit a label to the objects involved in the selected constraint, or to the object selected in the Constraint Viewer.

- To edit the label of the object selected in the Constraint Viewer, right-click in the Constraint Viewer and invoke this command from the context menu. The command in the main menu applies to the selection in the **Constraint List**.
- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.

To add or edit the registration label of the objects involved in a constraint:

1. Select the constraint in the **Constraint List**.
2. From the **Constraint** menu, select **Add/Edit Registration Label**. The **Registration Label** dialog appears.
3. Enter the registration label, and click **OK** to assign the registration label to the objects involved in the selected constraint.
4. Click **OK**. The objects involved in the constraint may be present in other ControlSpaces and ModelSpaces. The corresponding objects are renamed wherever they are found under the constraint's ScanWorlds.

To add or edit the registration label of the object selected in a Constraint Viewer:

1. Select the object in the **Constraint Viewer**.
2. Right-click in the **Constraint Viewer** to invoke the context menu. From the context menu, select **Add/Edit Registration Label**. The **Registration Label** dialog appears.
3. Enter the registration label, and click **OK** to assign the registration label to the selected object.
4. Click **OK**. The selected object may be present in other ControlSpaces and ModelSpaces. The corresponding objects are renamed wherever they are found under the constraint's ScanWorlds.

Add/Edit ScanWorld Annotations...

Window: Scan Control

Menu: Edit

Action: Opens dialog

Usage: The **Add/Edit ScanWorld Annotations...** command opens the **Annotations** dialog, which is used to attach annotations to the current destination ScanWorld.

- The **Annotations** dialog opened from the **Scan Control** window functions identically to the **Annotations** dialog opened from other windows (See the [Add/Edit Annotations](#) command),

except that there is no visibility option, there is no “Custom” tab, and the ScanWorld itself is used as the “object” to which all annotations are attached.

To edit ScanWorld annotations:

1. From the **Edit** menu, select **Add/Edit ScanWorld Annotations....** The **Annotations** dialog appears.
2. Make the desired changes to the **Value** and **Type** fields, and then click **OK**.

Add/Edit Target Height

Window: **Registration**

Menu: **Constraint**

Action: Opens dialog

Usage: The **Set Target Height** dialog is used to add (or edit) a target height to the objects involved in the selected constraint, or to the object selected in the Constraint Viewer.

- To edit the target height of the object selected in the Constraint Viewer, right-click in the Constraint Viewer and invoke this command from the context menu. The command in the main menu applies to the selection in the **Constraint List**.
- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.

To add or edit the target height of the objects involved in a constraint:

1. Select the constraint in the **Constraint List**.
2. From the **Constraint** menu, select **Add/Edit Target Height**. The **Set Target Height** dialog appears.
3. Select the ScanWorld containing the object to be edited, enter the target height, and click **OK** to assign the target height to that object.
4. Click **OK**. The objects involved in the constraint may be present in other ControlSpaces and ModelSpaces. The target height of the corresponding objects are set wherever they are found under the constraint’s ScanWorlds.

To add or edit the target height of the object selected in a Constraint Viewer:

1. Select the object in the **Constraint Viewer**.
2. Right-click in the **Constraint Viewer** to invoke the context menu. From the context menu, select **Add/Edit Target Height**. The **Set Target Height** dialog appears.
3. Enter the target height, and click **OK** to assign the target height to the selected object.
4. Click **OK**. The selected object may be present in other ControlSpaces and ModelSpaces. The target height of the corresponding objects are set wherever they are found under the constraint’s ScanWorlds.

Add Face

Window: ModelSpace

Menu: Tools | Mesh

Action: Executes command

Usage: The **Add Face** command adds a new triangle to a mesh object by connecting existing vertices and/or creating new vertices on the mesh.

- Mesh editing can be performed only on a mesh object, as opposed to a point cloud that is viewed as a mesh. To create a mesh object from a point cloud, see the [Create Mesh](#) command.

To add a face to a mesh:

- Multi-select three points in the ModelSpace View (the points do not have to be on the mesh object).
 - If none of the first three points selected is on a mesh object, a fourth selection must be made on a mesh object.
- From the **Tools** menu, point to **Mesh**, and then select **Add Face**. A new triangle is added.

Add Flat Cap Closest to Pick

Window: ModelSpace

Menu: Edit Object | End Caps

Action: Executes command

Usage: The **Add Flat Cap Closest to Pick** command creates a flat end cap on the selected object's end nearest to the picked point.

To add a flat cap to the selected object:

- Pick a point on the object near the end that you want to cap.
- From the **End Caps** submenu, select **Add Flat Cap Closest to Pick**. The cap appears.
 - To add flat caps to both ends of the selected object, see the *Add Both Flat Caps* entry in this chapter.
 - To remove end caps, see the *Remove Cap Closest to Picks* entry or the *Remove Both Caps* entry in this chapter.

Add Inside Fence

Window: ModelSpace

Menu: Selection | Point Cloud Sub-Selection

Action: Executes command

Usage: The **Add Inside Fence** command adds any points within the fence to the sub-selection set.

- For more information, see the [Advanced Segmentation](#) entry in the *Modeling* chapter.

Add Outside Fence

Window: ModelSpace

Menu: Selection | Point Cloud Sub-Selection

Action: Executes command

Usage: The **Add Outside Fence** command adds any points outside a fence to the sub-selection set.

- For more information, see the [Advanced Segmentation](#) entry in the *Modeling* chapter.

Add/Replace Coordinate...

Window: Scan Control (ScanStation, ScanStation 2, HDS6000)

Menu: Project

Action: Opens dialog

Usage: The **Add/Replace Coordinate** dialog is used to add or edit known or assumed coordinates that will be used in the Field Setup or Traverse procedures.

- The known or assumed coordinates are saved in the **Known Coordinates** ScanWorld under the current Project.
- For more information on Field Setup and Traverse, see the [Field Setup \(ScanStation, ScanStation 2\)](#) and [Traverse \(ScanStation, ScanStation 2\)](#) entries in the *Scanning with the ScanStation, ScanStation 2, and HDS3000* chapter, or the [Field Setup \(HDS6000\)](#) and [Traverse \(HDS6000\)](#) section in the *Scanning with the HDS6000 and HDS4500* chapter.

To add known or assumed coordinates:

- From the **Project** menu, select **Add/Replace Coordinate**. The **Add/Replace Coordinate** dialog appears.
- Enter the coordinate ID and position, and click **Add** to add a vertex with that ID and position to the **Known Coordinates** ScanWorld.

To edit or replace known or assumed coordinates:

- From the **Project** menu, select **Add/Replace Coordinate**. The **Add/Replace Coordinate** dialog appears.
- Select the existing coordinate ID, enter the new coordinate, and click **Replace** to replace the existing vertex's coordinates in the **Known Coordinates** ScanWorld.

Add ScanWorld...

Window: Registration

Menu: ScanWorld

Action: Opens dialog

Usage: The **Select ScanWorlds for Registration** dialog is used to add ScanWorlds to the current Registration.

To add a ScanWorld to the current registration:

1. From the **ScanWorld** menu, select **Add ScanWorld....** The **Select ScanWorlds for Registration** dialog appears. The left side of the dialog contains a Navigator pane.
2. In the Navigator pane, select the ScanWorld(s) that you want to add to the Registration, and then click the **Add** icon. The selected ScanWorld appears in the right-hand pane.
3. Continue adding ScanWorlds until all of the ScanWorlds that you want to add to the Registration appear in the right-hand window, and then click **OK**. The ScanWorlds are added to the Registration.

Add Semi-Elliptical Head Closest to Pick

Window: ModelSpace

Menu: Edit Object | End Caps

Action: Executes command

Usage: The **Add Semi-Elliptical Head Closest to Pick** command creates a semi-elliptical head on the selected object's end nearest to the picked point.

To add a semi-elliptical head to the selected object:

1. Pick a point on the object near the end that you want to cap.
2. From the **End Caps** submenu, select **Add Semi-Elliptical Head Closest to Pick**. The cap appears.
 - To add semi-elliptical heads to both ends of the selected object, see the *Add Both Semi-Elliptical Heads* entry in this chapter.
 - To remove flat caps, see the *Remove Cap Closest to Picks* entry or the *Remove Both Caps* entry in this chapter.

Adjust Exposure...

Window: Scan Control

Menu: Scanner Control

Action: Opens dialog

Usage: The **Adjust Exposure** dialog is used to adjust the exposure time used when capturing images from the HDS2500 scanner.

- When acquiring an image in the **Scan Control** window (using the [Get Image](#) command), the prevailing lighting conditions can greatly affect the quality and usability of the image. To compensate for the level of light in the scene, adjust the image exposure.

To adjust image exposure:

- From the **Scanner Control** menu in the **Scan Control** window, select **Adjust Exposure**. The **Camera Exposure** dialog appears.
- From the **Setting** list, make the appropriate selection for the current lighting conditions. Exposure times can range from 0.1 to 2000 milliseconds.
- Adjust settings for the option selected in the **Setting** field.
 - To specify exposure time in milliseconds, enter a value in the **Exposure** box.
 - To view a preview image using the current settings, click **Preview**.
- Click **OK**. The current settings are applied to the entry in the **Setting** field. These settings are persistent for the scanner object in the database.
- Acquire another image (using the [Get Image](#) command) to view the results of your adjustments.

Adjust Image...

Window: Image Viewer, Scan Control

Menu: Edit

Action: Opens dialog

Usage: The **Adjust Image...** command launches the **Brightness/Contrast** dialog, which is used to adjust the brightness and contrast of the current image.

To adjust image viewer images:

- From the **Edit** menu, select **Adjust Image....** The **Brightness/Contrast** dialog appears.
- Use the sliders or enter a specific value to set the brightness/contrast levels, and then click **OK**. The image is adjusted.
 - To exit the **Brightness/Contrast** dialog without saving changes, click **Cancel**.

Align

Window: ModelSpace

Menu: Edit Object

Action: Displays submenu

Usage: Pointing to **Align** displays submenu commands, submenus, and dialogs to align objects in a variety of ways.

The following commands, submenus, and dialogs are available from the **Align** submenu:

[Align Surfaces](#), [Align to Axis](#), [Coincident](#), [Colinear](#), [Coplanar](#), [Parallel](#), [Perpendicular](#), [Codirectional](#), [Put at Angle...](#), [Put at Distance....](#)

- For **Align** submenu command, submenu, and dialog details, see the individual command entries.

Align Surfaces

Window: ModelSpace

Menu: Edit Object | Align

Action: Executes command

Usage: The **Align Surfaces** command aligns the surface of one or more selected objects to the surface of a reference object.

To align the surfaces of objects:

- Multi-select the objects you want to align.
 - The last object selected is the reference object (the one to which the others align).
- From the **Align** submenu, select **Align Surfaces**. The surface of the object(s) is aligned to the surface of the reference object.

Align to Axis

Window: ModelSpace

Menu: Edit Object | Align

Action: Displays submenu

Usage: Pointing to **Align to** displays submenu commands to align one or more selected objects to the current X, Y, Z, or XYZ axis.

The following commands are available from the **Align to Axis** submenu:

X Axis, Y Axis, Z Axis, XYZ Axes

To align the selected object to the current X, Y, Z, or XYZ axes:

- Select the object(s) you want to align.
- From the **Align to Axis** submenu, select the axis to which you want the selected object aligned. The object is rotated about its center until its principal direction matches the selected axis.
 - To align to the XYZ axes, the object will take the path of the least rotation(s) to make the axes coincide.

Align to Selection

Window: ModelSpace

Menu: Viewpoint

Action: Executes command

Usage: The **Align to Selection** command orients the ModelSpace viewpoint to be parallel to the axis of the selected object.

To align the viewpoint to the selected object:

1. Select the object to whose axis you want to align the viewpoint.
2. From the **Viewpoint** menu, select **Align to Selection**. The viewpoint is moved the shortest possible distance in order to be parallel with the axis of the selected object.
 - If a point cloud is selected, the viewpoint is oriented to match the orientation of the point cloud's scanner. If a scanner object is selected, the viewpoint is oriented to the view of the scanner when it took the scan. For HDS3000, ScanStation, and ScanStation 2 ScanWorlds, this is a useful way to get to a photographic view of the scan data.

Align Vertices to Axes

Window: ModelSpace

Menu: Tools | Drawing

Action: Toggles menu item ON/OFF

Usage: The **Align Vertices to Axes** command toggles the restriction of each vertex to be along the same X or Y axes as the previously plotted vertex.

- After plotting the first vertex in a drawing, the next vertex snaps to the same X or Y axis (whichever is closer) on the drawing plane as the previous vertex when **Align Vertices to Axes** is ON.

Align View to Active Plane

Window: ModelSpace

Menu: Tools | Reference Plane

Action: Executes command

Usage: The **Align View to Active Plane** command orients the ModelSpace viewer to a top-down view of the active Reference Plane and centers the Reference Plane axes relative to the window borders.

Alignment and Section

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Alignment and Section** displays submenu commands used to generate and manage alignments and sections.

The following commands and dialogs are available from the **Alignment and Section** submenu:

Create Alignment, Append to Alignment, Switch Alignment Start/End, Create Sections..., Create Sections from Picks..., Sections Manager..., Set Active Alignment, Reset Active Alignment

- For more information about alignments and sections, see the *Sections Manager* chapter.

All in Selection

Window: ModelSpace

Menu: Create Object | Slice

Action: Executes command

Usage: The **All in Selection** command cuts each selected object(s) by any intersecting object that is selected.

To slice all selected objects at their intersections:

- Multi-select the objects that you want to cut.
- From the **Create Object** menu, point to **Slice**, and then select **All in Selection**. All selected objects are cut at their intersections.

Angle

Window: ModelSpace

Menu: Tools | Measure

Action: Executes command

Usage: The **Angle** command calculates the angle between the centerlines and/or planes of the selected objects.

To measure the angle between objects:

- Pick the objects you want to measure.
- From the **Tools** menu, point to **Measure**, and then select **Angle**. The measurement is represented graphically. The measurement results are also displayed in the **Output** box if it is enabled.
 - If multiple points are being measured, measurements are taken from the first picked object to

all other picked objects, and the results are displayed in the order in which they were picked.

Note: When measuring the angles to the faces of extruded patches, the measurement is made from/to the plane of the picked face rather than the centerline of the extrusion.

Animate...

Window: ModelSpace

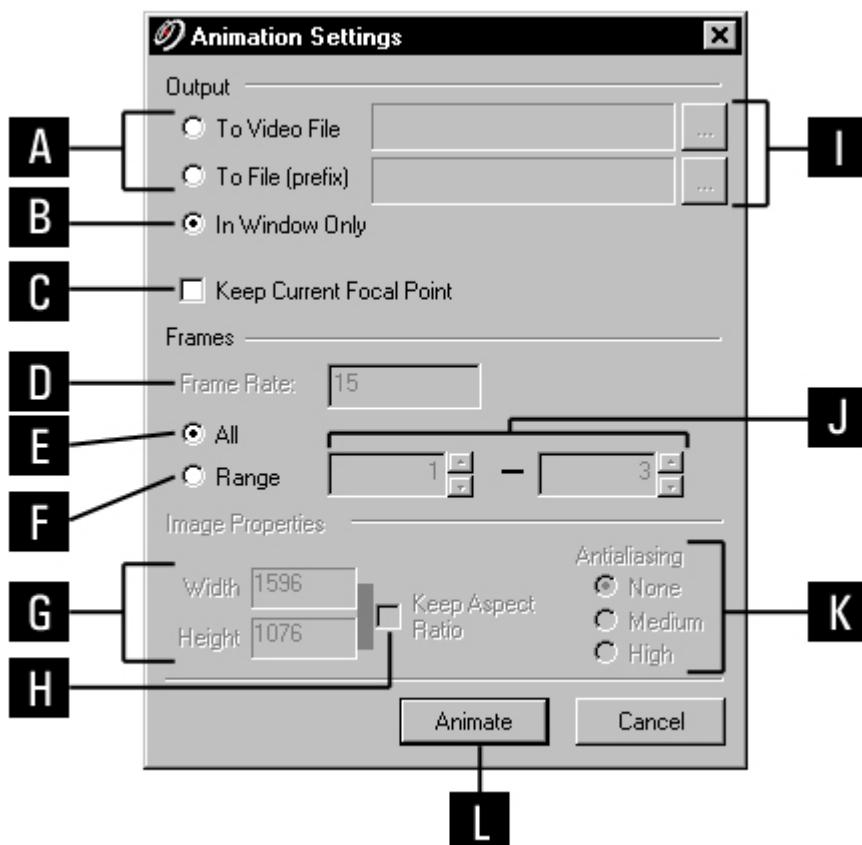
Menu: Tools | Animation

Action: Opens dialog

Usage: The **Animation Settings** dialog is used to specify animation output options and generate animation frames.

- For an overview of the animation process, see the [Animation](#) entry in the *Modeling* chapter.

Animation Settings Dialog



- A. Output frames to the designated file folder, using the indicated file name prefix.
- B. Preview the animation without generating frames.

- C. Override viewpoints designated by individual Camera objects and maintain the current focal point throughout the animation.
- D. Enter the frame rate.
- E. Generate or preview all frames.
- F. Generate or preview a specified range of frames.
- G. Enter a range of frames to generate or preview.
- H. Enter the width and height of the output images.
- I. Lock the aspect ratio for the size of the output images.
- J. Begin the preview or frame generation.
- K. Use the browser to select a folder for animation frames.
- L. Select **None**, **Medium**, or **High** antialiasing for the output images.

To generate animation frames:

1. From the **Animation** submenu, select **Animate**. The **Animation Settings** dialog appears.
2. Select **To File (prefix)**, and then click the **Browse** icon. The **Windows** browser appears.
3. Navigate to the desired file folder, select a file type, and enter a file name.
 - The file name is used as a prefix and each frame is given a sequentially numbered suffix when frames are generated.
4. Click **Save**.
5. Select **All** or designate a range of frames you wish to generate.
6. Adjust the output image size and select the desired amount of antialiasing.
7. Click **Animate**. A progress meter appears as the frames are generated and saved in the specified file folder.
 - To generate frames without the Camera objects or animation path, use the **Set Visibility** command to turn visibility OFF for those object types.
 - The level of detail for objects is kept at the highest setting by default. Edit the level of detail via the **View Object As** command.

To preview the animation in the current window:

1. From the **Animation** submenu, select **Animate**. The **Animation Settings** dialog appears.
2. Select **In Window Only**.
3. Select **All** or designate a range of frames you wish to generate.
4. Click **Animate**. The animation preview runs in the current **ModelSpace** window.

Animation Editor...

Window: ModelSpace

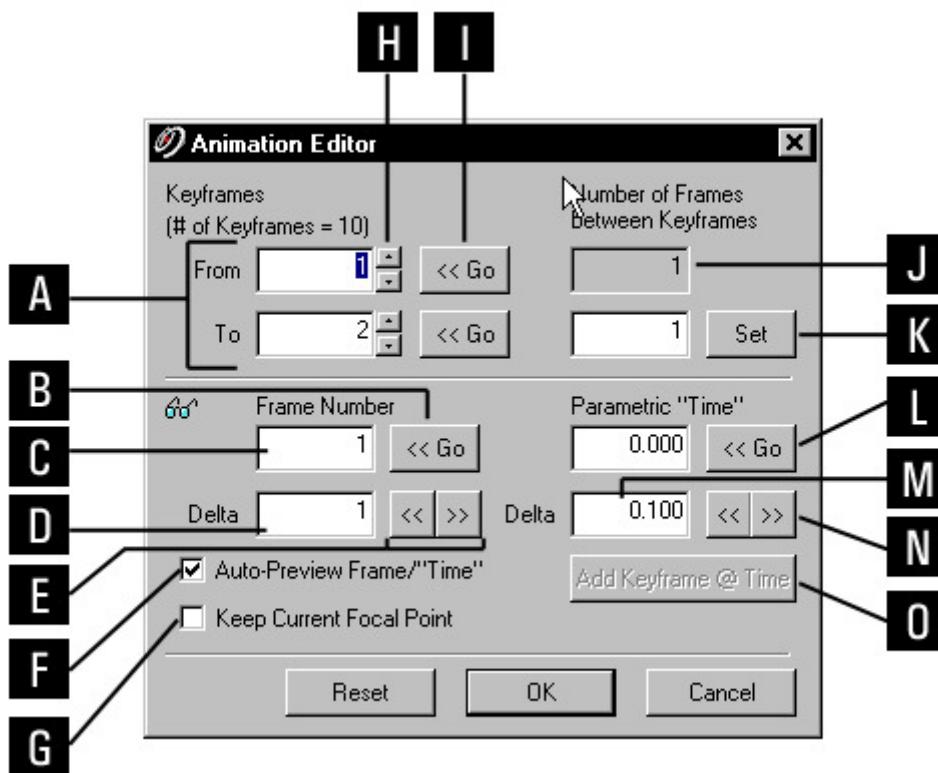
Menu: Tools | Animation

Action: Opens dialog

Usage: The **Animation Editor** dialog is used to set and add keyframes, specify the number of frames between keyframes, and navigate through the animation by frame or parametric time.

- For an overview of the animation process, see the [Animation](#) entry in the *Modeling* chapter.

Animation Editor Dialog



- Selected keyframes.
- Jump to the designated frame.
- Current frame.
- Enter a value for the increment to move forward/backward by frames.
- Jump forward/backward by the specified delta in frames.

F. Select to move the viewpoint each time the time or frame number change. When unselected, a current frame/time indicator moves along the animation path, but the viewpoint does not move.

G. Use the current focal point for each frame regardless of the view points of the keyframe Camera objects.

H. Cycle through keyframes.

I. Jump to the selected keyframe.

J. Current number of frames between the **From** and **To** keyframes

K. Enter a value for the number of frames inserted between the **From** and **To** keyframes.

L. Jump to the designated time.

M. Enter a value for the increment to move forward/backward in time.

N. Jump forward/backward by the specified time delta.

O. Add a keyframe at the designated time.

To edit animations:

1. From the **Animation** submenu, select **Animation Editor**. The **Animation Editor** dialog appears.
2. View keyframes and adjust viewpoints.
 - To jump to a specific keyframe, enter the number of the keyframe in the **To** or **From** field, and then click **Go**.
 - To adjust the viewpoint of a keyframe, select the Camera object and use its handles to edit its orientation.
 - To maintain the current focal point throughout the animation, select **Keep Current Focal Point**.
3. Add frames between keyframes.
 - To add frames between keyframes, enter the numbers of the two keyframes in the **To** and **From** fields, and then enter a value in the **Number of Frames Between Keyframes** field and click **Set**. Any existing frames or keyframes between the two selected keyframes are removed.
4. Navigate through/preview the animation.
 - To navigate through the animation, select **Auto-Preview Frame/Time**, and then click the left/right arrow icons under either **Frame Number** or **Parametric Time**. The viewpoint is moved forward/back through the animation by the increment of time or frames specified in the appropriate **Delta** field.
 - When **Auto-Preview Frame/Time** is not selected, a current frame/time indicator moves along the animation path, but the viewpoint does not move when the time or frame arrows are pressed.

5. Add additional keyframes if needed.
- To add a keyframe, navigate to the desired location on the animation path, and then click **Add Keyframe @ Time**. A new keyframe is inserted.
6. When finished, click **OK**. The changes are saved to the database, and the **Animation Editor** dialog closes. The animation output process may now be started via the **Animate...** command.

Annotations

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Annotations** displays submenu commands used add, edit, and delete annotations, custom annotations, feature codes, and camera images.

The following commands and dialogs are available from the **Annotations** submenu:

[Add/Edit Annotations...](#), [Add/Edit Custom Annotations...](#), [Show Annotations](#), [Show Annotation Keys](#), [Add/Edit Camera Images...](#), [Add/Edit Feature Code...](#)

- For **Annotation** submenu command details, see their individual entries.

Appearance

Window: ModelSpace

Menu: Edit Object

Action: Displays submenu

Usage: Pointing to **Appearance** displays submenu commands used to adjust the object's display properties.

The following commands, dialogs, and submenus are available from the **Appearance** submenu:

[Edit Color/Material...](#), [Add/Edit Colors](#), [Add/Edit Materials...](#), [Randomize Color of Visible Objects](#) (changes to selected objects if any objects are selected), [Create Objects with Random Colors](#), [Apply Color Map](#), [Edit Color Map](#), [Global Color Map](#), and [Edit Global Color Map](#).

Append to Alignment

Window: ModelSpace

Menu: Tools | Alignment and Section

Action: Executes command

Usage: The **Append to Alignment** command adds a line, polyline, or arc to the end of an existing alignment.

- For more information about alignments and the sections process, see the [Sections Manager](#) chapter.

To append to an alignment:

- Select the existing alignment to which you want to append.
- Multi-select the line, polyline, or arc you want to append to the selected alignment.
- From the **Tools** menu, point to **Alignment and Section**, and then select **Append to Alignment**. The new segment is added to the alignment and the station numbers from the original alignment are continued.

Apply Color Map

Window: ModelSpace

Menu: Edit Object | Appearance

Action: Displays submenu

Usage: Pointing to **Apply Color Map** displays submenu commands used to adjust color map styles to the selected point cloud or mesh.

- For an overview of color mapping in *Cyclone*, see the [Color Mapping](#) entry in the *Modeling* chapter.

Apply Color Map Submenu Commands

IMAGE TEXTURE MAP

Draws the points/vertices using the per-point color, if available. If not, it applies the hue-based color map or a single color if there are no intensities. In addition, any active texture maps are then drawn over the other colors.

COLORS FROM SCANNER

Draws the points/vertices using the per-point color, if available. If not, it applies the hue-based color map or a single color if there are no intensities.

INTENSITY MAP MULTI-HUE

Applies a multi-hue (red to blue) color map.

INTENSITY MAP GRayscale

Applies a grayscale color map.

ELEVATION MAP

Applies a color map based on elevation in the current UCS.

SINGLE COLOR

Applies a single color to all points.

Apply Multilmage

Window: Navigator

Menu: Context Menu

Action: Executes command

Usage: The **Apply Multilmage** command colors the Scans under the ScanWorld containing the selected Multilmage using the colors from the selected Multilmage.

To apply the colors from a Multilmage to the ScanWorld's Scans:

1. In the **Navigator** window, select the Multilmage that will provide the colors.
2. From the right-click context menu, select **Apply Multilmage**. The Scans under the ScanWorld containing the selected Multilmage are re-colored using the colors from the Multilmage.

Arc (3 Points on Arc)

Window: ModelSpace

Menu: Create Object | From Pick Points

Action: Executes command

Usage: The **Arc (3 Points on Arc)** command creates a circular arc through three picked points.

To create an arc:

1. Multi-select three non-colinear points to define an arc.
 - The resulting circular arc passes through all three points in the order in which they are picked. The first and last pick point serving as the endpoints.
2. From the **Create Object** menu, point to **From Pick Points**, and then select **Arc (3 Points on Arc)**. The arc is created.

Arc (Start, Center, End)

Window: ModelSpace

Menu: Create Object | From Pick Points

Action: Executes command

Usage: The **Arc (Start, Center, End)** command creates a circular arc through three picked points.

To create an arc:

1. Multi-select three points to define an arc.
 - The first and third pick points serve as the endpoints and must be equidistant from the second.
2. From the **Create Object** menu, point to **From Pick Points**, and then select **Arc (Start, Center, End)**. The arc is created.

Atmosphere

Window: Scan Control

Menu: Window

Action: Expands Tab

Usage: The **Atmosphere** command expands the **Atmosphere** tab in the Scanner Control Panel.

- Environmental conditions (ambient temperature and atmospheric pressure) can affect the range computations by a small amount. Although differences are minuscule over the ranges and environmental tolerances supported by the HDS3000, ScanStation, and ScanStation 2, controls are provided to compensate for such atmospheric conditions.
- The default values should be acceptable under most conditions.

Atmosphere Tab Options

AMBIENT TEMPERATURE

Enter the ambient temperature.

ATMOSPHERIC PRESSURE

Enter the atmospheric pressure.

PPM TOTAL

ppm Total is calculated automatically. To enter a value manually, select the radio button to the left of the field and enter the value in the field.

Auto-Add Cloud Constraints

Window: Registration

Menu: Cloud Constraint

Action: Executes command

Usage: The **Auto-Add Cloud Constraints** command automatically creates cloud constraints for all pairs of ScanWorlds that have overlapping point clouds, based on the current registration.

- The existing registration should be good enough to arrange the clouds to be constrained so their corresponding surfaces are close to each other.

To automatically add cloud constraints based on the current registration:

1. Set up the registration so that the clouds are roughly registered.
 - For example, make use of target- or geometry-based registration, Station Data, etc.
2. From the **Cloud Constraint** menu, select **Auto-Add Cloud Constraints**. The Registration creates a cloud constraint for each pair of ScanWorlds that have overlapping point clouds and do not yet have a cloud constraint. The Registration automatically optimizes the cloud

constraints.

This can be a time-consuming process since each pair of ScanWorlds must be checked for overlapping point clouds. To save time, the user may select one or more ScanWorlds and select **Auto-Add Cloud Constraints**. The command will only check ScanWorlds that overlap with the selected ScanWorlds. For example, if a ScanWorld is added to a registration and the user wants to automatically add cloud constraints involving only that ScanWorld, the user would select that ScanWorld and select **Auto-Add Cloud Constraints**, and the search would ignore other possible ScanWorld pairings that do not involve that ScanWorld.

Another option for the user is to use the **Add Cloud Constraint** command, which can be used to create a cloud constraint between a selected pair of ScanWorlds. This command does not require initial alignment pick hints for the cloud constraint if the ScanWorlds are already aligned using other constraints.

Auto-Add Constraints

Window: Registration

Menu: Constraint

Action: Executes command

Usage: The **Auto-Add Constraints** command adds constraints between user-specified ScanWorlds by matching registration labels and geometric shapes.

- Cloud Constraints are not created by this command (see the [Add Cloud Constraint command](#) for more information).
- Geometric objects that are potential constraints must be manually added to the ControlSpace of each ScanWorld before they will be considered by the **Auto-Add Constraints** command. See the [Copy to ControlSpace command](#) for more information.
- The **Auto-Add Constraints** process uses any currently existing constraints, if any, then searches the ControlSpaces for objects with matching registration labels. It then attempts to infer additional constraints from matching pairs of geometry in the involved ControlSpaces that are mutually consistent (that would be accurate given the pre-existing constraints and registration label constraint pairs). In the case that several possible sets of pairs exist, it chooses the set which aligns the largest number of objects.
- For more information on the registration process and ControlSpaces, see the [Registration](#) entry in the *Getting Started* chapter.
- For instructions on adding a registration label, see the [Add/Edit Registration Label command](#).

Auto-Pick Points

Window: ModelSpace

Menu: Selection

Action: Toggles menu item ON/OFF

Usage: The **Auto-Pick Points** command toggles the auto-picking function for the current ModelSpace viewer.

- The auto-picking function allows the user to specify a significant point (i.e., the end of a cylinder) that is automatically picked when an object of that object type is selected. When using auto-pick, the specified point is always the only picked point regardless of how the object was selected (i.e., selected by fence, or automatically selected after creation).

To use the Auto-Pick function:

- From **Selection** menu, toggle **Auto-Pick Points** ON.
- By default, when an object is picked directly or a default pick is supplied by a command, the **Auto-Pick** setting is ignored in favor of the specified pick point. To override the direct or default pick point, select **Use Auto Pick** (instead of **Ignore Auto-Pick**) from **ModelSpace** tab of the **Edit Preferences** dialog.
- From the **Edit** menu, select **Object Preferences**. The **Object Preferences** dialog appears.
- From the **Object Type** list, select the type of object with which you want to use auto-picking.
- Select the **Automatically pick point on selection** check box.
- Make a selection from the drop-down list, which displays points that can be automatically picked for the selected object type.
- Click **OK**. Auto-picking is enabled for the selected object type.

Auto-Update

Window: Registration

Menu: Registration

Action: Toggles menu item ON/OFF

Usage: The **Auto-Update** command toggles whether or not the Registration is recomputed after each change of a constraint or component ScanWorld.

AutoCAD...

Window: ModelSpace

Menu: File | Launch

Action: Executes command

Usage: The **AutoCAD** command launches the current ModelSpace View in AutoCADE2 using *Cyclone Object Exchange* (COE).

- When this command is initiated, objects in the current ModelSpace View are first transferred to a COE file, which is then loaded into AutoCAD. See COE for AutoCAD for information on working with COE files.
- Before executing this command, it is important to verify that AutoCAD and COE Data Transfer for AutoCAD are properly installed on your workstation.

To launch AutoCAD from a ModelSpace View:

1. From the **File** menu, point to **Launch**, and then select **AutoCAD**.
 - You may be prompted to locate the AutoCAD executable (**ACAD.EXE**). If this occurs, navigate to the folder containing the file and select it.
2. If you want to open an existing file, navigate to the appropriate folder and select the desired file; if you are opening a new drawing, enter a new file name.
3. Click **Open**. AutoCAD is launched.
 - You can open a temporary file by clicking **CANCEL**; a file will be created in your system's **TEMP** directory. To save any work done in a temporary file, be sure to save your drawing from AutoCAD.

Back

Window: ControlSpace, ModelSpace, Registration, Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)

Menu: Point Cloud Rendering

Action: Toggles menu item ON

Usage: The **Back** command determines the active side of the estimated surfaces in point clouds.

- The command affects the **Shaded** and **One-Sided Point Cloud Rendering** modes.

Back Angle

Window: ModelSpace

Menu: Tools | Measure

Action: Executes command

Usage: The **Back-Angle** command calculates the back angle between the centerlines and/or planes of the selected objects. The “back angle” added to the shorter angle measured by the **Angle** command forms a full circle.

To measure the back angle between objects:

1. Pick the objects you want to measure.
2. From the **Tools** menu, point to **Measure**, and then select **Back Angle**. The measurement is represented graphically. The measurement results are also displayed in the **Output** box if it is enabled.
 - If multiple points are being measured, measurements are taken from the first picked object to

all other picked objects, and the results are displayed in the order in which they were picked.

Note: When measuring the back angles to the faces of extruded patches, the measurement is made from/to the plane of the picked face rather than the centerline of the extrusion.

Batch Registration

Window: **Navigator**

Menu: **Tools**

Action: Executes command

Usage: The **Batch Registration** command executes the **Register** command for each selected Registration object and saves a log file to a specified folder.

- Batch registration is used for performing multiple registrations without user intervention.

To perform batch registration:

- From the **Navigator** window, select the Registration objects to be registered.
- The Registration objects should be in a state where you could perform the final **Register** command. The ScanWorlds should be added and the constraints should be created and edited as needed.
- From the **Tools** menu, select **Batch Registration**. A browser prompt appears.
- Use the browser to select a location for the log file.
 - To execute the batch registration without producing a log file, clear the **Save log file to** check box.
- When the registrations are complete, the **Batch Registration Logs** dialog appears. Registration logs can be viewed, printed, and saved.

Below Ref Plane (Mesh)

Window: **ModelSpace**

Menu: **Tools | Measure | Mesh Volume**

Action: Opens dialog

Usage: The **Below Ref Plane** command is used to measure “fill” volume of a selected mesh relative to the active Reference Plane.

- The **Below Ref Plane** command functions identically to the **Above Ref Plane...**, except that it is used to measure “fill” volume from the active Reference Plane down to the selected mesh.

Breaklines

Window: ModelSpace

Menu: Tools | Mesh

Action: Displays submenu

Usage: Pointing to **Breaklines** displays submenu commands used to manipulate breaklines with regards to mesh objects or vice versa.

The following commands are available from the **Breaklines** submenu:

[Project Polyline to TIN](#), [Extend TIN to Polyline](#), [Clear Breaklines](#).

Black/White Target

Window: ModelSpace

Menu: Create Object | Fit to Cloud

Action: Opens dialog

Usage: The **Black/White Target** command fits a vertex at the center of a tie-point. When a tie-point is created, it is added automatically to the ControlSpace as well.

To fit an HDS tie-point:

1. Segment the point cloud to which you want to fit a tie-point from the surrounding cloud points (if any).
2. Pick near the intersection of the black and white quadrants of the point cloud subset to which you want to fit a tie-point.
3. From the **Create Object** menu, point to **Fit to Cloud**, and then select [Black/White Target](#). The **Tie-Point ID** dialog appears.
4. Enter an identifier and a comment in the appropriate fields, and then click **OK**. A vertex is fit to the tie-point.

Bubble Level...

Window: Scan Control (ScanStation, ScanStation 2, HDS6000)

Menu: Scanner

Action: Opens dialog

Usage: The **Bubble Level...** command is used to display a visualization of the current tilt of the scanner.

To display the bubble level:

1. From the **Scanner** menu, select **Bubble Level....** The **Bubble Level** dialog appears.

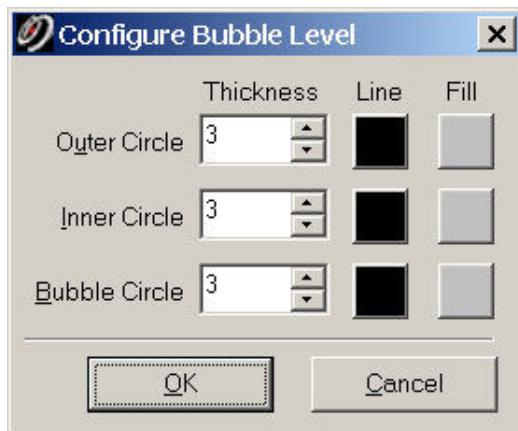
To re-calibrate the bubble level (HDS6000):

- From the **Bubble Level** dialog, click the **Find Zero** button. The scanner takes several readings and computes its actual offset from level.

To configure the appearance of the bubble level:

- From the **Bubble Level** dialog, click the **Configure...** button. The **Configure Bubble Level** dialog appears.

Configure Bubble Level Dialog



A. Outer Circle: Specify the line **Thickness**, **Line** color, and **Fill** color of the circle that represents the outer fixed ring of the display.

B. Inner Circle: Specify the line **Thickness**, **Line** color, and **Fill** color of the circle that represents the inner fixed ring of the display.

C. Bubble Circle: Specify the line **Thickness**, **Line** color, and **Fill** color of the circle that represents the current position of the virtual bubble.

D. OK: Click to accept the settings and close the **Configure Bubble Level** dialog.

E. Cancel: Click to close the **Configure Bubble Level** dialog without keeping any changes made to the settings.

by Last Selection

Window: ModelSpace

Menu: Create Object | Slice

Action: Executes command

Usage: The **by Last Selection** command cuts selected object(s) by the patch selected last.

To slice objects using a selected patch:

1. Multi-select the objects that you want to cut, and then multi-select the patch with which you want to cut them.
 - The last object selected must be a patch, which is then used to cut all of the other selected objects.
2. From the **Create Object** menu, point to **Slice**, and then select **by Last Selection**. The objects are cut by the plane of the last patch selected.

by Ref Plane

Window: ModelSpace

Menu: Create Object | Slice

Action: Executes command

Usage: The **by Ref Plane** command cuts selected object(s) by the active Reference Plane.

To slice objects using the Reference Plane:

1. Multi-select the objects that you want to cut.
2. From the **Create Object** menu, point to **Slice**, and then select **by Ref Plane**. The objects are cut by the Reference Plane.

Calibrate Camera...

Window: Scan Control

Menu: Scanner Control | Camera Calibration

Action: Opens dialog

Usage: The **Calibrate Camera** dialog is used to execute camera calibration for the HDS2500.

- For more information on camera calibration, see the [Camera Calibration](#) entry in the *Scanning with the HDS2500* chapter.

To calibrate the camera:

1. Place the scanner at a distance of roughly 3 to 5 meters from the wall as described in the *Camera Calibration* entry in the *Scanning with the HDS2500* chapter, aimed at the center of the wall. Be sure that you have room to move the scanner back at least 3 to 5 more meters for the second position.
- The view from the scanner to the wall should be unobstructed, and the wall should be a light, diffuse color (for example, white or beige); the process may not work with red, black, or shiny surfaces).
- Using a wall large enough to cover the entire scanner field of view at 10 meters (about 7

- meters on each side) is recommended.
2. Open the **Scan Control** viewer and connect to the scanner. See the *Connect* entry in this chapter.
 3. From the **Scanner Control** menu, point to **Camera Calibration** and select **Calibrate Camera....** The **Camera Calibration** dialog appears.
 4. Click **Calibrate** (do not move the scanner). When the first step is complete, the dialog reappears.
 5. Move the scanner (without disconnecting it) back at least 5 more meters from the wall.
 - If you have room to move it back to 15 meters from the wall, do so, although 10 meters from the wall will also work (although if you use 10 meters in this step, you should place it 5 meters from the wall in step 1).
 6. Click **Continue**.
 7. A dialog notifies you when a successful calibration is complete; the scanner stores the calibration parameters, used for future scans and images.
 8. Test the calibration with a few scans. Alternately, use the **Verify Calibration** command, which automatically tests the accuracy of the calibration, using the scene in front of the scanner.

Note: It is possible for the external camera in the 2500 to drift out of calibration over time.

If issues with the calibration are noted, the camera should be recalibrated.

Cancel Drawing

Window: **ModelSpace**

Menu: **Tools | Drawing**

Action: Executes command

Usage: The **Cancel Drawing** command discards the current 2D line drawing.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

Cancel Scan

Window: **Scan Control**

Menu: **Scanner Control**

Action: Executes command

Usage: The **Cancel Scan** command ends the current scan operation.

- Any points collected before the scan's cancellation are inserted into the ScanWorld and destination ModelSpace and made available for modeling.

Center on Fence

Window: **Scan Control**

Menu: **Scanner Control**

Action: Executes command

Usage: The **Center on Fence** command centers the viewpoint on the current fence.

Centerline Length

Window: ModelSpace

Menu: Tools | Measure

Action: Executes command

Usage: The **Centerline Length** command calculates and displays the length of the centerline of the selected object(s).

To measure centerlines:

1. Select the object(s) you want to measure.
2. From the **Tools** menu, point to **Measure**, and then select **Centerline Length**. The measurement is displayed in the Output Box.

Check Calibration

Window: Scan Control

Menu: Scanner

Action: Executes command

Usage: The **Check Calibration** command is used to scan selected targets to judge if the scanner requires calibration.

- To check the calibration of a connected scanner:

1. Pick one or more black/white targets in the Scan Control viewer or in the attached ModelSpace viewer.
2. From the **Scanner** menu, select **Check Calibration**.
3. Each target is acquired from each side of the scanner and the distance between each is measured.
4. The **Calibration Check Results** dialog displays the measurement results. The distances should be within acceptable values.

Check Fit Tolerances

Window: ModelSpace

Menu: Create Object | Fit to Cloud Options

Action: Toggles menu item ON/OFF

Usage: The **Check Fit Tolerances** command enables/disables a warning message that appears if a cloud fit is beyond one or more of the tolerance limits.

- In order to use fit constraints, the **Fit Tolerances** box must also be selected and tolerance values must be specified within the **Object Preferences** dialog for the type of object being fit. See the [Object Preferences](#) command.

- For an overview of fitting objects in *Cyclone*, see the [Modeling](#) entry in the *Getting Started* chapter.

Check/Re-Level HDS6000...

Window: Scan Control (HDS6000)

Menu: Scanner

Action: Opens dialog

Usage: The **Check/Re-Level HDS6000...** command opens the **Check/Re-Level HDS6000** dialog, which displays the angular “tilt” of the scanner.

To check or re-level the scanner:

1. From the **Scanner** menu, select **Check/Re-Level HDS6000....** The **Check/Re-Level HDS6000** dialog appears.
2. The **TiltL** and **TiltR** values indicate the degree of inclination from horizontal. When both are 0, the ScanStation/ScanStation 2 is perfectly level. As long as the **TiltL** and **TiltR** are **IN RANGE**, the HDS6000 can sense if the scanner inclination changes. When the **Tilt Sensor Status** is **OUT OF RANGE**, only the **Cancel** button is enabled.
 - If the tilt sensor state changes significantly (e.g., a large amount of drift, manual adjustment of the tribrach, or out-of-range), for data integrity, the current ScanWorld is automatically advanced to the next ScanWorld. You may re-select the previous ScanWorld for additional work, but be sure that the data are consistent.

Check/Re-Level ScanStation...

Window: Scan Control (ScanStation, ScanStation 2)

Menu: Scanner

Action: Opens dialog

Usage: The **Check/Re-Level ScanStation...** command opens the **Check/Re-Level ScanStation** dialog, which displays the angular “tilt” of the scanner.

To check or re-level the scanner:

1. From the **Scanner** menu, select **Check/Re-Level ScanStation....** The **Check/Re-Level ScanStation** dialog appears.
2. The **TiltL** and **TiltR** values indicate the degree of inclination from horizontal. When both are 0, the ScanStation/ScanStation 2 is perfectly level. As long as the **TiltL** and **TiltR** are **IN RANGE**, the ScanStation/ScanStation 2 will compensate for the scanner’s tilt. When the **Dual-Axis Compensator Status** is **OUT OF RANGE**, only the **Cancel** button is enabled.
 - If the dual-axis compensator state changes significantly (e.g., a large amount of drift, manual

adjustment of the tribrach, or out-of-range), for data integrity, the current ScanWorld is automatically advanced to the next ScanWorld. You may re-select the previous ScanWorld for additional work, but be sure that the data are consistent.

Circular Fence Mode

Window: ModelSpace

Menu: Edit | Modes

Action: Executes command

Usage: The **Circular Fence Mode** command enters **Circular Fence** mode and selects the **Circular Fence** mode cursor.

- For more information on working with fences, see the [Using a Fence](#) entry in the *Modeling* chapter.

To draw a circular fence:

1. From the **Edit** menu, point to **Modes**, and then select **Circular Fence Mode**. The **Circular Fence** cursor appears.
2. Drag the cursor around the area that you want to fence.
 - The newly drawn fence can adjusted by dragging its sides or vertices with the **Circular Fence** cursor.
 - To clear the newly drawn fence, select **Clear** from the **Fence** submenu.

Clear

Window: ModelSpace

Menu: Edit | Fence

Action: Executes command

Usage: The **Clear** command removes a fence from the screen but does not affect its contents.

- For more information, see the [Using a Fence](#) entry in the *Modeling* chapter.

Clear Breaklines

Window: ModelSpace

Menu: Tools | Mesh | Breaklines

Action: Executes command

Usage: The **Clear Breaklines** command removes breaklines from the selected mesh object.

- If an individual breakline is picked, only that breakline is removed.

- See the [*Project Polyline to TIN*](#) command for more information on creating a breakline.

Clearances

Window: ModelSpace

Menu: Tools | Measure

Action: Opens dialog

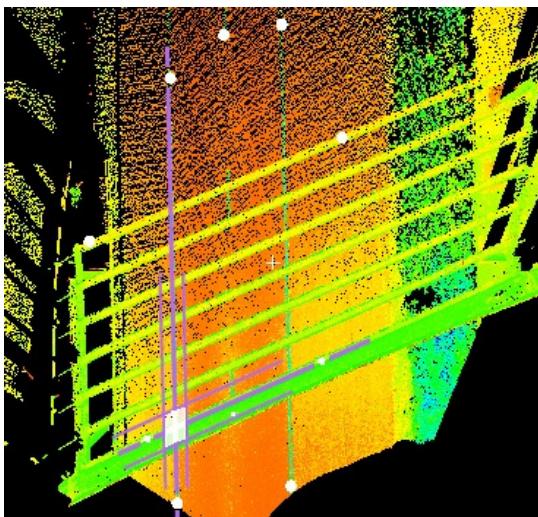
Usage: The **Clearances** command is used to measure the vertical and horizontal clearances to overhead and adjacent structures in a roadway/overpass/tunnel setting.

- This topic describes a typical workflow. Your process may differ.
- The **Clearances** command requires:
 - that the points from transient surfaces (e.g., passing automobiles) first be removed.
 - that the road surface.
- This command requires selection of at least 2 points on the road surface to establish the travel lane(s) and 2 or 4 points on the lowest edge(s) of the overhead structure (bridge, sign structure). The points must be selected in a specific order.

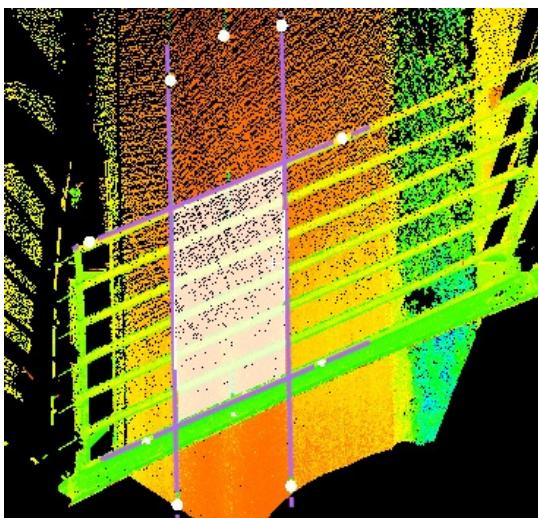
The pick points on the road surface must be along the travel lanes (edge of pavement to edge of pavement) to compute the minimum vertical clearance for traveling vehicles.

- The **Lane Measurement Tolerance** (< 1.0 m) defines the width of a strip of points centered along each travel lane border. The **Overhang Measurement Tolerance** (<1.0 m) is 2x the largest horizontal distance that the points picked on the overhead surface can be from the lowest edge. The intersection of the **Lane** and **Overhead** tolerances determines the points on the road surface and the points on the overhead surface used to compute the vertical distance at each crossing.
- Additionally, with 2 points picked on each of the approach and departure sides of the overhead structure, the lines defined by each successive pair of picks enclose the area in which the minimum vertical clearance is measured.

Area used for vertical distance at each crossing



Area used for minimum vertical distance



To measure vertical clearances:

1. Shift+Pick two points on your lower surface (e.g., a road surface). Pick the points such that the second pick point indicates the forward direction of travel from the first pick point.
 - Shift+Pick additional pairs of points wherever a vertical clearance is to be measured (e.g., on the edge of pavement or along a lane stripe).
 - Each pair should be roughly parallel to the line described by the first two pick points.
 - Each additional pair should be to the right of the previous pairs; the first pair of pick points should be on the left-most side.
 - For purposes of this topic, the line described by each pair of pick points on the lower surface is a lane line.
2. Shift+Pick two points on your upper surface (e.g., an overpass or tunnel). Pick the points such that the two pick points are close to the edge of the upper surface.

- (Optional) Shift+Pick an additional pair of points on the opposite end of the upper surface.
 - The edges indicated on the upper surface are referred to as Approach and Departure. The Approach is the first edge encountered, based on the direction of travel. For purposes of this topic, these edges are on an overhang.
3. From the **Tools** menu, point to **Measure**, and then select **Clearances....** The **Measure Clearances** dialog appears.
 4. Adjust the labeling that will be applied to the measurements and sections. Each measurement and section generated by this command has a distinct name.
 - Enter the **Base Label** for the measurement names. For example, if the **Base Label** is "A", the first lane line is "A", the second is "B", the third is "C", etc.
 - By default, each vertical measurement is named by appending the **Base Label** onto "A" (for approach) or "D" (for departure), incrementing the **Base Label** for each additional lane line.
 - Click ... next to the **Base Label** to invoke the **Detailed Labeling** dialog for custom labeling.
 - Click **Update** to apply changes to the labeling to the existing measurements and sections.
 5. Adjust the **Measurement Tolerances** for the **Vertical Clearance** measurements.
 - The points within the **Lane** tolerance of the lane lines are used in the lane-to-overhang measurements.
 - The points within the **Overhang** tolerance of the overhang edges are used in the lane-to-overhang measurements.
 6. Click **Measure Vertical** to measure the vertical clearances from the lower surface to the upper surface. Two sets of vertical measurements are generated:
 - The vertical distance between each lane line and overhang edge, where they cross in plan view.
 - The minimum vertical clearance, where the points on the upper surface come closest to the lower surface. Only the points within the boundary defined by the picked points are used.
 7. (Optional) Measure the horizontal clearances.
 8. Click **Create Lines** to create a line segment for each clearance measurement. These lines are available for manipulation, export, etc.
 9. Click **Report** to display the **Clearance Measurement Report** dialog.
 10. Close the dialog. Click **OK** to keep the measurements and sections, or **Cancel** to discard the measurements and sections.

To measure horizontal clearances:

1. Before measuring horizontal clearances, the vertical clearances must first be measured. Measure the vertical clearances. (See earlier.)
2. Specify the elevation at which the horizontal clearances will be measured in **at Elevation**.
3. Specify the **Region Tolerance**. All points within this vertical distance of **at Elevation** will be used to measure the horizontal clearances.
4. Select the measurement options.
 - Select **Left Clearance** to measure the minimum distance to a point to the left of the left-most lane.
 - Select **Right Clearance** to measure the minimum distance to a point to the right of the right-most lane.
 - Select **Perpendicular** to measure distances perpendicular to each lane. When not selected, the distances are measured parallel to the edge on the upper surface.
5. Click the **Measure Horizontal** button to measure the horizontal clearances between the

lanes, and optionally the left and right lateral clearances. Up to three sets of horizontal measurements are generated:

- The horizontal distance between each lane line.
- (Optional) The minimum horizontal clearance to the left of the left-most lane line, at the specified elevation.
- (Optional) The minimum horizontal clearance to the right of the left-most lane line, at the specified elevation.

Clear Calibration Info

Window: Scan Control, Image Viewer

Menu: Scan Control: Scanner Control | Camera Calibration

Image Viewer: Image | Calibration

Action: Executes command

Usage: The **Clear Calibration Info** command is used to remove camera calibration data from the current image if the calibration seems to be incorrect.

- For a general overview of camera calibration, see the [Camera Calibration](#) entry in the *Scanning with the HDS2500* chapter.

Note: Clearing the calibration disables the ability to texture map the current image onto point clouds or meshes, and also removes the distortion correction of the image when viewed in the **Scan Control** or **Image Viewer** windows.

Clear Image

Window: Scan Control

Menu: Edit

Action: Executes command

Usage: The **Clear Image** command removes the image from the viewing window in the **Scan Control** window.

Clear Locks

Window: ModelSpace

Menu: Viewpoint | View Lock

Action: Executes command

Usage: The **Clear Locks** command turns OFF all **View Locks**.

Clear Path

Window: ModelSpace
Menu: Tools | Animation
Action: Executes command
Usage:

- For an overview of the animation process, see the [Animation](#) chapter.

Clear Target

Window: Scan Control
Menu: Scanner Control
Action: Executes command
Usage: The Clear Target command removes the current target box in the **Scan Control** window.

Clear Temporary Measurements

Window: ModelSpace
Menu: Tools | Measure
Action: Executes command
Usage: The Clear Temporary Measurements command deletes the last unsaved measurement taken.

- Subsequent measurements are not considered temporary when the **Save Measurements** command is toggled ON. Once saved, a measurement is no longer temporary and cannot be cleared using this command.

Clear Undo/Redo

Window: ModelSpace, Navigator, Registration, Scan Control, Image Viewer
Menu: Edit
Action: Executes command
Usage: The Clear Undo/Redo command clears all previous actions that are subject to **Undo** and **Redo** from memory.

- After the **Clear Undo/Redo** command has been used, the ability to **Undo** and **Redo** previous actions is lost for this viewer. Subsequent operations may still generate Undo and Redo events.

- **Clear Undo/Redo** can be used to free memory otherwise consumed by undo and redo events.

Close

Window: **ModelSpace, Image Viewer, Registration, Scan Control**

Menu: **File**

Action: Executes command

Usage: The **Close** command closes the current window.

- To exit *Cyclone*, select **Exit** from the **File** menu in the **Navigator** window.
- When closing a ModelSpace View window that was created via the **Copy to New ModelSpace** command in the **Launch** menu, you will be prompted whether to merge the ModelSpace View back into the original, and whether to keep or remove the launched ModelSpace.

Cloud Constraints Wizard...

Window: **Registration**

Menu: **Cloud Constraint**

Action: Opens dialog

Usage: The **Cloud Constraints Wizard** launches a series of wizard dialogs that guide the user through the process of creating cloud constraints.

- For more information on the cloud registration process, see the *Cloud Registration* entry in the *Registration* chapter.

This is the *recommended* method for creating cloud constraints. The Cloud Constraints Wizard guides you through the entire process of creating cloud constraints and can save considerable time compared to the other methods. First, specify which ScanWorld pairs are overlapping, and then the Wizard steps through the process of creating a cloud constraint for each specified pair of ScanWorlds. See the *Cloud Constraints Wizard* entry in the *Commands* chapter for detailed instructions. The Cloud Constraints Wizard offers the following advantages:

- It automatically copies all scan clouds (except HDS Target clouds) into the ControlSpace for the selected ScanWorlds.
- It automatically opens the ControlSpaces side-by-side in the Constraint viewers, and advances to the next ScanWorld pair after the user has picked corresponding points and pushed the Cloud Constraints Wizard's **Create Constraint** button.
- It provides a quick way to check for overlap between two ScanWorlds. Pressing the eyeglass button in the Cloud Constraint Wizard's grid control displays the corresponding ScanWorlds' ControlSpaces in the Constraint viewers.

To create cloud constraints via the Cloud Constraints Wizard:

1. Open a new or existing **Registration** window, and add the ScanWorlds that are to be registered.
 - The registration may already have target or object constraints.
2. From the **Cloud Constraint** menu, select **Cloud Constraints Wizard**. The first wizard page appears, displaying a matrix of all ScanWorlds that have been added to the registration.
 - An initial check mark indicates that the corresponding pair of ScanWorlds already has an existing Cloud Constraint.
3. Select the check boxes in the matrix to indicate that a Cloud Constraint should be created between the two ScanWorlds (if one does not already exist), and clear check boxes to remove any existing Cloud Constraints that are no longer needed from the Registration.
 - Click the **View** icon at the intersection of two ScanWorlds to see their ControlSpaces. If a ControlSpace has no point clouds or meshes, all non-target scans in the ScanWorld are automatically added to the ControlSpace.
 - To select all check boxes, click **Select All**.
 - To clear all check boxes, click **Deselect All**.
 - Click **Select Cycle** to select the check boxes between each ScanWorld and the next ScanWorld in the sequence (including the check box between the last and first ScanWorlds to complete the cycle).
 - Click **Cancel** to exit the **Cloud Constraints Wizard** without saving changes.
4. Click **Update**. The **Cloud Constraints Wizard** initiates the cloud constraint construction sequence for the selected ScanWorld pairs (pairs with existing cloud constraints are skipped). The ControlSpaces for the first selected ScanWorld pair are displayed in the Constraint viewers.
 - If the two ScanWorlds already have a cloud constraint, the pick points (if any) of the cloud constraint are shown.
 - 5. Multi-pick two or more matching points in each ControlSpace of the ScanWorld pair.
 - If the ScanWorlds are already registered, or the default ScanWorld coordinates are already specified in global coordinates, then no pick points are needed.
 - See the *Cloud Registration* section in the *Registration* chapter for information on picking good points.
 - To preview the point cloud alignment based on the current picks, click **Preview**. A temporary ModelSpace opens and displays the alignment of the ScanWorld pair's point cloud/meshes. Close the temporary ModelSpace view to return to the Cloud Constraint Wizard.
 - To move on to the next ScanWorld pair in the sequence without creating a cloud constraint, click **Next**.
 - To return to the previous ScanWorld pair in the sequence, click **Back**.
 - To return to the first **Cloud Constraints Wizard** page, click **Return**.
 - 6. Click **Constrain**. The constraint is created and the **Cloud Constraints Wizard** opens the next ScanWorld pair in the Constraint viewers.
 - 7. Repeat steps **5** and **6** for each selected ScanWorld pair as needed.

Note: To be used in global registration, cloud constraints must be optimized via the **Optimize Cloud Alignment** command.

Codirectional

Window: **ModelSpace**

Menu: **Edit Object | Align**

Action: Executes command

Usage: The **Codirectional** command aligns one or more selected objects so that they are parallel and facing in the same direction as a reference object.

To make objects codirectional:

1. Multi-select the objects you want to make codirectional.
 - The last object selected is the reference object (the one to which the others align).
2. From the **Edit Object** menu, point to **Align**, and then select **Codirectional**. The objects are rotated about their origins to align with their axes in the same direction as the reference object.

Coincident

Window: ModelSpace

Menu: Edit Object | Align

Action: Executes command

Usage: The **Coincident** command moves and aligns one or more objects to the corresponding axis and origin of a reference object.

To make objects coincident:

1. Multi-select the objects you want to make coincident.
 - The last object selected is the reference object (the one to which the others align).
2. From the **Edit Object** menu, point to **Align**, and then select **Coincident**. The selected objects are moved to align their origins and axes to the reference object.

Colinear

Window: ModelSpace

Menu: Edit Object | Align

Action: Executes command

Usage: The **Colinear** command moves and aligns one or more objects to the axis of a reference object.

To make objects colinear:

1. Multi-select the objects you want to make colinear.
 - The last object selected is the reference object (the one to which the others align).
2. From the **Edit Object** menu, point to **Align**, and then select **Colinear**. The axes of the selected objects are moved and aligned to the axis of the reference object.

Collapse

Window: ModelSpace

Menu: Tools | Exclusion Volume

Action: Executes command

Usage: The **Collapse** command resizes the selected exclusion volume to the minimum

required to contain its contents without changing its orientation.

To minimize the size of an exclusion volume:

1. Select the exclusion volume.
2. From the **Tools** menu, point to **Exclusion Volume**, and then select **Collapse**. The sides are moved in to contain the minimum volume without changing its orientation.

Collapse (Limit Box Manager)

Window: **Limit Box Manager**

Menu: **View**

Action: Executes command

Usage: The **Collapse** command resizes the selected limit box to the minimum volume required to contain the currently selected objects (if any) or to contain all objects (if there is no selection) without changing the orientation of the limit box..

Collapse Limit Box

Window: **ModelSpace**

Menu: **View | Collapse Limit Box**

Action: Executes command

Usage: The **Collapse Limit Box** command resizes the limit box to the minimum volume required to contain the currently selected objects (if any) or to contain all objects (if there is no selection) without changing the orientation of the limit box.

- To obtain the minimum possible limit box size regardless of limit box orientation, use the **Minimize Limit Box** command

Connect

Window: **Scan Control**

Menu: **Scanner**

Action: Executes command

Usage: The **Connect** command is used to connect to the selected scanner.

- To connect to a scanner: From the **Scanner** menu, select **Connect**. When a successful connection is made, the **Scanner Status** box at the bottom of the **Scan Control** window reads "Connected and Ready".

Connect Piping

Window: ModelSpace

Menu: Tools | Piping

Action: Executes command

Usage: The **Connect Piping** command finds and establishes any valid connections among selected objects.

To connect piping:

1. Select objects among which you want to establish a piping connection.
 - *Cyclone supports* two types of connectivity: linear (end-to-end) and branching. An end-to-end connection may exist between two opposing piping components that share a common endpoint. The diameters of the components need not match. A branching connection may exist where one piping component's endpoint sits on the centerline of a pipe. The branch's endpoint must lie within the extents of the run. The angle between the two centerlines does not need to be perpendicular.
 - To set preferences that determine whether or not a connection is made, see the **Piping: Max. Offset** and **Piping: Max. Angle** settings in the **Modeling** tab of the *Preferences* entry in this chapter.
2. From the **Tools** menu, point to **Piping** and then select **Connect Piping**. Any valid connections are established, and selected objects that are not connected with any other selected object are deselected.
 - Once connected, two piping components can only be disconnected by executing the **Disconnect Piping (Shared)** or **Disconnect Piping (All)** commands or by changing the geometry of object such that the connection is automatically broken (because the connection now exceeds the current **Piping: Max. Offset** and **Piping: Max. Angle** settings).

Note: The **Connect Piping** command may take longer to finish depending on the number of objects selected. Selecting more objects may greatly increase the time required to find the shared connections.

Constrain Motion to

Window: ModelSpace

Menu: Edit Object | Handles

Action: Displays submenu

Usage: Pointing to **Constrain Motion to** displays submenu commands to select the plane or axis along which objects can be moved using their handles.

The following commands are available from the **Constrain Motion to** submenu:

User Coordinate System, Object Coordinate System, X Axis, Y Axis, Z Axis, Screen Plane (perpendicular to the current viewpoint), **X-Y Plane, X-Z Plane, Y-Z Plane**

- This command limits handle motion to be along a specified axis or plane. Constraining the object handles to move only along a certain plane or axis prevents you from moving the object in an unwanted direction.
- Some handles may not be able to move according to the constraints (e.g., rotation and other custom handles).
- For more information on using handles, see the [Handles](#) entry in the *Modeling* chapter.

Contours

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Contours** displays submenu commands used to create and manipulate contour lines.

The following commands are available from the **Contours** submenu:

Create..., Decimate Contours, Delete Line.

- For **Contours** submenu command functionality, see the individual command entries.

Contours to Mesh Deviations...

Window: ModelSpace

Menu: Tools | Measure

Action: Opens dialog

Usage: The **Contours to Mesh Deviations...** command compares the line segments in the selected contour set with the faces in a selected mesh (or originating mesh if no mesh is selected), and uses the data to calculate deviations in accordance with user defined parameters.

- This command is active only for mesh objects that have been created from a point cloud as opposed to point clouds that are merely viewed as a mesh. See the *Create Mesh* command for more information on creating a mesh object.

To calculate contours to mesh deviations:

1. Select a contour line for which to measure deviation from the original mesh object.
 - Selecting one line effectively selects the entire contour line set to which the selected line belongs, enabling you to measure deviations for the entire contour line set.
 - It is recommended that you hide the mesh so that only contour lines are visible. This makes it

easier to see deviating line segments. To hide the mesh, see the *Set Object Visibility* entry in this chapter.

2. Select the mesh object for which to take the deviation measurements. If no mesh object is selected, the mesh object used to create the contours is used.
3. From the **Tools** menu, point to **Measure**, then select **Contours to Mesh Deviations....**
Initialization commences, which culminates with the display of the **Deviation Analysis** dialog.
4. Use the slider bar to set the threshold on which to measure deviations. As the slider is moved, the graphical display is adjusted continually to highlight only those line segments with deviations from the mesh object that fall above the threshold.

Note: The **Minimum** and **Maximum** fields are display only, and indicate the range for which deviations can be measured.

5. Select whether to highlight line segments based on **Maximum** or **Average** deviation, and whether to use **Absolute** values (no positive or negative values) or **Signed** values.

Note: Deviation analysis results are displayed in the **Details** field.

6. Click **OK** when you are done. The **Deviation Analysis** dialog is closed.

Convert Elbow to Miter...

Window: ModelSpace

Menu: Tools | Piping

Action: Executes command

Usage: The **Convert Elbow to Miter...** replaces the selected elbow connector with a mitered connector.

- The command may also be used to change the value of an existing miter.

To convert an elbow to a mitered connector:

1. Select the elbow you want to convert.
2. From the **Tools** menu, point to **Piping**, and then select **Convert Elbow to Miter....** The **Number of Miter Segments** dialog appears.
3. Enter the number of segments you want to represent the miter and click **OK**. The elbow is converted.

Coordinate System

Window: ModelSpace

Menu: View

Action: Displays submenu

Usage: Pointing to **Coordinate System** displays submenu commands used to set and show the current coordinate axes, set the origin, and save and edit user-defined coordinate systems.

- Setting your own coordinate system as you begin modeling is helpful in maintaining your orientation.

The following commands, submenus and dialogs are available from the **Coordinate System** submenu:

[Save/Edit Coordinate Systems](#), [Show Axes](#), [Set Origin](#), [Set From Points](#), [Set Using One Axis](#), [Set Using Two Axes](#), [Set ScanWorld Coordinate System](#), [Reset ScanWorld Coordinate System](#)

Coplanar

Window: ModelSpace

Menu: Edit Object | Align

Action: Executes command

Usage: The **Coplanar** command aligns one or more selected objects so that they lie in the same plane as a reference object.

To make objects coplanar:

- Multi-select the objects you want to make coplanar.
 - The last object selected is the reference object (the one to which the others align).
- From the **Edit Object** menu, point to **Align**, and then select **Coplanar**. The selected objects are aligned to the plane of the reference object.

Copy

Window: ModelSpace

Menu: Create Object

Action: Opens dialog

Usage: The **Copy** dialog is used to make a copy (or an array of copies) of the selected object at a specific location relative to the original object.

To place a copy of an object at a specific location:

- Select the object you want to copy.
- From the **Create Object** menu, select **Copy**. The **Copy** dialog appears, and a graphical indicator appears in the ModelSpace viewer illustrating the direction and distance of the move at the current settings.
 - If multiple objects are selected, each object is labeled with a unique identifier (such as *cylinder 1*, *cylinder 2*, etc.) in the ModelSpace to aid in differentiating objects and points on those objects.
 - Significant points (i.e., the end points of a cylinder) as well as current and subsequent pick points are numbered and displayed on the selected objects. These numbers can be used as

references to move objects.

- If multiple objects are selected, you may specify a subset of the selected objects to be affected by the move from the **Apply to** list.
3. Click the arrow icon to the right of the **Direction of Move** row. The **Direction of Move** subdialog appears.
 4. Specify the direction for the move using axes, points or picks, and then click **OK**. The **Direction of Move** subdialog closes.
 5. Click the arrow icon to the right of the **Distance of Move** row. The **Distance** subdialog appears.
 6. Specify the distance for the move using points, picks, or custom distance, and then click **OK**. The **Distance** subdialog closes.
 7. In the **Number of Copies** field, enter the number of copies you want to create.
 - If you are creating more than one copy, the copies are moved so that each copy after the first is translated from the previous copy according to the current settings.
 8. Select **Copy Annotations** to duplicate any annotations that are attached to the original object for each new copy.
 9. Click **Copy**. The copy appears.
 - To create another copy using the current settings, click **Copy** again.
 10. When you are finished creating copies of the object, click **Close** to exit the dialog.

To place a copy of an object at a specific angle:

1. Select the object you want to copy.
2. From the **Create Object** menu, select **Copy**. The **Copy** dialog appears.
 - If multiple objects are selected, each object is labeled with a unique identifier (such as *cylinder 1*, *cylinder 2*, etc.) in the ModelSpace to aid in differentiating objects and points on those objects.
 - Significant points (i.e., the end points of a cylinder) as well as current and subsequent pick points are numbered and displayed on the selected objects. These numbers can be used as references to move objects.
3. Select the **Copy at Angle** tab. A graphical indicator appears in the ModelSpace viewer illustrating the rotation angle and axis of the current settings.
4. Click the arrow icon to the right of the **Axis of Rotation** row. The **Axis of Rotation** subdialog appears.
5. Specify the axes around which you want to rotate using standard or custom axes, points or picks, and then click **OK**. The **Axis of Rotation** subdialog closes.
6. Click the arrow icon to the right of the **Center of Rotation** row. The **Center of Rotation** subdialog appears.
7. Specify the centerpoint around which you want to rotate using picks or reference points, or custom coordinates, and then click **OK**. The **Center of Rotation** subdialog closes.
8. Click the arrow icon to the right of the **Angle of Rotation** row. The **Angle of Rotation** subdialog appears.
9. Specify the angle by which you want to rotate using reference axes, points, or custom values, and then click **OK**. The **Angle of Rotation** subdialog closes.
10. If multiple objects are selected, you may specify a subset of the selected objects to be affected by the move from the **Apply to** list.
11. In the **Number of Copies** field, enter the number of copies you want to create.

- If you are creating more than one copy, the copies are swept so that each copy after the first is rotated from the previous copy according to the current settings.
- 12.** Select **Copy Annotations** to duplicate any annotations that are attached to the original object for each new copy.
- 13.** Click **Copy**. The copy appears.
- To create another copy using the current settings, click **Copy** again.
- 14.** When you are finished creating copies of the object, click **Close** to exit the dialog.

Copy (Limit Box Manager)

Window: Limit Box Manager

Menu: Limit Box

Action: Executes command

Usage: The **Copy** command creates a new limit box that is a copy of the currently selected limit box. The new limit box is automatically identified as a copy of the original limit box.

Copy Fenced to New ModelSpace

Window: ModelSpace

Menu: File | Launch

Action: Executes command

Usage: The **Copy Fenced to New ModelSpace** command creates a new ModelSpace, copies the fenced object(s) into the new ModelSpace, and opens a ModelSpace View for the new ModelSpace.

- This command is very useful for editing objects in a separate ModelSpace containing only the objects with which you are currently working.

To copy fenced objects into a new ModelSpace for editing:

1. Draw a fence around the object(s) or group you want to copy.
 - The fence is projected forward in the view plane. Partially fenced objects are included. Partially fenced clouds are not segmented, but only the points within the fence are copied.
2. From the **File** menu, point to **Launch** and then select **Copy Fenced to New ModelSpace**. A new ModelSpace is created, and a ModelSpace View containing the fenced objects is opened.
3. Edit the objects as desired.
 - To copy the edited objects back to the original ModelSpace, see the *Update Original ModelSpace* entry in this chapter, or close the launched ModelSpace viewer.
4. When finished editing, close the ModelSpace. The **Closing ModelSpace Viewer** dialog appears.
5. Select closing options for the copied ModelSpace and objects.
 - Select **Merge into original ModelSpace** to copy all objects that were either edited or created

in the descendant ModelSpace back into the original ModelSpace. Objects from the source ModelSpace that have not changed are not duplicated in the resulting merged ModelSpace.

- Select **Remove link from original ModelSpace** to save the copied ModelSpace.
 - Select **Delete after close** to mark the current ModelSpace for deletion.
6. Click **Close**. The copied ModelSpace is closed.

Copy Pick Point(s)

Window: **ModelSpace**

Menu: **Edit**

Action: Executes command

Usage: The **Copy Pick Point(s)** command copies picked points to the Windows clipboard in text format.

Copy Selection to New ModelSpace

Window: **ModelSpace**

Menu: **File | Launch**

Action: Executes command

Usage: The **Copy Selection to New ModelSpace** command creates a new ModelSpace, copies the selected object(s) into the new ModelSpace, and opens a ModelSpace View for the new ModelSpace.

- This command is very useful for creating a smaller ModelSpace that contains only the objects with which you are currently working.

To edit selected objects in a new ModelSpace:

1. Select the object(s) or group you want to edit.
2. From the **File** menu, point to **Launch** and then select **Copy Selection to New ModelSpace**. A new ModelSpace is created, and a ModelSpace View containing the fenced objects is opened.
3. Edit the objects as desired.
 - To copy the edited objects back to the original ModelSpace, see the *Update Original ModelSpace* entry in this chapter, or close the launched ModelSpace viewer.
4. When finished editing, close the ModelSpace. The **Closing ModelSpace Viewer** dialog appears.
5. Select closing options for the copied ModelSpace and objects.
 - Select **Merge into original ModelSpace** to copy all objects that were either edited or created in the descendant ModelSpace back into the original ModelSpace. Objects from the source ModelSpace that have not changed are not duplicated in the resulting merged ModelSpace.
 - Select **Remove link from original ModelSpace** to save the copied ModelSpace.
 - Select **Delete after close** to mark the current ModelSpace for deletion.
6. Click **Close**. The copied ModelSpace is closed.

Copy to ControlSpace

Window: ModelSpace

Menu: Tools | Registration

Action: Executes command

Usage: The **Copy to ControlSpace** command adds the select object(s) to the designated ControlSpace(s).

- If there are Registration-specific ControlSpaces a dialog appears and allows you to choose the ControlSpace(s) to which the object is copied.

Note: Many objects are added to the ControlSpace automatically, such as objects with registration labels, HDS Targets, and sphere targets. The **Copy to ControlSpace** command provides you with a method for adding objects manually to a ControlSpace that should be considered by the [Auto-Add Constraints](#) command.

- For more information on the ControlSpace and its use in the registration process, see the [Registration](#) section in the *Getting Started* chapter.

Create... (Contours)

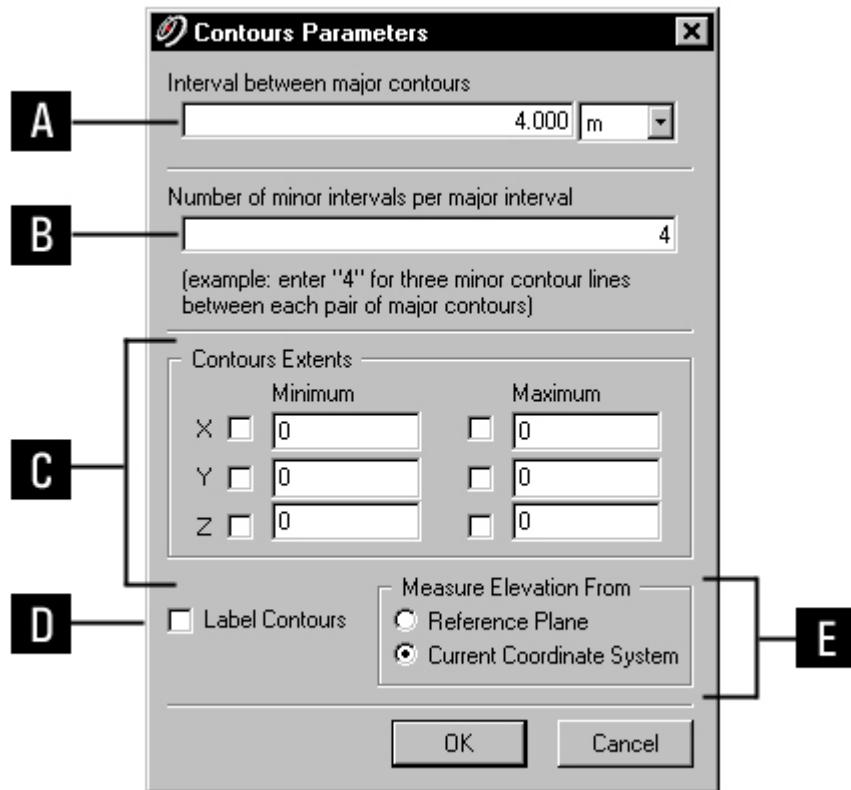
Window: ModelSpace

Menu: Tools | Contours

Action: Opens dialog

Usage: The **Contours Parameters** dialog is used to generate and customize contours parallel to either the active Reference Plane or the current coordinate system as defined by the user.

Contours Parameters Dialog



- A. Enter a value representing the distance between major contour lines and select the units of distance desired.
- B. Enter a value representing the number of minor contours to be displayed between each major contour.
- C. Set boundaries within which to generate contours by checking Minimum and/or Maximum for the desired plane(s). A value representing the extent of each selected limit can be entered as desired. Contours extent values are represented by the same units as those selected for the distance between major contours.
- D. Check this box to display labels for each contour line.
- E. Select whether to use the Reference Plane or the current coordinate system as the base for contour measurements.

To create contours:

1. Select the point cloud(s) or the mesh that you want to contour.
 - Make sure that the clouds are displayed as a mesh. See the *View Object As* entry in this chapter for more information.
2. From the **Tools** menu, point to **Contours**, then select **Create....** The **Contours Parameters**

- dialog appears.
3. Set the distance between major contours, the units by which that distance is measured, and the number of minor contours to be drawn between major contours.
 4. If you want to set boundaries within which to draw contours, check **Minimum** and/or **Maximum** in the **Contours Extents** box for the desired plane(s), then enter a value representing the extent of each selected limit (optional).
 - Values entered in the **Contours Extents** fields are measured in units as defined in the **Interval between Major Contours** field
 5. Check the **Label Contours** box to display contour line labels.
 - Contour label properties can be edited by selecting the contour line, then selecting **Edit Properties** from the **Edit Objects** menu. You can change text color and size, you can elect the type of notation for elevation values and whether or not to display elevation unit suffixes (e.g., ft), and you can change the alignment of the contour label text relative to the contour lines.
 6. Select either **Reference Plane** or **Current Coordinate System** as the base from which to draw the contours ($Z=0$).
 7. Click **OK** to create the contours. Contours are drawn parallel either to the active Reference Plane or to the current coordinate system as defined in the previous step.
 - By default, major contours are drawn with a heavier line weight than minor contours. Use the **Edit Properties** dialog (**Edit Object | Edit Properties...**) to modify contour line properties.

Create (Exclusion Volume)

Window: ModelSpace

Menu: Tools | Exclusion Volume

Action: Executes command

Usage: The **Create** command replaces an object or group of objects with a box containing the objects.

- An exclusion volume is an object that encloses other objects. It is typically used to contain clouds of points for which you need only a rough approximation of the volume occupied.

To create an Exclusion Volume:

1. Select the object(s) or group you want to enclose.
2. From the **Tools** menu, point to **Exclusion Volume**, and then select **Create**. The object(s) or group is replaced by a rectangular volume containing the objects.
- To restore the contents of an exclusion volume, select the exclusion volume, and then select **Disassemble** from the **Tools | Exclusion Volume** submenu.

Create (Limit Box Manager)

Window: Limit Box Manager

Menu: Limit Box

Action: Displays dialog

Usage: The **Create** command displays the **Create Limit Boxes** dialog, which is used to add new limit boxes with specified locations and size.

- If more than one pick point is present, the limit box is initialized to the minimum bounding box of the pick points. This is useful for quickly isolating cloud data with a few picks.

Create Alignment

Window: ModelSpace

Menu: Tools | Alignment and Section

Action: Opens dialog

Usage: The **Create Alignment** dialog is used to generate an alignment object for a selected line, polyline, or arc.

- For more information about alignments and the sections process, see the [Sections Manager](#) chapter.

To create an alignment:

1. Select the line, polyline, or arc you want to use as an alignment.
 - The object used to create an alignment can be imported from your CAD software via the COE exchange format (see the *Cyclone Object Exchange* chapter). The object may also be created in a variety of ways including using *Cyclone*'s Drawing tools and creating the object via the **From Pick Points** submenu commands (see the *From Pick Points* entry in the *Commands* chapter).
 - When a single object is selected, the object must be picked near the end-point at which the starting station is assigned.
 - If the start station is assigned to the wrong end, use the **Switch Alignment Start/End** command in the **Tools | Alignment and Section** menu.
 - When multiple objects are selected, the objects must be geometrically connected in a single unbroken sequence. The object picked last provides the end-point at which the starting station is assigned.
 - Alignments may be horizontal, skewed, or a mixture of both, but may not be vertical.
2. From the **Tools** menu, point to **Alignment and Section**, and then select **Create Alignment**. The **Create Alignment** dialog appears.
3. Enter a **Start Station** and click **OK**. The alignment object is created.
 - Station numbers increase with the linear distance "traveled" along the alignment, from the starting station.

Create and Open ModelSpace

Window: Registration

Menu: Registration

Action: Executes command

Usage: The **Create and Open ModelSpace** command creates and opens a ModelSpace from the registered ScanWorld created from a successful registration.

- The **Create and Open ModelSpace** command behaves identically to the [**Create ModelSpace**](#) command.
- The data contained in the resulting ModelSpace depends on which component ModelSpaces are selected in the **ModelSpaces** tab when the **Create ModelSpace** command is executed.
- For more information on the registration process, see the [**Registration**](#) entry in the *Getting Started* chapter.

Create and Open ModelSpace View

Window: Navigator

Menu: File

Action: Executes command

Usage: The **Create and Open ModelSpace View** command creates and opens a ModelSpace View for the selected ModelSpace.

- Views created using this command are stored in the ModelSpace folder from which they were created.
- Multiple views can be created from the same ModelSpace using this command.
- ModelSpace View names can be changed; select the ModelSpace View in Navigator, select [**Rename**](#) from the **Edit** menu, then type in the new name.

Create Branch

Window: ModelSpace

Menu: Tools | Piping

Action: Executes command

Usage: The **Create Branch** command creates an intersection of two or more selected pipes.

- The supported piping connectors are: T, Y, and Cross. The geometric arrangement of the selected pipes determine which (if any) branch is created.
- To set preferences that determine whether or not a branch can be made, see the **Piping: Max. Offset** and **Piping: Max. Angle** settings in the **Modeling** tab of the [Edit Preferences](#) dialog.

To create a piping connector:

1. Select the pipes you want to connect.
 - A "T" consists of one pipe (the "run") intersected (but not crossed) by another pipe (the "branch").
 - A "Y" consists of three non-overlapping pipes intersecting at a shared point.
 - A Cross consists of two crossing pipes, or one pipe (the "run") intersected (but not crossed) by two other pipes (the "branches").
2. Open the **Piping Mode** dialog and select the SKEY appropriate for the type of branch to be created. See the *Piping Mode* entry in this chapter.
3. From the **Tools** menu, point to **Piping**, and then select **Create Branch**. If the geometric arrangement supports a connection, a branch is created.
 - The branch is represented by a placeholder object to which the selected pipes are connected.
 - The placeholder for the branch is assigned the current SKEY appropriate to the type of branch represented. The pipes are automatically extended or sliced as necessary, and a piping connection is established (see the *Connect Piping* entry in this chapter).

Create Drawing

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Create Drawing** command is the final step to finish the current drawing procedure.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

Create Drawing from Object

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Create Drawing from Object** command converts an existing line object back into a drawing object, projecting the existing object onto the drawing plane as a polyline.

- The **Create Drawing from Object** command works with previously created line drawings (including polylines, arcs, rectangles, etc.) and imported line objects.
- The **Create Drawing from Object** command can be used to convert a line object into a drawing to be used with the [**Export Template**](#) command.

Create Lines from Cuts

Window: ModelSpace

Menu: Tools | Cutplane

Action: Executes command

Usage: The **Create Lines from Cuts** command creates polylines where the active Cutplane intersects objects and meshes.

To create polylines where the active Cutplane intersects objects and meshes:

1. Position the Cutplane to intersect the objects or meshes from which you want polylines.
2. From the **Tools** menu, point to **Cutplane** and then select **Create Lines from Cuts**. Polylines are created where the active Cutplane intersects objects or meshes.

Create Mesh...

Window: ModelSpace

Menu: Tools | Mesh

Action: Executes command

Usage: The **Create Mesh...** command creates a mesh object from the selected point clouds, polylines, vertices, or meshes.

- Mesh objects created using this command can be modified using commands in the **Tools | Mesh** submenu. For information on mesh editing, see the [**Meshes**](#) section in the *Modeling* chapter, or any of the **Mesh** submenu option commands.
- The number of points used when creating a mesh from a point cloud is set via the **Computation: Create TIN: Max Points** setting in the **Point Cloud** tab of the **Edit Preferences** dialog.

To create a mesh object:

1. Select the point clouds, polylines, vertices, or meshes to be used for creating the new mesh object.
2. From the **Tools** menu, point to **Mesh**, and then select **Create Mesh...**. The **Meshing Style** dialog appears.

3. Select the type of mesh object you wish to create.
- Select **Basic Meshing** to create a basic mesh consisting of triangles drawn by using trios of adjacent points as triangle vertices. Select **Complex Meshing** to create a mesh consisting of triangles drawn by using trios of adjacent points that are likely to lie in the same plane. Select **TIN Meshing** to create a mesh wherein no two vertices share the same horizontal coordinates; the current coordinate system determines the “up” direction for a TIN mesh.

Note: **Basic Meshing** and **Complex Meshing** cannot be done with more than one point cloud.

- When there are multiple selections in Step 1, only the TIN Meshing option is available.
- **Basic Meshing** and **Complex Meshing** can only be used with a single point cloud selection.
- 4. Click **OK**. The mesh object is created.
- Click **Cancel** to abort the command.
- To make the point cloud visible again, use the **ScanWorld Explorer**. See the *ScanWorld Explorer* entry in this chapter for more information.

Create ModelSpace

Window: **Registration**

Menu: **Registration**

Action: Executes command

Usage: The **Create ModelSpace** command creates a ModelSpace from the Registered ScanWorld created from a registered ScanWorld.

- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.
- The data contained in the resulting ModelSpace depends on which component ModelSpaces are selected in the **ModelSpaces** tab when the **Create ModelSpace** command is executed.

To create a ModelSpace from a successful registration:

1. From the **ModelSpaces** tab, select the ModelSpaces whose data you want included in the new ModelSpace.
 - If no ModelSpace is selected, the resulting ModelSpace contains only the raw scan data from all of the component ScanWorlds.
2. From **Registration** menu, select **Create ModelSpace**. The ModelSpace is created with the data from the selected component ModelSpaces.
 - The ModelSpace is located beneath the registered ScanWorld in the **Navigator** window.
 - To create and open a ModelSpace with a single command, use the **Create and Open ModelSpace**.

Create Objects with Random Colors

Window: ModelSpace

Menu: Edit Object | Appearance

Action: Toggles menu item ON/OFF

Usage: The **Create with Random Colors** command toggles the assignment of a random color to all created or fit objects.

- When this command is checked, an object is assigned a random color when it is created in order to make it stand out. When unchecked, objects are created with the default color.

Create Path

Window: ModelSpace

Menu: Tools | Animation

Action: Executes command

Usage: The **Create Path** command creates a spline curve through the selected Camera objects, following the order in which they were selected.

- For an overview of the animation process, see the [Animation](#) chapter.

Create Path (Loop)

Window: ModelSpace

Menu: Tools | Animation

Action: Executes command

Usage: The **Create Path (Loop)** command creates a closed spline loop through the selected Camera objects, following the order in which they were selected.

- For an overview of the animation process, see the *Animation* entry in the [Modeling](#) section of the *Getting Started* chapter.

Create Redlines

Window: ModelSpace

Menu: Tools | Redlining

Action: Toggles menu item ON/OFF

Usage: The **Create Redlines** command toggles **Redline** mode, which is used to create 2D lines and add comments to a ModelSpace View's current viewpoint.

- Redlining behaves identically to 2D drawing except that 2D drawing is done on the active Reference Plane, and Redlining is done on an imaginary plane that is perpendicular to and centered in the current ModelSpace View.
- Redlines are tied to a specific viewpoint. Loading a redline restores the viewpoint from which it was created. When the viewpoint changes, the redline is no longer visible (you can restore visibility by reloading the redline).
- To load and manage saved redlines, see the [Edit Redlines](#) command.

To create a Redline:

- From the **Tools** menu, point to **Redlining** and then select **Create Redlines**. You are now in **Redlining** mode.
- Alternatively, you can load a Redline from the **Edit Redline** dialog to enter the redline mode (see the *Add/Edit Redlines* entry in this chapter). This allows you to add drawings to an existing set of redlines.
- Follow the procedures for 2D drawing (see the *Drawing* entry in the *Modeling* chapter) to create your redlines.
- All lines and text created during one redlining session are saved and restored as a single redline in the redline list (see the *Add/Edit Redlines* entry in this chapter).
- From the **Tools** menu, point to **Redlining** and then select **Create Redlines** to toggle it OFF and exit **Redlining** mode.

Create ScanWorld/Freeze Registration

Window: **Registration**

Menu: **Registration**

Action: Executes command

Usage: The **Create ScanWorld/Freeze Registration** command uses the current registration to create a single ScanWorld that includes the registered (component) ScanWorlds in a unified coordinate system. It also locks all constraints in place. At this point constraints can no longer be modified unless the registration is unfrozen via the **Unfreeze Registration** command.

- When this command is executed, a ScanWorld appears in the Navigator in place of the Registration object, which is made a child of the newly created ScanWorld.
- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.

Create Sections

Window: ModelSpace

Menu: Tools | Alignment and Section

Action: Opens dialog

Usage: The **Create Sections** dialog is used to generate sections along the selected alignment object.

- For more information about alignments and the sections process, see the *Sections Manager* chapter.

Create Sections Dialog Options

INITIAL STATION

Specify the distance along the alignment at which you want to place the first section.

END STATION

Specify the distance along the alignment at which you want to place the last section.

SPACING

Specify the interval between sections.

INCLUDE TRANSITION POINTS

Select this option to also create a section at each transition point (the point at which two of the original lines or arcs connect) that falls within the creation interval.

SECTION EXTENTS

All **Section Extents** settings are relative to the current "up" direction and the direction of the alignment (from the start station towards the end station).

CREATE LINES

Select this option to create lines based on a temporary TIN mesh that is created from the points within each section. More controllable results may be obtained by first creating and adjusting the sections, and then creating lines afterward via the **Create Lines at Section** command in the **Sections Manager** dialog.

COLLECT POINTS

Select this option to introduce merged clouds from the points that fall within the boundaries of each section. This function can also be performed after sections have been created via the **Collect Points in Section** command in the [Sections Manager](#) dialog.

To create sections:

1. Select the desired alignment object.
 2. From the **Tools** menu, point to **Alignment and Section**, and then select **Create Sections**. The **Create Sections** dialog appears.
- The **Create Sections** command can also be accessed from the **Sections** menu in the

Sections Manager dialog.

3. Choose the desired alignment settings, and then click **OK**. The sections are created and the **Sections Manager** dialog appears.

Create Sections from Picks

Window: ModelSpace

Menu: Tools | Alignment and Section

Action: Opens dialog

Usage: The **Create Sections** dialog is used to create sections between sequential pairs of pick points.

- The sections created are managed in the Sections Manager.

To create sections from pick points:

1. Pick the point at the left-most end of the section to be created. Shift+Pick the point at the right-most end. The section will pass between these two points.
 - When the section is activated in the Sections Manager, its “Left” and “Right” are determined by the order in which the points were picked.
2. Shift+Pick any additional pairs of points.
3. From the **Tools** menu, point to **Alignment and Section**, then **Create Sections from Picks**. The **Create Sections** dialog appears.
4. Select how the Top and Bottom parameters of each section are set.
 - Select **Automatic Top and Bottom** to set the **Top** and **Bottom** parameters of each section based on the highest and lowest points within that section.
 - Select **Specify Section Elevation** to set the **Top** and **Bottom** parameters of each section based on user-specified absolute elevations, relative to the current UCS.
5. Enter the **Depth** of the section. This is the distance between the **Front** and **Back** parameters of the section.
6. Select the **Create Lines** checkbox to create cross-section lines within each section.
7. Select the **Collect Points** checkbox to introduce merged clouds from the points that fall within each section.
8. Click **OK**. The sections are created and added to the Sections Manager. The cross-section lines and points are optionally processed.

Create Simulated Scanner...

Window: ModelSpace

Menu: Tools | Scanner

Action: Executes command

Usage: The **Create Simulated Scanner...** command launches a **Scan Control** window for a simulated scanner.

- The **Scan Control** window created by this command operates identically to a **Scan Control** window connected to a real scanner. For information on **Scan Control** commands, see their individual entries.
- For general information on the **Scan Control** window, see the *Scanning* entry in the *Scanning with the HDS2500* chapter.

Cubic Spline

Window: ModelSpace

Menu: Create Object | From Pick Points

Action: Executes command

Usage: The **Cubic Spline** command creates a cubic spline curve that passes through a series of picked points.

To create a cubic spline:

1. Multi-select a series of points.
2. From the **Create Object** menu, point to **From Pick Points**, and then select **Cubic Spline**.
The cubic spline is created.
 - The cubic spline curve is drawn from point to point in the same order in which the points were picked.
 - To edit a cubic spline, select it and then use the vertex handles to change its shape.

Curve

Window: ModelSpace

Menu: Create Object | From Intersections

Action: Executes command

Usage: The **Curve** command creates a line at the intersection of two planar objects.

To create a line at the intersection of objects:

1. Select the planar objects at whose intersection you want to use to create a line.
2. From the **Create Object** menu, point to **From Intersections**, and then select **Curve**. The line is created.
 - Even if the selected planes are not physically touching, the line is created at the planar intersection.

Customize Hotkeys...

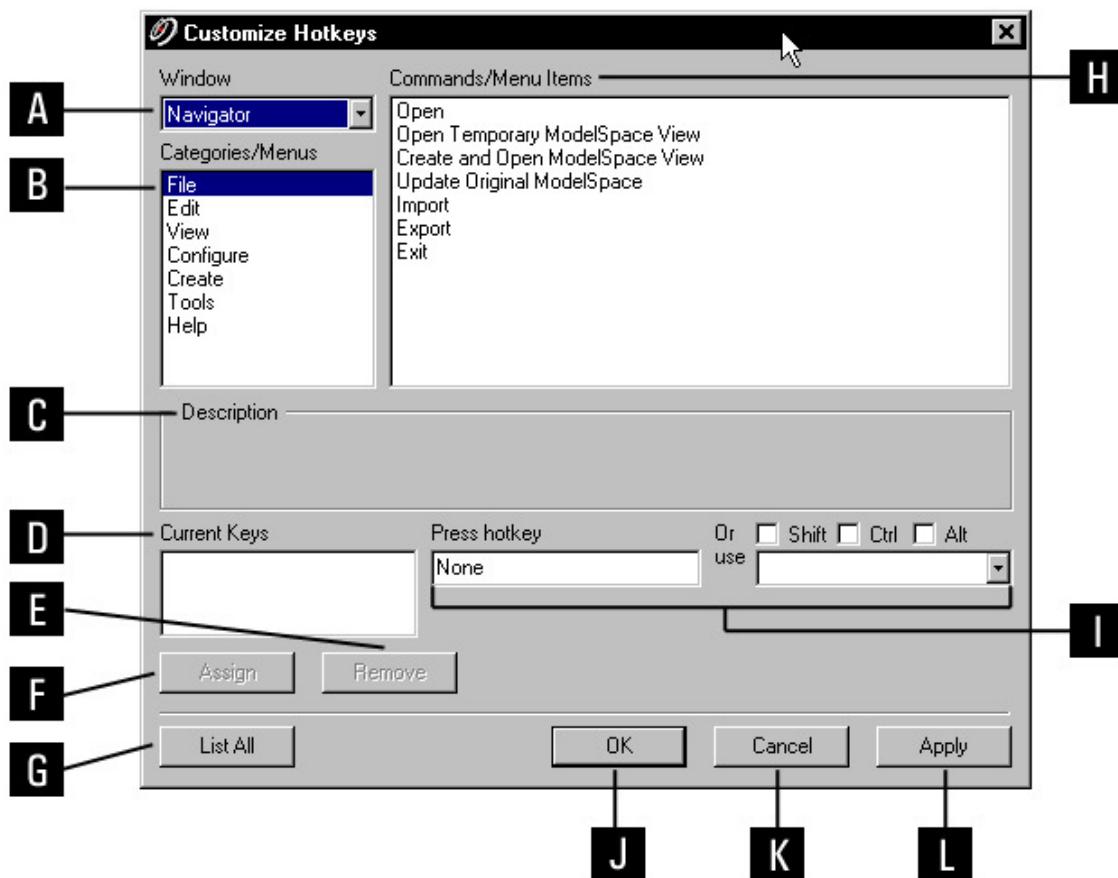
Window: ModelSpace, Navigator, Image Viewer, Scan Control, Registration

Menu: Edit

Action: Opens dialog

Usage: The **Customize Hotkeys** dialog is used to manage your hotkeys, which allow you to activate commands by a simple key combination, bypassing the toolbar and menu.

Customize Hotkeys Dialog



- A. List of all commands in the selected category/menu.
- B. List of windows.
- C. List of categories/menus in the selected window.
- D. Description of the selected command.
- E. View any currently assigned hotkey(s) for the selected command.
- F. Click to view a list of all commands and assigned hotkeys by window.

- G. Assign the entered hotkey to the selected command.
- H. Remove the selected current hotkey from the selected command.
- I. Use these fields to assign hotkey combinations to selected commands.
- J. Cancel hotkey changes.
- K. Click to save hotkey changes and continue working in this dialog.
- L. Click to save hotkey changes and close the dialog.

To assign a keyboard hotkey:

1. From the **Edit** menu, select **Customize Hotkeys...**. The **Customize Hotkeys** dialog appears.
2. From the **Window** list, select the type of window that contains the command for which you want a hotkey.
3. From the **Category/Menus** list, select the menu that contains the command for which you want a hotkey.
4. From the **Commands/Menu Items** list, select the command for which you want a hotkey.
5. Click in the **Press new shortcut/hotkey** field and press the key or combination of keys that you want as a hotkey for the selected command.
 - Certain key combinations must be entered using the check boxes and the drop-down list in the **Or use** section.
6. Click **Assign**. The new hotkey appears in the **Current Keys** list.
7. Click **Apply** or **OK** to save changes.

To remove a keyboard hotkey:

1. From the **Edit** menu, select **Customize Hotkeys...**. The **Customize Hotkeys** dialog appears.
2. From the **Current Keys** list, select the hotkey you want to delete.
3. Click **Remove**. The selected hotkey is removed from the **Current Hotkeys** list.
4. Click **Apply** or **OK** to save changes.

Customize Lookup Lists...

- Window:** **Navigator**
- Menu:** **Edit**
- Action:** Opens dialog
- Usage:** This command opens the **Annotation Lookup Lists** dialog, which is used to import and export text files used as lookup list contents, as well as to delete lookup lists.
- Note:** Actual editing of lookup lists is not performed in this dialog. Instead, the editing is done using the text editor of your choice.
- Once a lookup list is created, it is available for custom annotations creation for all databases residing on the local server. For example, a custom lookup list created for one project is also visible in all other projects.

To create a quick lookup list:

1. From the **Navigator** window, open the **Edit** menu and select **Customize Lookup Lists....** The **Annotation Lookup Lists** dialog is displayed.
2. To create a text file you can use as a template for proper formatting of a list file, click the **Export a sample quick lookup list file** icon in the toolbar. The **Select a file name for example lists** dialog appears.
3. Type a file name in the **File Name** field; be certain to type in the *.txt* extension as well. Select a folder destination for the file, and click **SAVE**.
4. Use the text editor of your choice (e.g., Notepad) and open the newly created text file. Instructions for creating lookup lists are included in the text file. Follow the directions precisely, or you will not be able to import the file back into the **Annotation Lookup Lists** dialog.

Note: Below the instructions, you may see previously defined lists (if they exist). Because these lists are in the proper format, you may opt to use one as a template for your list.

5. Enter your new list name using the following format:

List Name=*enter list name*

List Type=*enter list type (text, integer, or decimal)*

Note: “List Name=...” and “List Type=...” should be typed *exactly* as shown; put spaces where indicated, and none where they are not. Use uppercase and lowercase exactly as indicated.

6. Below the list name and type, enter your list contents, with each entry appearing on a new line. You can copy/paste data into the document, as long as the list conforms to the formatting instructions.
7. Save the *.txt* document, close the text editor, and return to *Cyclone* Navigator.
8. If it is not already open, open the **Annotation Lookup Lists** dialog, and click the **Import a lookup list file** icon.
9. Find and select the *.txt* file you just saved, and click **OPEN**. The newly created list is added, and its contents are displayed in the Lookup List Contents window.

To edit lookup list contents:

1. From the **Navigator** window, open the **Edit** menu and select **Customize Lookup Lists....** The **Annotation Lookup Lists** dialog is displayed.
2. Click the **Export lookup lists to a text file** icon in the toolbar. The **Select a file for export** dialog is displayed.
3. Select a previously existing text file, or type in a new file name in the **File Name** field. Be sure to type in the *.txt* extension as well.

Note: All currently defined lists are exported when the **Export lookup lists to a text file** icon is selected; it is not possible to export only a selected list. However, as long as you edit only your targeted list, the other lists will remain unchanged when this file is imported back into *Cyclone*.

4. Select a destination folder for the file, and click **Save**.

5. Open the newly created text file in the text editor of your choice (e.g. Notepad). Instructions for creating quick lookup lists are included in the text file. Follow the directions precisely, or you will be unable to import the file back into the **Annotation Lookup Lists** dialog.
6. Scroll down the text in the file and search for the list you wish to edit, then make changes as necessary; keep in mind that you must comply with the formatting instructions, or *Cyclone* will not be able to import the list.
7. Save the .txt document, close the text editor, and return to the **Navigator** window.
8. If it is not already open, open the **Annotation Lookup Lists** dialog, and click the **Import a lookup list** icon.
9. Find and select the .txt file you just edited and saved, and click **Open**. You will receive a notice that a list already exists with the same name. Select **Overwrite** to replace existing lists (where the list name has not changed) with new contents, or **Merge** to keep the existing lists' entries, but also add entries from the new file that are not in the existing list. New lists are added without prompting. All lists in the window reflect the latest changes.

To delete an annotation lookup list:

1. From the **Navigator** window, open the **Edit** menu and select **Customize Quick Lookup List....** The **Annotation Lookup Lists** dialog is displayed.
2. Highlight the list to be deleted, and click the **Delete** icon in the toolbar. The lookup list is deleted.
 - Deleting an annotation lookup list does not affect lists in use in other windows.

Customize Toolbars...

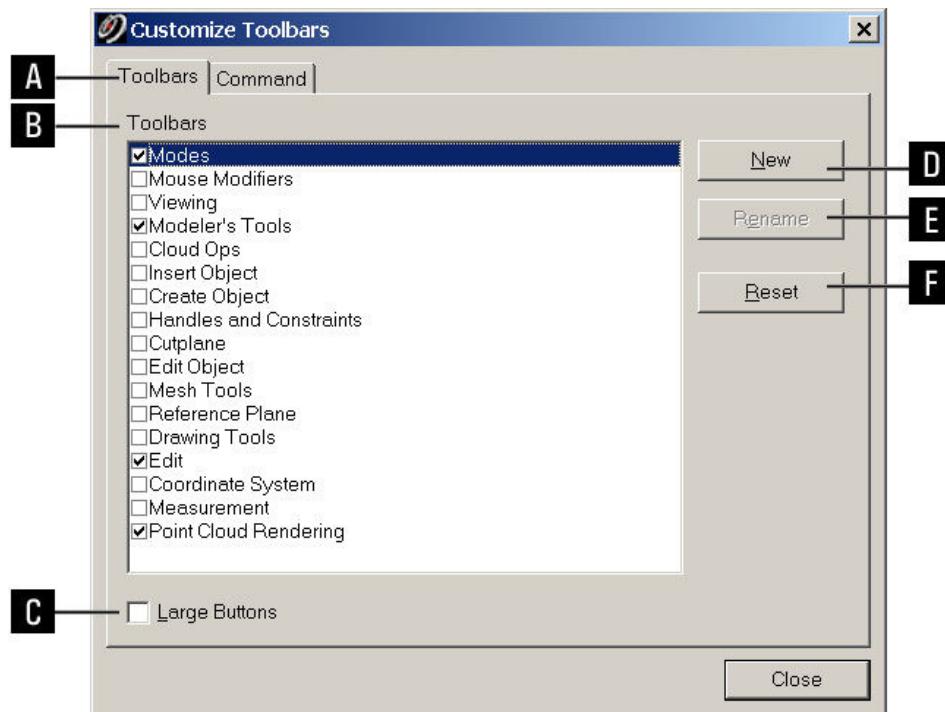
Window: ModelSpace, Navigator, Image Viewer, Scan Control, Registration

Menu: Edit

Action: Opens dialog

Usage: The **Customize Toolbars** dialog is used to customize all aspects of the toolbars that are available in *Cyclone*.

Customize Toolbars Dialog



- A.** The **Toolbars** tab allows you to create new toolbars, as well as elect whether to display new or existing toolbars. The Commands tab allows you to add available buttons to visible toolbars.
- B.** List of all available toolbars for the window from which the dialog was opened. Select a toolbar name to display that toolbar in the current window.
- C.** Check Large Buttons to display enlarged toolbar buttons.
- D.** Click the New button to create a new toolbar. You will be prompted to name the new toolbar. Once you OK the name, the toolbar is added to the list; it is checked automatically and displayed in the current window without contents. Use the Commands tab to add buttons.
- E.** Use the Rename button to give new names to existing toolbars.
- F.** The Reset button is available for standard *Cyclone* toolbars; the Delete button is available for toolbars that you have added. Use the Reset button to reset the configurations of the selected *Cyclone* toolbar to the default configurations. Use the Delete button to permanently remove a user-defined toolbar both from the toolbar list and from the current window.

To display/hide existing *Cyclone* toolbars:

1. From the **Edit** menu of the window for which you want to display/hide toolbars, select **Customize Toolbars....** The **Customize Toolbars** dialog appears.
2. In the **Toolbars** window of the **Toolbars** tab, click on a checkbox to display/hide the toolbar named. If checked, the toolbar is displayed; if not, the toolbar is hidden. As you select/deselect toolbar options, the corresponding toolbars appear/disappear from the current window display.
 - To rename a toolbar, highlight the desired toolbar and click **Rename**. Type in the new name and click **OK**. The toolbar is renamed
 - To reset a toolbar so that the original *Cyclone* contents are reinstated, and all user edits are eliminated, highlight the desired toolbar and click **Reset**. The toolbar is returned to its default state.
3. Check **Large Buttons** to display larger versions of toolbars and button faces.
4. Click **Close** when done.

To create a new toolbar:

1. From the **Edit** menu of the window for which you want to display/hide toolbars, select **Customize Toolbars....** The **Customize Toolbars** dialog appears.
2. In the **Toolbars** tab, click on **New**. The **New Toolbar** dialog appears.
3. Enter a toolbar name and click **OK**. The toolbar is added to the toolbars list; it is automatically checked and displayed in the current window.
 - Click **Cancel** to exit this dialog without adding a new toolbar.

To add/remove buttons from toolbars:

1. From the **Edit** menu of the window for which you want to display/hide toolbars, select **Customize Toolbars....** The **Customize Toolbars** dialog appears.
2. From the **Command** tab, select the category for which you want available buttons displayed. To the right, a series of buttons is displayed.
 - It is not necessary to be in the **Command** tab to remove buttons from toolbars.
 - The available categories correspond to the available menus for the current window.
 - Click any button to see a description of that button's function.
3. To add a button to a toolbar, click the desired button and drag it from the dialog to the desired location. You can add buttons to both existing and new toolbars.
4. To remove a button from a toolbar, click the desired button in its present toolbar location and drag it out of the toolbar. When you let go, the button disappears.
 - You can use the click and drag method to move buttons from one toolbar to another.
 - Use the **Reset** button to restore the original contents of a *Cyclone* toolbar.

Cut

Window: ModelSpace, Navigator

Menu: Edit

Action: Executes command

Usage: The **Cut** command deletes a selected object from its current location and copies it to the *Cyclone* clipboard.

Note: Cutting an object from a ModelSpace places it on the clipboard and removes it from its current location and from the database. Though cutting a point cloud removes it from the ModelSpace, it *does not* remove the original scan from the database. Original scans are always recoverable via the [ScanWorld Explorer](#) dialog.

Note: The Navigator's clipboard behaves differently from the ModelSpace clipboard; when **Copy** is selected, the selected object is *marked* for copying; however, it is not actually copied until you initiate the **Paste** command.

Cut by Distance from Point

Window: ModelSpace

Menu: Create Object | Segment Cloud

Action: Opens dialog

Usage: The **Segment Cloud by Distance from Point** dialog is used to segment the selected point cloud(s) based on each point's distance from a central point.

- For an overview of the segmentation process, see the [Segmentation](#) entry in the *Modeling* chapter.

To segment a point cloud by distance from a point:

1. Select the point cloud you want to segment.
2. From the **Create Object** menu, point to **Segment Cloud**, and then select **Cut by Distance from Point....** The **Segment Cloud by Distance from Point** dialog appears.
3. Click the button next to the **Point** field to specify the central point's coordinate.
 - Select **Last Pick Point** to use the last pick point.
 - Select **Last Pick Point Always** to use the last pick point, even when the pick point changes.
 - Select **Origin** to use the UCS origin.
 - Select **Custom** to manually enter a coordinate.
4. Select **Interval** to enter the interval at which the cloud will be regularly segmented, within the **Min** and **Max** distances.
5. Use the slider to interactively adjust the distances at which you want to divide the point cloud.
 - Pick one point on the cloud before invoking this command to automatically establish the lower bound, based on the distance to that pick point from the central point.
 - Shift+Pick two points on the cloud before invoking this command to automatically establish the lower and upper bounds, based on the distance to those pick points from the central point.
6. Click **Reset** to reinitialize this dialog, based on any changes to the selection or pick points.
7. Click **Segment**. The points within the indicated distance of the indicated point are now a separate point cloud that can be selected independently. Optionally, these points are also segmented at regular intervals.

Cut by Fence

Window: ModelSpace

Menu: Create Object | Segment Cloud

Action: Executes command

Usage: The **Cut by Fence** command segments the selected point cloud(s) into two subsets.

- For an overview of the segmentation process, see the [Segmentation](#) entry in the *Modeling* chapter.

To create a new segment of a point cloud:

1. Draw a fence around the points that you want to segment. (See the *Creating a Fence* entry in the *Modeling* chapter).
2. From the **Create Object** menu, point to **Segment Cloud**, and then select **Cut with Fence**.
The points within the fence are now a separate point cloud and can be selected independently.

Cut by Intensity

Window: ModelSpace

Menu: Create Object | Segment Cloud

Action: Opens dialog

Usage: The **Segment Cloud by Intensity Range** dialog is used to segment the selected point cloud(s) based on each point's intensity value.

- For an overview of the segmentation process, see the [Segmentation](#) entry in the *Modeling* chapter.

To segment a point cloud by intensity:

1. Select the point cloud that you want to segment.
2. From the **Create Object** menu, point to **Segment Cloud**, and then select **Cut by Intensity....**
The **Segment Cloud by Intensity** dialog appears.
3. Use the slider to interactively adjust the intensities at which you want to divide the point cloud.
 - Shift+Pick two points on the cloud before invoking this command to automatically establish the lower and upper bounds, based on the intensities at those pick points.
4. Click **Reset** to reinitialize this dialog based on any changes to the selection or pick points.
5. Click **OK**. The points above and below the indicated intensities are now separate point clouds that can be selected independently. The points between those bounds are separated from the rest of the points in the cloud.

Cut by Offset from Plane

Window: ModelSpace

Menu: Create Object | Segment Cloud

Action: Opens dialog

Usage: The **Segment Cloud by Offset from Plane** dialog is used to segment the selected point cloud(s) based on each point's offset from the indicated plane.

- For an overview of the segmentation process, see the [Segmentation](#) entry in the *Modeling* chapter.

To segment a point cloud by distance from a plane:

- Select the point cloud you want to segment.
- From the **Create Object** menu, point to **Segment Cloud**, and then select **Cut by Offset from Plane**.... The **Segment Cloud by Offset From Plane** dialog appears.
- Click the button next to the **Point** field to specify the coordinate through which the plane passes.
 - Select **Last Pick Point** to use the last pick point.
 - Select **Last Pick Point Always** to use the last pick point, even when the pick point changes.
 - Select **Origin** to use the UCS origin.
 - Select **Custom** to manually enter a coordinate.
- Click the button next to the **Normal** field to specify the plane's normal vector.
 - Select **Up Direction** to use the Up direction, even when it changes.
 - Select **X-Axis (Easting)** to use the UCS X-axis (Easting).
 - Select **Y-Axis (Northing)** to use the UCS Y-axis (Northing).
 - Select **Z-Axis (Elevation)** to use the UCS Z-axis (Elevation).
 - Select **Reference Plane Normal** to use the active reference plane's normal vector.
 - Select **Pick Point Normal** to use the normal vector at the last picked point.
 - Select **Custom** to manually enter a normal vector.
- Select **Interval** to enter the interval at which the cloud will be regularly segmented, within the **Min** and **Max** offsets.
- Use the slider to interactively adjust the offsets at which you want to divide the point cloud.
 - Pick one point on the cloud before invoking this command to automatically establish the lower bound, based on the distance to that pick point from the plane.
 - Shift+Pick two points on the cloud before invoking this command to automatically establish the lower and upper bounds, based on the distance to those pick points from the plane.
- Click **Reset** to reinitialize this dialog, based on any changes to the selection or pick points.
- Click **Segment**. The points within the indicated offsets of the indicated plane are now a separate point cloud that can be selected independently. Optionally, these points are also segmented at regular intervals.

Cut Near Ref Plane...

Window: ModelSpace

Menu: Create Object | Segment Cloud

Action: Opens dialog

Usage: The **Cut Near Ref Plane** dialog is used to segment a point cloud into one cloud that contains points near the active Reference Plane and another cloud that contains all other points.

- For an overview of the segmentation process, see the [Segmentation](#) entry in the *Modeling* chapter.

To segment point clouds near the Reference Plane:

1. Position the active Reference Plane, and then select the point cloud(s) that you want to segment.
2. From the **Create Object** menu, point to **Segment Cloud**, and then select **Cut Near Ref Plane**. The **Cut Near Ref Plane** dialog appears.
3. Enter a value, or use the **Region Thickness** slider to interactively adjust the thickness of the slice (or partition) near the active Reference Plane.
4. Click **OK**. The selected clouds are segmented.

Cut Sub-Selection

Window: ModelSpace

Menu: Create Object | Segment Cloud

Action: Executes command

Usage: The **Cut Sub-Selection** command segments a point cloud(s) into two subsets based on a point cloud sub-selection.

- For an overview of the segmentation process, see the *Point Cloud Segmentation* entry in the *Modeling* chapter.

To cut a point cloud sub-selection:

1. Draw a fence around the points that you want to segment.
2. Use the **Selection | Point Cloud Sub-Selection** submenu commands to make an initial sub-selection.
- You may continue to refine the selection via subsequent **Selection | Point Cloud Sub-Selection** submenu commands. See the *Advanced Segmentation* entry in the *Modeling* chapter for more information.
2. From the **Create Object** menu, point to **Segment Cloud**, and then select **Cut Sub-Selection**. The points within the sub-selection are now a separate point cloud and can be selected independently.

Cutplane

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Cutplane** displays submenu commands used to execute Cutplane operations and change settings.

- The Cutplane is a 3D plane with the primary function of "cutting" through objects in a ModelSpace and producing a 2D cross-section.
- For an overview of Cutplanes, see the [Cutplanes](#) entry in the *Modeling* chapter.

The following commands and dialogs are available from the **Cutplane** submenu:

[Add/Edit Cutplanes...](#), [Set on Object](#), [Set from Active Ref Plane](#), [Raise Active Cutplane](#),
[Lower Active Cutplane](#), [Set Offset...](#), [View Half-Space](#), [View Slice](#), [Set Slice Thickness...](#),
[Create Lines from Cuts](#), [Set Half-Space at Pick](#), [Set Slice from Picks](#)

Databases...

Window: Navigator

Menu: Configure

Action: Opens dialog

Usage: The **Configure Databases** dialog is used to add, remove, destroy, compact, and optimize databases.

Note: For detailed information on administering databases in *Cyclone*, see the [Databases](#) chapter.

To add a database:

1. Select the server to which you want to add a database.
2. From the **Configure** menu, select **Databases....** The **Configure Databases** dialog appears.
3. Click **Add**. The **Add Database** dialog appears.
4. In the **Database Name** field, type a logical name as an identifier for your logical database.
 - The logical name is the name displayed in the **Navigator**.
5. In the **Databases File name** field, type a file name for your database.
 - The database file name is the name used for the database file, which is saved in the **Cyclone\ Databases** directory by default.
 - If you leave the **Database File name** field empty, *Cyclone* bases the file name on the logical name you entered in the **Databases Name** field.
 - You may select a new location for the database file by clicking the **Browse** icon to the right of the **Database Name** field.
6. Click **OK**. The database is created.

To remove a database:

- When a database is removed, the database is removed from the list of databases available on the server; however, the database file is not deleted and may be restored using the steps to add a database (above).
1. From the **Configure** menu, select **Databases....** The **Configure Databases** dialog appears.
 2. From the list of databases, select the database that you want to remove, and then click **Remove**. The database is removed.
- To close the **Configure Databases** dialog and return to the **Navigator** window, click **Close**.

To destroy a database:

- When a database is destroyed, the database is removed and its file is permanently deleted and cannot be recovered from within *Cyclone*.
1. From the **Configure** menu, select **Databases....** The **Configure Databases** dialog appears.
 2. From the list of databases, select the database that you want to destroy, and then click **Destroy**. The database is deleted.
- To close the **Configure Databases** dialog and return to the **Navigator** window, click **Close**.

To toggle database visibility:

1. From the **Configure** menu, select **Databases....** The **Configure Databases** dialog appears.
- To toggle visibility, select/deselect the box below the **Visibility** icon next to the desired database.
- To close the **Configure Databases** dialog and return to the **Navigator** window, click **Close**.

To compact a database:

1. Select the server that contains the database you wish to compact.
2. From the **Configure** menu, select **Databases....** The **Configure Databases** dialog appears.
3. Select the database you wish to compact, and click **Compact**. A confirmation dialog appears.
4. Click **Yes**. A dialog appears prompting you to create a backup before beginning the compacting process.
5. Click **Yes** to create a backup (strongly recommended). The compaction process begins.
6. When finished, a notification dialog appears. Click **OK** to resume working in *Cyclone*.

To optimize a database:

1. Select the server that contains the database you wish to optimize.
2. From the **Configure** menu, select **Databases....** The **Configure Databases** dialog appears.
3. Select the database you wish to optimize, and click **Optimize**. A confirmation dialog appears.
4. Click **Yes**. A progress meter appears as the database is optimized.

To remotely administer databases:

1. Use the **Set Server Password** applet (installed with *Cyclone*) to set a password for local server.
2. From the **Configure** menu, select **Databases....** The **Configure Databases** dialog appears.
3. Select the database you wish to remotely administer, and click **Admin Login....**

4. Enter the password (set with the **Set Server Password** applet) from a remote client in order to administer (add, remove, etc.) databases on the remote database.

Note: To add a database on a remote server, the database file name must be entered using the server's drive letter and directories. For example, to add a database file name located at "D:\ Databases\ Example.imp" on the remote server "Remote", enter "D:\ Databases\ Example.imp" as the file name (not \\ REMOTE\ D\$\ Example.imp).

Datum

Window: ModelSpace

Menu: Tools | Measure

Action: Displays submenu

Usage: Pointing to **Datum** displays submenu commands used to measure the distance from a picked point to a specified datum.

The following commands are available from the **Datum** submenu:

[Point Distance](#), [Set Datum to Pick Point](#), [Enter Datum](#), [Show Datum](#)

- For **Datum** submenu command functionality, see the individual command entries.

Decimate Contours

Window: ModelSpace

Menu: Tools | Contours

Action: Opens dialog

Usage: The **Decimate Contours** dialog is used to reduce the number of vertices in a selected contour line.

- When *Cyclone* reduces the number of vertices in a contour line, it does so with the mandate of minimizing the deviation from the originating mesh (Deviation is measured perpendicular to the contour's plane).

To reduce the number of vertices in a contour line:

1. Select the contour you want to reduce.
 - If the contour is selected with pick points, only lines on the picked elevations are decimated. If the contour is selected by any other method (e.g., by a fence) all lines are decimated.
2. From the **Tools** menu, point to **Contours** then select **Decimate Contours....** Deviations are initialized and then the **Decimate Contours** dialog appears.
3. Set the maximum absolute deviation either by entering a value, or by moving the slider.
 - As this value is changed, the values for absolute deviation, maximum deviation, and minimum deviation, as well as the number of line segments, change accordingly and are displayed in

- the lower half of the dialog.
- To exit the dialog without decimating the contour line, click **Cancel**.
 - 4. Click **OK**. The number of vertices in the contour line is reduced, and the line is redrawn.

Decimate Mesh

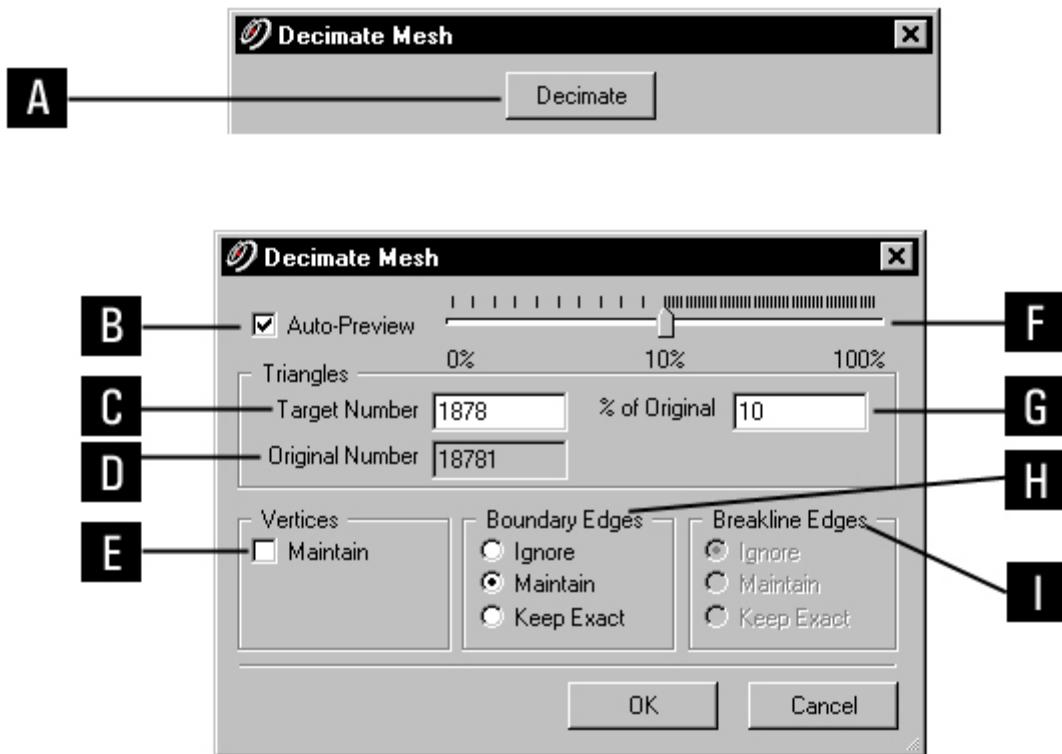
Window: ModelSpace

Menu: Tools | Mesh

Action: Opens dialog

Usage: The **Decimate Mesh** dialog is used to reduce the number of triangles in a mesh object.

Decimate Mesh Dialog



- A. Click to perform the decimation. When clicked this button is replaced by an Auto-Preview option and a slider bar (see B and F below).
- B. When selected, the display in the ModelSpace viewer is adjusted continually as the Decimation Percentage slider to the right of this option is moved.
- C. Displays the number of triangles resulting from movement of the slider or entry of a percentage in the **% of Original** field. You may also enter an exact number in this field. Note that entering the target number automatically adjusts the value in the **% of Original** field. Note also that the value in this field is automatically updated when the slider is used.

- D. Displays the original number of triangles in the selected mesh object.
- E. When selected, original vertices are maintained as much as possible. When unselected, vertices may be moved to maintain the overall shape of the mesh more accurately.
- F. Use this slider to adjust the percentage of decimation. If **Auto-Preview** is selected (see B), the viewer is automatically updated as you move the slider.
- G. **Displays** the percentage of the original number of triangles by which the mesh is to be decimated. You may also enter an exact number in this field. Note that entering a value in this field automatically adjusts the value in the **Target Number** field. Note also that the value in this field is automatically updated when the slider is used.
- H. When **Ignore** is selected, the integrity of the mesh object's boundary is not protected. When **Maintain** is selected, added weight is given to maintaining boundary edge shape during decimation. When **Keep Exact** is selected, the mesh object's boundaries remain intact, no matter how much the mesh is decimated.
- I. When **Ignore** is selected, breakline integrity is not protected. When **Maintain** is selected, added weight is given to maintaining breakline shape during decimation. When **Keep Exact** is selected, breaklines remain intact, no matter how much the mesh is decimated.

Mesh editing can be performed only on a *mesh object* as opposed to a point cloud that is viewed as a mesh. To create a mesh object from a point cloud, see the *Create Mesh* entry in this chapter.

To reduce the number of triangles in a mesh:

1. Select the mesh object that you want to reduce.
2. From the **Tools** menu, point to **Mesh**, and then select **Decimate Mesh**. The **Decimate Mesh** dialog appears.
3. Enter a decimation target either by triangle count in the **Target Number of Triangles** field, or by percentage of original in the **% of Original** field.
4. Select whether to maintain vertices and how *Cyclone* should deal with boundary edges and breaklines during decimation.
5. Click the **Decimate** button to preview the settings, and display a slider tool which enables you to further adjust the mesh
 - If **Auto-Preview** is selected, as the slider is adjusted the mesh is adjusted in real-time, reflecting the value represented.
6. Click **OK** when the mesh is displayed as desired. The mesh is replaced by a subsampled version of the original mesh.
 - To disregard changes and exit the dialog, click **Cancel**.

Note: If **Maintain** or **Ignore** check box is changed at any time while the dialog is open, the **Decimate** button reappears, as *Cyclone* needs to recompute the decimation.

Decimate Polyline/Patch

Window: ModelSpace

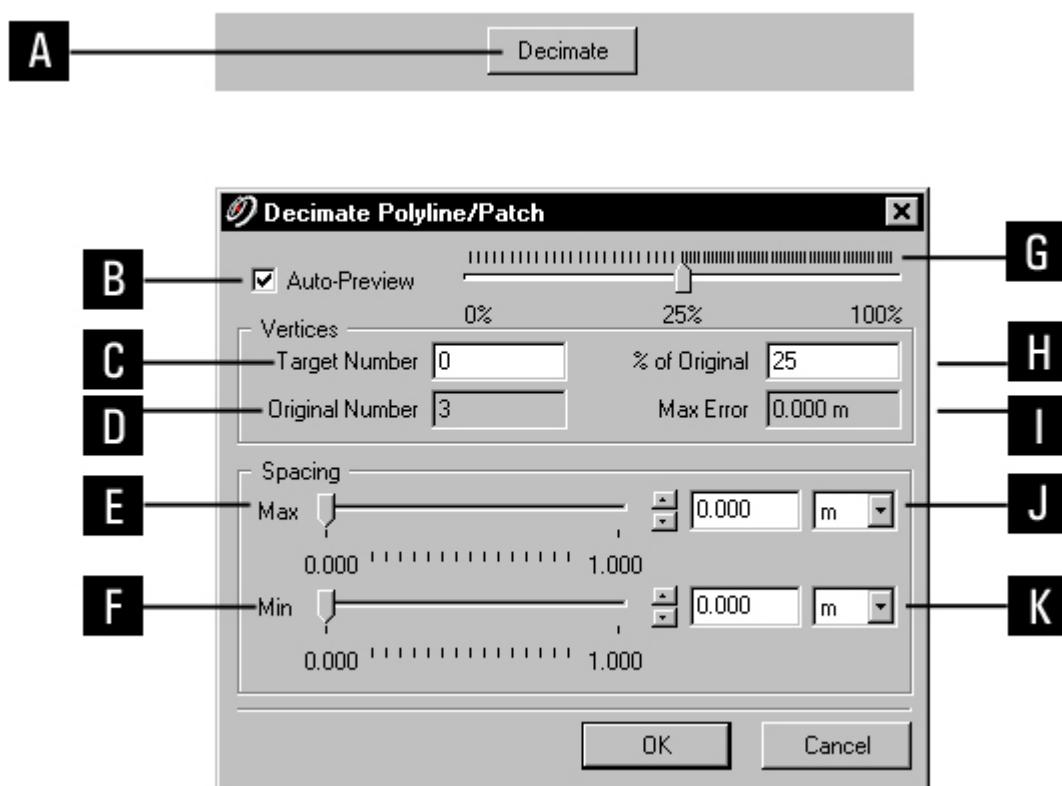
Menu: Edit Object

Action: Opens dialog

Usage: The **Decimate Polyline/Patch** dialog is used to intelligently reduce the number of vertices in a polyline or patch, preserving the original shape as much as possible.

- Polylines provide a potentially compact representation of cross-sections and other contours. However, some polylines may have vertices numbering in the hundreds (if not thousands), where tens of vertices may suffice.

Decimate Polyline/Patch Dialog



- Click to update the display in the ModelSpace viewer. When clicked, this button is replaced by the **Auto-Preview** option and a slider bar (see B and F below).
- When selected, the display in the viewer is adjusted continually as the **Decimation Percentage** slider to the right of this option is moved.
- Displays the number of vertices resulting from movement of the slider or entry of a percentage in the **% of Original** field. You may also enter an exact number in this field. Note that entering the target number automatically adjusts the value in the **% of Original** field. Note also that the value in this field is continually updated when the slider is used.

- D. Displays the original number of vertices in the selected polyline object.
- E. Use this slider to set a maximum distance between each vertex along the decimated polyline.
- F. Use this slider to set a minimum distance between each vertex along the decimated polyline.
- G. **Use** this slider to adjust the percentage of decimation. If **Auto-Preview** is selected (see B), the viewer is continually updated as you move the slider.
- H. Displays what percentage of the original number of vertices in the polyline object by which the polyline object is to be decimated. You may also enter an exact number in this field. Note that entering a value in this field automatically adjusts the value in the **Target Number** field. Note also that the value in this field is continually updated when the slider is used.
- I. Displays the largest distance measurement from any original vertex to the resulting (decimated) polyline.
- J. Enter a value for the maximum distance between each vertex along the decimated polyline.
- K. Enter a value for the minimum distance between each vertex along the decimated polyline.

To reduce the number of vertices in a polyline or patch:

1. Select the polyline or patch you want to reduce.
2. From the **Edit Object** menu, select **Decimate Polyline/Patch**. The **Decimate Polyline/Patch** dialog appears.
3. Set the decimation parameters.
 - To decimate by a percentage of original vertices, use the slider or enter a value in the **% of Original** field.
 - To decimate by constraining the range of distances between vertices, use the **Min** and **Max** sliders, and then click **Decimate**. This method produces spacing between vertices along the polyline such that adjacent vertices are no closer or further than desired.
 - To exit the dialog without decimating, click **Cancel**.
4. Click **OK**. The number of vertices in the polyline or patch is reduced, and the polyline or patch is redrawn.

Note: In some cases it can be difficult to get a satisfactory reduction with a polyline. It may be helpful to break the polyline into several pieces (using the **Create Object | Slice | by Last Selection** command) and reduce each piece separately.

Decrease Point Width

- Window:** **ControlSpace, ModelSpace, Registration, Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)**
- Menu:** **Point Cloud Rendering**
- Action:** Toggles menu item ON
- Usage:** The **Decrease Point Width** command decrements by one pixel the thickness of each cloud point displayed in the ModelSpace View.

Defect Report...

Window: All windows

Menu: Help

Action: Opens dialog

Usage: The **Defect Report** dialog is provided for users to report bugs, limitations, or other perceived defects in the *Cyclone* software.

To report a defect:

1. From the **Help** menu, select **Defect Report**.... The **Defect Report** dialog appears.
2. Enter personal data (name, company, email, phone, etc.) in the appropriate fields. It is important that you provide this data so we can contact you to follow up if necessary.
 - The **Date** and **Time** fields are populated automatically with the current date and time.
3. Enter a summary of the problem being reported in the **Summary** field, followed by a description of the problem in the **Detailed description** field. Provide as much detail as possible, such as what your actions were prior to the problem occurrence, whether you are able to duplicate the problem, etc.
 - When possible, include the text from any error message that appears. To copy the text from an error message, click the program icon in the top left corner of the message box and select **Copy**. The text can then be pasted into the defect report.
4. Check the **Include log files** option to attach any log files generated by *Cyclone*, along with information about your current computer configuration (recommended).
5. Click **Create Report** to open the **Save Report to File** dialog; enter a file name for the .txt file (or accept the default, "defect.txt"), select the folder into which you want to save the file, and click **Save**. The report is created and saved to the designated file and location.
6. E-mail the newly created .txt file to support@hds.leica-geosystems.com.

Delete

Window: ModelSpace, Navigator, Registration, Image Viewer

Menu: Edit

Action: Executes command

Usage: The **Delete** command removes a selected object *without* placing it on the *Cyclone* clipboard.

Delete (Limit Box Manager)

Window: Limit Box Manager

Menu: Limit Box

Action: Executes command

Usage: The **Delete** command deletes the selected limit box.

Delete Inside

Window: ModelSpace

Menu: Edit | Fence

Action: Executes command

Usage: The **Delete Inside** command removes all cloud points, polyline segments, and mesh faces within the data fence.

- For more information on working with fences, see the [Using a Fence](#) entry in the *Modeling* chapter.

Delete Line

Window: ModelSpace

Menu: Tools | Contours

Action: Executes command

Usage: This command removes selected contour lines from the ModelSpace without affecting the remaining contour lines.

Note: Attempting to delete a contour line using the **Delete** command in the **Edit** menu results in deleting the *entire set of contours*, rather than just the selected contour line.

To delete a contour line:

1. Pick the contour line to be deleted.
2. From the **Tools** menu, point to **Contours** and then select **Delete Line**. The selected contour line is deleted from the ModelSpace.

Delete Outside

Window: ModelSpace

Menu: Edit | Fence

Action: Executes command

Usage: The **Delete Outside** command removes all cloud points outside of a data fence.

Delete Selected Inside

Window: ModelSpace

Menu: Edit | Fence

Action: Executes command

Usage: The **Delete Selected Inside** command removes all points in selected point clouds within a fence.

- For more information on working with fences, see the [Using a Fence](#) entry in the *Modeling* chapter.

Delete Selected Outside

Window: ModelSpace

Menu: Edit | Fence

Action: Executes command

Usage: The **Delete Selected Outside** command removes all cloud points in selected point clouds outside of a data fence.

- For more information on working with fences, see the [Using a Fence](#) entry in the *Modeling* chapter.

Delete Selection

Window: ModelSpace

Menu: Tools | Mesh

Action: Executes command

Usage: The **Delete Selection** command deletes selected triangles from a mesh.

To delete triangles:

1. Multi-select the triangles in the mesh object to be deleted.
 - Mesh editing can be performed only on a *mesh object*, as opposed to a point cloud that is viewed as a mesh. To create a mesh object from a point cloud, see the *Create Mesh* entry in this chapter.
2. From the **Tools** menu, point to **Mesh**, and then select **Delete Selection**. The selected triangles are deleted.

Deselect

Window: ModelSpace, Image Viewer

Menu: Selection

Action: Executes command

Usage: The **Deselect** command clears the current selection.

- All objects can also be deselected by pressing **ESC**.

Deselect Fenced

Window: ModelSpace

Menu: Selection

Action: Executes command

Usage: The **Deselect Fenced** command deselects every object and point cloud within or overlapping the drawn data fence.

- For more information on working with fences, see the [Using a Fence](#) entry in the *Modeling* chapter.

Disable

Window: Registration

Menu: Constraint

Action: Executes command

Usage: The **Disable** command marks the selected constraint as OFF, so that it is not used when computing the registration.

- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.

To disable a constraint:

1. In either the **ScanWorlds' Constraints** or the **Constraint List** tab, select the constraint you want to disable.
2. From the **Constraint** menu, select **Disable**. The constraint is disabled and its status is marked as OFF.

Disassemble

Window: ModelSpace

Menu: Tools | Exclusion Volume

Action: Executes command

Usage: The **Disassemble** command replaces an exclusion volume with its original component(s).

To restore the contents of an exclusion volume:

1. Select the exclusion volume.
2. From the **Tools** menu, point to **Exclusion Volume**, and then select **Disassemble**. The original component(s) reappear.

Disconnect

Window: Scan Control

Menu: Scanner

Action: Executes command

Usage: The **Disconnect** command disconnects *Cyclone* from the currently connected scanner.

Disconnect (All)

Window: ModelSpace

Menu: Tools | Piping

Action: Executes command

Usage: The **Disconnect Piping (All)** command breaks all connections to the selected object.

- To connect piping, see the [Connect Piping](#) and [Piping Mode](#) commands.

Disconnect (Shared)

Window: ModelSpace

Menu: Tools | Piping

Action: Executes command

Usage: The **Disconnect Piping** command breaks the connections shared between two or more selected objects.

- This command breaks any shared connections between any two of the selected objects. All other connections are unaffected.
- To connect piping, see the [Connect Piping](#) and [Piping Mode](#) commands.

Display License

Window: Navigator, ModelSpace, Image Viewer, Registration, Scan Control

Menu: Help

Action: Opens dialog

Usage: This command displays a dialog box containing *Cyclone* licensing information. Licenses may be uninstalled from this dialog.

- To uninstall the licenses, click **Uninstall** and follow the prompts.

Distance

Window: ModelSpace

Menu: Tools | Measure

Action: Displays submenu

Usage: Pointing to **Distance** displays submenu commands used to measure a variety of linear distances.

The following commands are available from the **Distance** submenu:

[Point to Point](#), [Point to Centerline/Point](#), [Point to Unbounded Surface](#), [Center to Center](#), [Point to Scanner](#)

Distance of Point to Submenu Commands:

POINT TO POINT

Calculates the distance between picked points.

POINT TO CENTERLINE/POINT

- Calculates the distance between a picked point and the centerline or center point of the other selected object(s).
- The type of object selected as the second object determines whether the measurement is to a centerline or a center point. (For example, a sphere is measured to its center point and a cylinder is measured to its centerline.)

POINT TO UNBOUNDED SURFACE

Calculates the distance between the first picked point and the nearest intersection with the surface plane of the selected object.

- For measurements, surfaces are extended beyond the endpoint or boundary of the actual object to the nearest point of intersection. For example, a measurement to the surface of a patch is made to the nearest point on the underlying plane of the patch, which is not necessarily a point within the boundary of the patch.

CENTER TO CENTER

Calculates the distance between the center of the first selected object and the center of the second selected object.

- The object type determines whether the measurement is to a centerline or a center point. (For example, a sphere is measured to its center point, and a cylinder is measured to its centerline.)

POINT TO SCANNER

Calculates the distance between the picked point(s) in a scanned point cloud and the selected scanner.

- If a scanner object is not selected, *Cyclone* determines the distance from the selected point to the scanner that was used to capture the point.

To take a measurement:

1. Pick the point or object from which you want to measure a distance.
2. If necessary, multi-select a point, surface, or object (with a center point or centerline if required by the command) for the second point of the measurement.
3. From the **Tools** menu, point to **Distance**, and then select the appropriate command for the type of object to which you are measuring. The measurement is calculated and displayed.
 - If there are more than two points or objects selected, multiple measurements are taken from the first point or object to the subsequent selection items (i.e., first to second, first to third, etc.).
 - Center points, centerlines, and surfaces are measured differently depending on the type of object selected, and some measurements are not supported by all object types.

Download HDS6000 Scans...

Window: **Navigator**

Menu: **Tools**

Action: Executes command

Usage: The **Download HDS6000 Scans** command is used to download ZFS files from the HDS6000 that were not automatically downloaded. Only HDS6000 scans that were not automatically downloaded (as per the **HDS4500/HDS6000: Max # Points Auto-Imported** preference) are considered.

To download HDS6000 scans:

1. In the **Navigator** window, select the container under which HDS6000 Scans eligible for download are located.
 - The container can be a database, Project, ScanWorld, Registration, or Scan.
 - You can select multiple containers.
2. Turn on the HDS6000 and physically connect it to the network.
3. This command does not need the Scan Control to be connected to the scanner.
3. In the **Navigator** window on the **Tools** menu, select **Download HDS6000 Scans....**
4. If there are multiple HDS6000 scanners configured on the computer, select the specific HDS6000 from which to download, using the **Download Scans from HDS6000** dialog that appears in this case.
5. The ZFS files are downloaded from the HDS6000.
6. Use the **Re-Import Scans...** command to import the downloaded scans.

Draw Arc (3 Points on Arc)

Window: **ModelSpace**

Menu: **Tools | Drawing**

Action: Executes command

Usage: The **Draw Arc (3 Points on Arc)** command enters drawing mode and sets the drawing tool to draw an arc in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw an arc:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Arc (3 Points on Arc)**. The **Drawing** cursor appears.
2. In the ModelSpace, click once to place a vertex that serves as the first endpoint.
3. Click again to place a second vertex that serves as a point on the arc.
4. Click a point or drag the cursor to place a third vertex that serves as the second end point.
- A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
5. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The arc is drawn, and you are ready to draw another arc.
- To delete the newly created arc, either select Undo from the **Edit** menu, or select the arc, and then select **Delete** from the **Edit** menu.

Draw Arc (Start, Center, End)

Window: **ModelSpace**

Menu: **Tools | Drawing**

Action: Executes command

Usage: The **Draw Arc (Start, Center, End)** command enters drawing mode and sets the drawing tool to draw an arc in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw an arc:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Arc (Start, Center, End)**. The **Drawing** cursor appears.
2. In the ModelSpace, click once to place a vertex that serves as the first end point.
3. Click again to place a second vertex that serves as the center point.
4. Click a point or drag the cursor to place a third vertex that serves as the second end point.
- A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
5. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The arc is drawn, and you are ready to draw another arc.
- To delete the newly created arc, either select Undo from the **Edit** menu, or select the arc, and then select **Delete** from the **Edit** menu.

Draw Arc (Start, End, Center)

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Arc (Start, End, Center)** command enters drawing mode and sets the drawing tool to draw an arc in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw an arc:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Arc (Start, End, Center)**. The **Drawing** cursor appears.
2. In the ModelSpace, click once to place a vertex that serves as the first end point.
3. Click again to place a second vertex that serves as the second end point.
4. Click a point or drag the cursor to place a third vertex that serves as the center point.
- A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
5. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The arc is drawn, and you are ready to draw another arc.
- To delete the newly created arc, either select Undo from the **Edit** menu, or select the arc, and then select **Delete** from the **Edit** menu.

Draw Circle (3 Points on Circle)

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Circle (3 Points on Circle)** command enters drawing mode and sets the drawing tool to draw a circle in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.
-

To draw a circle:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Circle (3 Points on Circle)**. The **Drawing** cursor appears.
2. In the ModelSpace, click once to place a vertex at a point on the circle.
3. Click again to place a second vertex on the circle.
4. Click a point or drag the cursor to place a third vertex through which the circle must pass.
- A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
5. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The circle is drawn, and you are ready to draw another circle.

- To delete the newly created circle, either select Undo from the **Edit** menu, or select the circle, and then select **Delete** from the **Edit** menu.

Draw Circle (Point, Center)

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Circle (Point, Center)** command enters drawing mode and sets the drawing tool to draw a circle in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw a circle:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Circle (Point, Center)**. The **Drawing** cursor appears.
2. In the ModelSpace, click once to place a vertex at a point on the circle.
3. Click again or drag the cursor to place a vertex that serves as the center point.
- A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
4. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The circle is drawn, and you are ready to draw another circle.
- To delete the newly created circle, either select Undo from the **Edit** menu, or select the circle, and then select **Delete** from the **Edit** menu.

Draw Cubic Spline

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Cubic Spline** command enters drawing mode and sets the drawing tool to draw a cubic spline in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.
- The procedures for drawing a cubic spline are identical to those for drawing a polyline, except that the vertices are connected by the curves of a cubic spline curve rather than by straight lines. Also, vertices cannot be added between existing vertices while drawing a cubic spline.

Draw Cubic Spline Loop

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Cubic Spline Loop** command enters drawing mode and sets the drawing tool to draw a cubic spline loop in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.
- The procedures for drawing a cubic spline loop are identical to those for drawing a cubic spline, except that the first and last vertices are automatically connected.

Draw Ellipse

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Ellipse** command enters drawing mode and sets the drawing tool to draw an ellipse in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw an ellipse:

- An ellipse is drawn by first placing two vertices, and then placing a third vertex to adjust the eccentricity of the ellipse. The midpoint between the first two vertices becomes the center of the ellipse that passes through both points.
1. From the **Tools** menu, point to **Drawing**, and then select **Draw Ellipse**. The **Drawing** cursor appears.
 2. In the ModelSpace, click once to place a vertex.
 3. Click again to place a second vertex. The midpoint between the first two vertices becomes the center of a circle that passes through both vertices.
 4. Click a point or drag the cursor to place a third vertex, which adjusts the eccentricity of the ellipse.
 - The third vertex can be moved by dragging or clicking in a new location.
 - A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
 5. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The ellipse is drawn, and you are ready to draw another ellipse.
 - To delete the newly created ellipse, either select Undo from the **Edit** menu, or select the ellipse, and then select **Delete** from the **Edit** menu.

Draw Isometric Polygon

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Isometric Polygon** command enters drawing mode and sets the drawing tool to draw a polygon with a user-specific number of symmetric sides in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw an isometric polygon:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Isometric Polygon**. The **Drawing** cursor appears.
 2. In the ModelSpace, click once to place a vertex at the center of the isometric polygon. The **Isometric Polygon** dialog appears.
 3. Enter the number of sides to be drawn for the isometric polygon, and then click **OK**.
 4. Click again to place one vertex on the polyline. The other vertices are drawn automatically.
 5. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The isometric polygon is drawn, and you are ready to draw another.
- To delete the newly created isometric polygon, either select Undo from the **Edit** menu, or select the isometric polygon, and then select **Delete** from the **Edit** menu.

Draw Line

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Line** command enters drawing mode and sets the drawing tool to draw a line segment in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw a line:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Line**. The **Drawing** cursor appears.
 2. In the ModelSpace, click once to place a vertex that serves as first endpoint.
 3. Click again to place a vertex for the second endpoint.
 - The second vertex may be moved by dragging or clicking in a new location.
 - A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
 4. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The line is drawn, and you are ready to draw another line.
- To delete the newly created line, either select Undo from the **Edit** menu, or select the line, and then select **Delete** from the **Edit** menu.

Draw Polygon

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Polygon** command enters drawing mode and sets the drawing tool to draw a polygon in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.
- The procedures for drawing a polygon are identical to those for drawing a polyline, except that the first and last vertices are automatically connected.

Draw Polyline

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Polyline** command enters drawing mode and sets the drawing tool to draw a polyline in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw a polyline:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Polyline**. The **Drawing** cursor appears.
2. In the ModelSpace, click once to place a vertex at the first endpoint.
3. Click again to place a vertex and create a line segment from the previous point.
 - To continuously add vertices as you move the mouse, press and hold **ALT**, and drag the cursor.
4. Continue adding vertices as needed.
 - A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
 - Vertices may also be added by clicking on an existing line segment.
5. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The polyline is drawn, and you are ready to draw another polyline.
 - To delete the newly created polyline, either select **Undo** from the **Edit** menu, or select the polyline, and then select **Delete** from the **Edit** menu.

Draw Rectangle

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Rectangle** command enters drawing mode and sets the drawing tool to draw a square in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw a rectangle:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Rectangle**. The **Drawing** cursor appears.
2. In the ModelSpace, click once to place a vertex on one corner of the rectangle.
3. Click again to place a vertex at the second corner of the rectangle.
4. Click at the desired distance from either side of the line segment to determine the length and direction of the sides parallel to the first segment. The angle and length of the fourth side matches the first line segment.
 - This segment can be adjusted by dragging or clicking in a new location.
 - A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
5. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The rectangle is drawn, and you are ready to draw another rectangle.
- To delete the newly created rectangle, select **Undo** from the **Edit** menu, or select the rectangle, and then select **Delete** from the **Edit** menu.

Draw Square

Window: ModelSpace

Menu: Tools | Drawing

Action: Executes command

Usage: The **Draw Square** command enters drawing mode and sets the drawing tool to draw a square in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw a square:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Square**. The **Drawing** cursor appears.
2. In the ModelSpace, click once to place a vertex on one corner of the square.
3. Click again to place a vertex at the second corner of the desired square. This line segment determines the length of the sides of the square.
4. Click on either side of the line segment to place the remaining three sides. The angle and length of the remaining three sides are determined by the first line segment.

- The final three sides can be moved by dragging either vertex of the first line segment.
- A vertex can be moved by dragging it to a new position, or deleted by clicking on it without moving the cursor.
- 5. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The square is drawn, and you are ready to draw another square.
- To delete the newly created square, either select Undo from the **Edit** menu, or select the square, and then select **Delete** from the **Edit** menu.

Draw Text...

Window: ModelSpace

Menu: Tools | Drawing

Action: Opens dialog

Usage: The **Draw Text** dialog is used to enter drawing mode and add text in the active Reference Plane.

- For more information on drawing, see the [Drawing](#) entry in the *Modeling* chapter.

To draw text:

1. From the **Tools** menu, point to **Drawing**, and then select **Draw Text....** The **Drawing** cursor appears.
2. In the ModelSpace, click to select a point to anchor the text. The **2D Drawing : Text** dialog appears.
3. In the **Text to be Drawn** field, type the text you want, and click **OK**.
 - The text can be moved by dragging or clicking in a new location.
4. From the **Tools** menu, point to **Drawing**, and then select **Create Drawing**. The text is placed, and you are ready to add more text.
 - To view the newly created text, the **Show Annotations** command must be **ON**.
 - To delete the newly created text, either select Undo from the **Edit** menu, or select the text, and then select **Delete** from the **Edit** menu.

Drawing

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Drawing** displays submenu commands used to access a variety of drawing tools and settings.

The following commands and dialogs are available from the **Drawing** submenu:

[Create Drawing](#), [Cancel Drawing](#), [Draw Line](#), [Draw Polyline](#), [Draw Square](#), [Draw Rectangle](#), [Draw Isometric Polygon](#), [Draw Polygon](#), [Draw Text...](#), [Draw Arc \(3 Points on Arc\)](#), [Draw Arc](#)

[\(Start, End, Center\)](#), [Draw Arc \(Start, Center, End\)](#), [Draw Circle \(3 Points on Circle\)](#), [Draw Circle \(Point, Center\)](#), [Draw Ellipse](#), [Draw Cubic Spline](#), [Draw Cubic Spline Loop](#), [Align Vertices to Axes](#), [Set Distance to Next Vertex...](#), [Edit Drawing Parameters...](#), [Create Drawing from Object](#), [Export Template](#)

- For complete more information, see the individual entries in this chapter or the [Drawing](#) entry in the *Modeling* chapter.

Eccentric Reducer Connectors

Window: ModelSpace

Menu: Tools | Piping

Action: Executes command

Usage: The **Eccentric Reducer Connectors** command connects two parallel selected pipes via a reducer that has one perpendicular leg.

To connect parallel pipes:

1. Pick a point on the first pipe near the end you want connected.
2. Multi-select the second pipe near the end that you want connected.
3. From the **Tools** menu, point to **Piping**, and then select **Reducer Connectors**. The pipes are connected with a reducer.

Edit Active Plane...

Window: ModelSpace

Menu: Tools | Reference Plane

Action: Opens dialog

Usage: The **Edit Active Plane...** command launches the **Reference Plane Parameters** dialog, which is used to set various display and orientation options for the active Reference Plane.

Note: The active Reference Plane may not be edited while a 2D drawing is in progress.
Reference Plane Parameters Dialog Options:

PLANE NORMAL

Set direction vector used for the plane's normal (the Reference Plane's positive Z axis).

PLANE ORIGIN

Set coordinates for the plane's center.

PLANE CROSS AXIS

Enter a direction vector for the cross axis (the Reference Plane's positive X axis).

GRID SPACING

Enter the desired distance between grid lines.

GRID LINE WEIGHT

Enter a value for the thickness of grid lines.

GRID COLOR

Use the color picker to select the color used for the grid.

Edit Clipping Planes...

Window: ModelSpace

Menu: View

Action: Opens dialog

Usage: The **Clipping Planes** dialog is used to set the near and far boundaries of what is visible in the ModelSpace View.

- Clipping planes are the boundaries that determine the visible volume of a scene.
- The six default boundaries are the top, bottom, left, right, near and far clipping planes. The top, bottom, left, and right planes are at the corresponding edges of the ModelSpace window; the near and far clipping planes automatically update to contain the scene, but may be adjusted by the user.

To set near and far clipping planes:

1. From the **View** menu, select **Edit Clipping Planes...**. The **Clipping Planes** dialog appears.
2. Choose values for the near and far clipping planes.
 - To set custom clipping planes, select the **Use Custom Clipping Planes** box, and then enter values in the **Near Clipping Plane** and **Far Clipping Plane** boxes.
 - To reset near and far clipping planes to their default values, select the **Use Default** box, and then click **Apply**.
3. Click **OK**. The near and far clipping planes are set.

Edit Color Map...

Window: ModelSpace

Menu: Edit Object | Appearance

Action: Opens dialog

Usage: The **Color Map Parameters** dialog is used to specify color and range settings for the selected color map.

- For an overview of color mapping in *Cyclone*, see the [Color Mapping](#) entry in the *Modeling* chapter.
- Settings made in this dialog are overridden if [Global Color Map](#) is toggled ON.
- The **Minimum** and **Maximum** settings typically range from 0 to 1. These values correspond to

the range from dark to bright (red to blue for the **Multi-Hue** setting). For a greater difference in shades, set a narrower range.

Color Map Parameters Dialog Options

MODE

From the drop-down list, select **Image Texture Map**, **Colors from Scanner**, **Intensity Map**, **Elevation Map**, or **Single Color**.

SCHEME

From the drop-down list, select **Multi-Hue/Rainbow**, **Grayscale**, a topography-based palette, or a scaled individual color with which to display the intensity or elevation map (for the corresponding **Mode**). If the **Mode** is **Image Texture Map** or **Colors from Scanner**, this scheme is used when the cloud/mesh lacks per-point color.

NUMBER OF COLORS

Enter a value for the total number of colors that will be used by the color scheme.

REPEAT COLORS OUTSIDE RANGE

Select this to use a cyclic color scheme. For example, if the **Minimum** is 0 and the **Maximum** is 1, a value of 1.3 would be same color as a value of 0.3. Otherwise, values lower than the **Minimum** (or **Start**) are given the color at the low end of the selected color scheme, and values higher than the **Maximum** are given the color at the high end of the selected color scheme.

GAMMA CORRECTION

Select this to enable gamma correction, which can accentuate subtle differences in intensities. A value of 1.0 is equivalent to disabling gamma correction.

APPLY TO

Select what the parameters affect.

When Mode is **Image Texture Map**, **Colors from Scanner** or **Intensity Map**:

MINIMUM

Enter a value for the lowest intensity to be mapped to the low end of the selected color scheme.

MAXIMUM

Enter a value for the highest intensity to be mapped to the high end of the selected color scheme.

ACTUAL

Displays the object's original minimum and maximum intensity values.

When Mode is **Elevation Map**:

START

Enter a value for the elevation to be mapped to the low end of the selected color scheme.

DELTA

Enter a value for the difference in elevation represented by each color in the selected color scheme (see **Number of Colors** earlier).

Edit Color/Material...

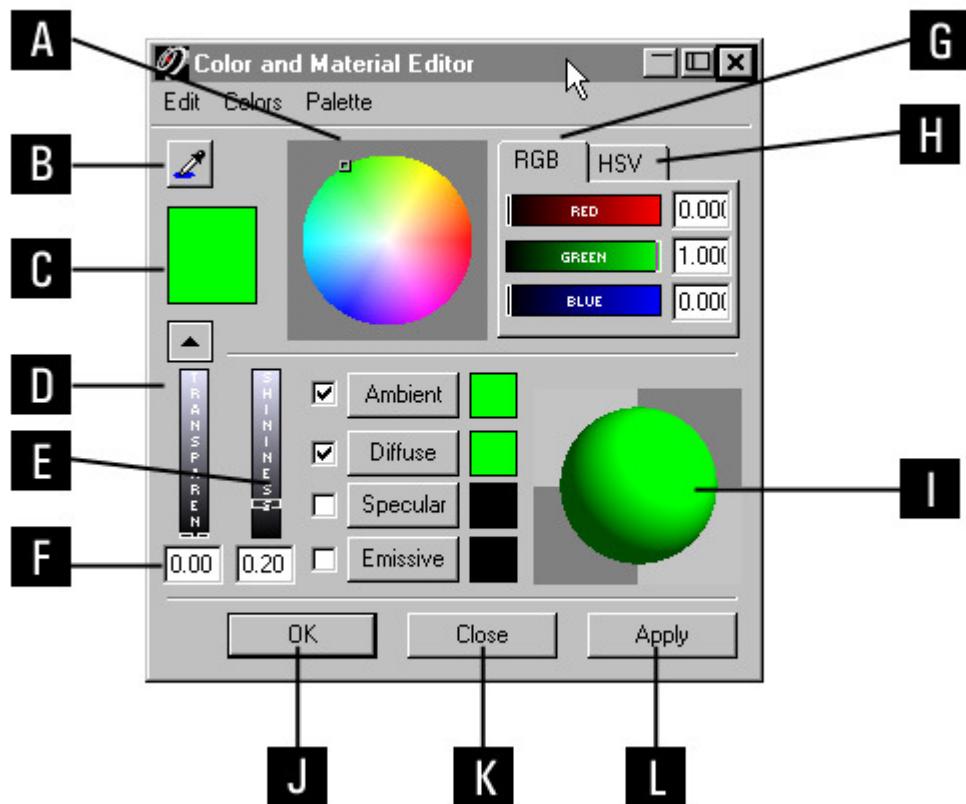
Window: ModelSpace

Menu: Edit Object | Appearance

Action: Opens dialog

Usage: The **Color and Material Editor** dialog is used to change the appearance of the selected object.

Color and Material Editor Dialog



A. The color wheel allows you to select hue and saturation simultaneously.

B. Click and drag the color picker to select any color on your screen (including those outside of *Cyclone*). The selected color appears in the current color swatch.

- When the current color swatch is changed, any selected material swatches are also changed.
 - The current color swatch can be edited by using the color picker, the color wheel, the **HSV** and **RGB** sliders, or any of the four material buttons (click the button to copy the color in the material's swatch to the current color swatch).
- C.** The current color swatch displays the color settings from the color wheel and the **HSV** and **RGB** sliders. When a change is made to any of the **HSV** or **RGB** sliders, the current color swatch, or the color wheel, the change is reflected in all three areas.
- D.** Open/Close the lower portion of the Color and Material Editor dialog, which contains material editing controls.
- When the lower portion is closed, color changes affect only diffuse and ambient properties (equally).
- E.** Use this slider to adjust the amount of shine or glare that an object reflects.
- This setting interacts with the **Specular** setting. Values range from 0 (dull) to 1 (glossy).
- F.** Use this slider to adjust the degree to which you can see through an object. Values range from 0 (opaque) to 1 (invisible).
- G.** Click to access sliders that edit the mix of red, green, and blue that define the current color. Values range from 0 to 1.
- H.** Click to access sliders that edit the mix of hue, saturation, and value that define the current color. Values range from 0 to 1.
- I.** Displays the current material.
- J.** When checked, the associated material property is linked to the current color swatch so that any changes made to the current color are duplicated in the selected materials.
- K.** Click to copy the material property to the current color swatch, which in turn also changes all selected material swatches.
- **Ambient** is the color reflected by an object in response to the ambient lighting. See Edit Environmental Lights entry in this chapter.
 - **Diffuse** is the base color of an object in response to a white light source.
 - **Specular** is the color reflected in a surface's highlights.

- This setting interacts with the **Shininess** setting to determine the overall highlighting effect of the material.
- Emissive is the color generated by the object. It does not need an external light source, but does not necessarily emit light.

L. Material swatches display the color of their various properties. Right click to use the slider to adjust individual brightness.

- When the current color swatch is changed, any selected material swatches are also changed.

Color and Material Editor Menus

The **Edit** menu contains Copy and Paste commands:

Copy: Copies the current color to the *Cyclone* color clipboard.

Paste: Copies the color from the *Cyclone* color clipboard to the current color swatch and any selected material swatch(es).

The **Colors** menu contains commands that assign standard colors and values to the current color swatch and any selected material swatches.

The **Palette** menu contains commands used to open the Color Palette and the Material Palette (see the Add/Edit Colors and Add/Edit Materials entries in this chapter).

To edit an object's color and material:

1. Select the object whose color and material you want to change.
2. From the **Edit Object** menu, point to **Appearance**, and then select **Edit Color/Material...**. The **Color and Material Editor** appears.
3. Make the desired changes, and then click **OK**. The selected object's color and material are updated.

Edit Drawing Parameters...

Window: ModelSpace

Menu: Tools | Drawing

Action: Opens dialog

Usage: The **Drawing Parameters** dialog is used to set the color and line weight parameters for 2D drawing.

- Changes in the **Drawing Parameters** dialog are reflected in current and subsequent 2D drawings only. 2D drawings created prior to any changes are not affected.

To set default drawing parameters:

1. From the **Tools** menu, point to **Drawing**, and then click **Edit Drawing Parameters....** The **Drawing Parameters** dialog appears.
2. Edit the line weight and color parameters.

Edit Elbow BendRatio...

Window: ModelSpace

Menu: Tools | Piping

Action: Opens dialog

Usage: The **Edit Elbow Bend Ratio** dialog is used to edit the ratio of elbow bend radius to pipe OD (outside diameter) for the selected pipes and elbows.

To adjust the elbow bend ratio:

1. Select the objects that you want to edit in the following order: pipe, (joining) elbow, pipe.
2. From the **Tools** menu, point to **Piping**, and then select **Edit Elbow Bend Ratio....** The **Elbow Bend Ratio** dialog appears.
3. Enter a value for the ratio of elbow bend radius to pipe OD.
4. Click **OK** to complete the operation.

Edit Environmental Lights...

Window: ModelSpace

Menu: Tools | Lighting

Action: Opens dialog

Usage: The **Edit Environmental Lights** dialog is used to adjust Head Light and Ambient Light settings.

- For more information about lighting, see the [Lighting](#) entry in the *Modeling* chapter.

Light Settings Dialog Options (Head Light tab):

HEAD LIGHT COLOR

Use the color picker to assign a color to the Head Light.

HEAD LIGHT INTENSITY

Set the Head Light intensity from 0 (dark) to 1 (bright).

HEAD LIGHT STATE

Turn the Head Light ON or OFF.

Light Settings Dialog Options (Ambient Light tab):

AMBIENT LIGHT COLOR

Use the color picker to assign a color to the Ambient Light.

AMBIENT LIGHT INTENSITY

Set the Ambient Light intensity from 0 (dark) to 1 (bright).

To adjust Light Settings:

1. From the **Tools** menu, point to **Lighting**, and then select **Edit Environmental Lights....** The **Light Settings** dialog appears.
2. Make the changes you want to the various settings.
 - To return all options to factory default settings, click **Reset**.
 - 3. Click **OK** to complete the operation.
 - To see the changes immediately, click **Apply**.
 - To exit without saving unapplied changes, click **Close**. Changes that have been applied are already saved.

Edit Global Color Map...

Window: ModelSpace

Menu: Edit Object | Appearance

Action: Opens dialog

Usage: The **Global Color Map** dialog is used to specify color and range settings for the global color map.

- For an overview of color mapping in *Cyclone*, see the [Color Mapping](#) entry in the *Modeling* chapter.
- The **Minimum** and **Maximum** settings typically range from 0 to 1. These values correspond to the range from dark to bright (red to blue for the **Multi-Hue** setting). For a greater difference in shades, set a narrower range.

Color Map Parameters Dialog Options:

MODE

From the drop-down list, select **Image Texture Map**, **Colors from Scanner**, **Intensity Map**, **Elevation Map**, or **Single Color**.

SCHEME

From the drop-down list, select **Multi-Hue/Rainbow**, **Grayscale**, a topography-based palette, or a scaled individual color with which to display the intensity or elevation map (for the corresponding **Mode**). If the **Mode** is **Image Texture Map** or **Colors from Scanner**, this scheme is used when the cloud/mesh lacks per-point color.

ENABLE GLOBAL COLOR MAP

Select this to apply the global color map to all clouds and meshes in the ModelSpace, overriding individual color map parameters.

NUMBER OF COLORS

Enter a value for the total number of colors that will be used by the color scheme.

REPEAT COLORS OUTSIDE RANGE

Select this to use a cyclic color scheme. For example, if the **Minimum** is 0 and the **Maximum** is 1, a value of 1.3 would be same color as a value of 0.3. Otherwise, values lower than the **Minimum** (or **Start**) are given the color at the low end of the selected color scheme, and values higher than the **Maximum** are given the color at the high end of the selected color scheme.

GAMMA CORRECTION

Select this to enable gamma correction, which can accentuate subtle differences in intensities. A value of 1.0 is equivalent to disabling gamma correction.

When Mode is **Image Texture Map, Colors from Scanner or Intensity Map**:

MINIMUM

Enter a value for the lowest intensity to be mapped to the low end of the selected color scheme.

MAXIMUM

Enter a value for the highest intensity to be mapped to the high end of the selected color scheme.

ACTUAL

Displays the object's original minimum and maximum intensity values.

When Mode is **Elevation Map**:

START

Enter a value for the elevation to be mapped to the low end of the selected color scheme.

DELTA

Enter a value for the difference in elevation represented by each color in the selected color scheme. (See **Number of Colors** earlier.)

Edit Measurements...

Window: ModelSpace

Menu: Tools | Measure

Action: Opens dialog

Usage: The **Edit Measurements** dialog is used to view, save, delete, and copy measurements to the Windows clipboard.

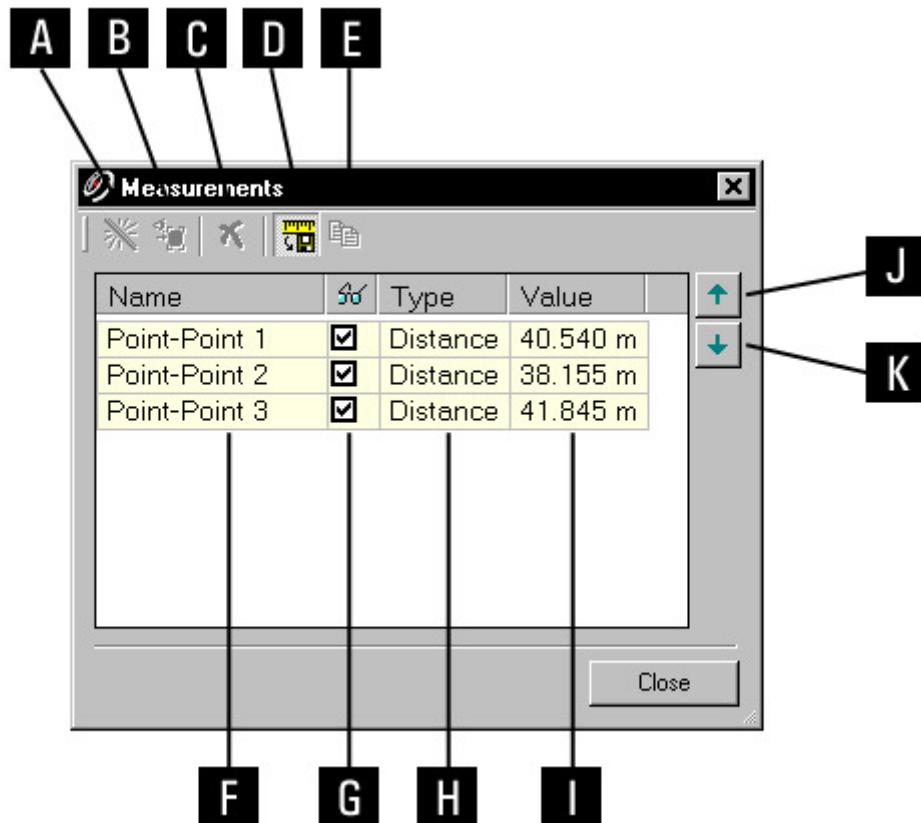
To customize the Edit Measurements dialog:

1. Right-click anywhere in the column header and select **Customize** from the context menu. The **Customize Columns** dialog appears.
2. Select the columns of information you wish to display in the **Edit Measurements** dialog, and then click **Exit**. The dialog's columns are customized to your specifications.
- Note that additional information columns such as the delta X, Y, Z, and Horizontal, Vertical, and projected measurements are *not* displayed by default.

To copy Measurement values to the Windows clipboard:

1. Select the measurement(s) you want to copy.
2. Click the **Copy to Clipboard** icon. The measurement value(s) are copied as tab-delimited text to the Windows clipboard, from which it may be pasted into another application (e.g., Notepad).

Measurements Dialog



- A. Highlight the selected measurement in the ModelSpace window.
- B. Center the selected measurement in the ModelSpace window.
- C. Delete the selected measurement.

D. When the button is toggled OFF, each new measurement replaces the prior in the ModelSpace and no measurement data is added to this dialog. When the button is toggled ON, all measurement data taken while the button is active, is displayed in this dialog, and all measurements are displayed in the ModelSpace (unless individual measurements are made invisible – see F below).

E. Copy selected data to the clipboard.

F. List of measurements in current ModelSpace

G. Toggle visibility of the selected measurements.

H. Measurement type.

I. Measurement value.

J. Move the selected measurement up in the list.

K. Move the selected measurement down in the list.

Edit Parameters... (Cloud Registration)

Window: Registration

Menu: Cloud Constraint

Action: Opens dialog

Usage: The **Edit Parameters...** command opens the **Cloud Constraint Parameters** dialog, which allows the cloud constraint's individual behavior to override the relevant preferences from the **Registration** tab of the **Edit Preferences** dialog.

- See the [Preferences](#) command for details.

To restore the previous values, click **Reset**.

To change the cloud constraint to no longer override the preferences, click **Default**.

- For more information on the cloud registration process, see the *Cloud Registration* section of the *Registration* chapter.

Edit Parameters... (Region Grow)

Window: ModelSpace

Menu: Create Object | Region Grow

Action: Opens dialog

Usage: The **Edit Parameters** command opens the **Modeling** tab within the **Edit**

Preferences dialog.

Edit Point Properties...

Window: Image Viewer

Menu: Annotation

Action: Opens dialog

Usage: The **Edit Point Properties** dialog is used to change the color and location of existing annotation pick points.

To edit pick point properties in the Image Viewer:

1. Select the pick point you want to edit.
 - You must be in **Pick** mode in order to select pick points; in addition, the selected point must have at least one assigned annotation.
2. From the **Edit** menu, select **Edit Point Properties**. The **Image Point Properties** dialog appears.
3. Make the changes you want for the color and position of the selected pick point.
 - To edit the pick point's color, click **Color** and use the color picker to select the color.
 - To edit the pick point's position, click **Position** and enter values for X and Y coordinates. The range of values is based on the number of pixels in the image.
4. Click **OK**. Your color and position changes are made.

Edit Point/Spot Lights...

Window: ModelSpace

Menu: Tools | Lighting

Action: Opens dialog

Usage: The **Edit Point/Spot Lights...** command opens the **Light Properties** dialog, which is used to set lighting attributes for the selected point or spot light.

- For more information about lighting, see the [Lighting](#) entry in the *Modeling* chapter.

Light Properties Dialog Options

COLOR

Select the amounts of red, green, and blue that make up the color of the light emitted.

ATTENUATION TYPE

Select **Constant** (light does not fade over distance), **Linear** (light fades evenly over distance), or **Quadratic** (light fades more quickly as distance from the source increases).

ATTENUATION FACTOR

Adjust the factor used by the **Linear** and **Quadratic** attenuation settings.

ORIGIN

Adjust X, Z, and Y coordinates for the origin of the light source.

DIAMETER (POINT LIGHT ONLY)

Adjust the diameter of the point light sphere. It can be helpful to match the scale of your lights to that of the model.

- The diameter of a point light does not affect the light that it casts, only its visualization and ease of picking.

ANGLE (SPOT LIGHT ONLY)

Adjust the spot light's cut-off angle, which is the angle out from the spot light's direction beyond which the light ends.

AXIS (SPOT LIGHT ONLY)

Adjust the X, Z, and Y vector for the spot light's direction.

HEIGHT (SPOT ONLY)

Adjust the height of the spot light. The height does not affect the light that it casts, only its visualization and ease of picking.

ON/OFF STATE

Turn the selected light **ON** or **OFF**.

To set point/spot light parameters:

1. Select the point or spot light you want to edit.
2. From the **Tools** menu, point to **Lighting**, and then select **Edit Point/Spot Lights....** The **Light Properties** dialog appears.
3. Make the changes you want to the various settings.
- To exit without saving changes, click **Cancel**.
4. Click **OK** to complete the operation.

Edit Properties...

Window: **ModelSpace**

Menu: **Edit Object**

Action: Opens dialog

Usage: The **Object Properties** dialog is used to edit the size and location of the selected object.

To edit object properties:

1. Select the object you want to edit.
2. From the **Edit Object** menu, select **Edit Properties**. The **Object Properties** dialog appears.
- The editable properties that appear depend on the type of object selected.

- If more than one object is selected, only those editable properties that are shared are available to be edited. In addition, if the shared properties of the selected objects differ, you must check **Value Override** within the property that you are editing in order to effect a change. For example, if you want to change the diameter of two selected cylinders of differing diameters, you must: Select both cylinders, open the **Edit Properties** dialog, open the **Diameter** tab, check the **Value Override** box, and then enter a new value. The diameters of both cylinders are changed to reflect the new value regardless of their original values.
3. Make the changes you want to the size, position, or other properties of the selected object. A transparent preview object appears in the ModelSpace View.
 - If the selected object supports the use of parts tables, you may make selections from the **Table** and **Item** fields. See the *Modeling* chapter for more information on parts tables.
 4. Click **Apply** to apply the changes and continue in the **Object Properties** dialog.
 5. Click on **Close** to close the **Object Properties** dialog. The dialog is closed.

Edit Properties (Limit Box Manager)

Window: Limit Box Manager

Menu: View

Action: Executes command

Usage: The **Edit Properties** command displays the Object Properties dialog for the selected limit box.

Note: This dialog differs from the Object Properties dialog accessed through the **ModelSpace Edit Object** menu because it only displays the properties of the selected limit box in the Limit Box Manager even if other objects in the ModelSpace are currently selected.

Edit Redlines...

Window: ModelSpace

Menu: Tools | Redlining

Action: Opens dialog

Usage: The **Redlines** dialog is used to manage and load saved redlines.

Redlines Dialog Buttons

LOAD

Loads the selected redline.

- Redlines are tied to a specific viewpoint. Loading a redline restores the viewpoint from which it was created. When the viewpoint changes, the redline is no longer visible.
- Loading a redline will toggle on Redline mode, allowing you to add to the loaded redline.
- The currently loaded redline set is denoted by "(active)" next to the name in the redline list.

- You can only load redlines that were created in the current ModelSpace View.

DELETE

Deletes the selected redline.

- The active Redline set cannot be deleted.

RAISE

Move the selected redline one position higher in the list.

LOWER

Move the selected redline one position lower in the list.

Edit Snap to Object Threshold...

Window: ModelSpace

Menu: Edit Object | Handles

Action: Opens dialog

Usage: The **Set Handle Snapping** dialog is used to set the distance at which handles snap to selected objects.

- For more information on using handles, see the [Handles](#) entry in the *Modeling* chapter.

To set the Snap to Object threshold:

1. From the **Edit Object** menu, point to **Handles**, and then select **Edit Snap to Object Threshold....** The **Set Handle Snapping** dialog appears.
2. In the **Snap Handle to Object Threshold** field, enter a value for the distance at which a handle snaps to a selected object.
3. Click **OK**. The threshold is set.

Editor...

Window: Scan Control

Menu: Scanner Control | Script

Action: Opens dialog

Usage: The **Editor** command launches the **Scanner Script** window, which is used to manage all scanner scripting functions.

Scanner Script Window Commands

SCRIPT MENU COMMANDS

Insert...

Insert the contents of a saved script.

Save

Save the contents of the current script.

RECORD MENU COMMANDS**Targeted Scan**

Record the target position and scanner settings to the current script.

Scan Around Picked Point

Record a detailed scan to the current script at a user-specified size and spacing around a 3D point picked in the connected ModelSpace viewer.

Targeted Image (HDS3000, ScanStation, and ScanStation 2 only)

Record the target image and settings to the current script.

CONTROL MENU COMMANDS**Start**

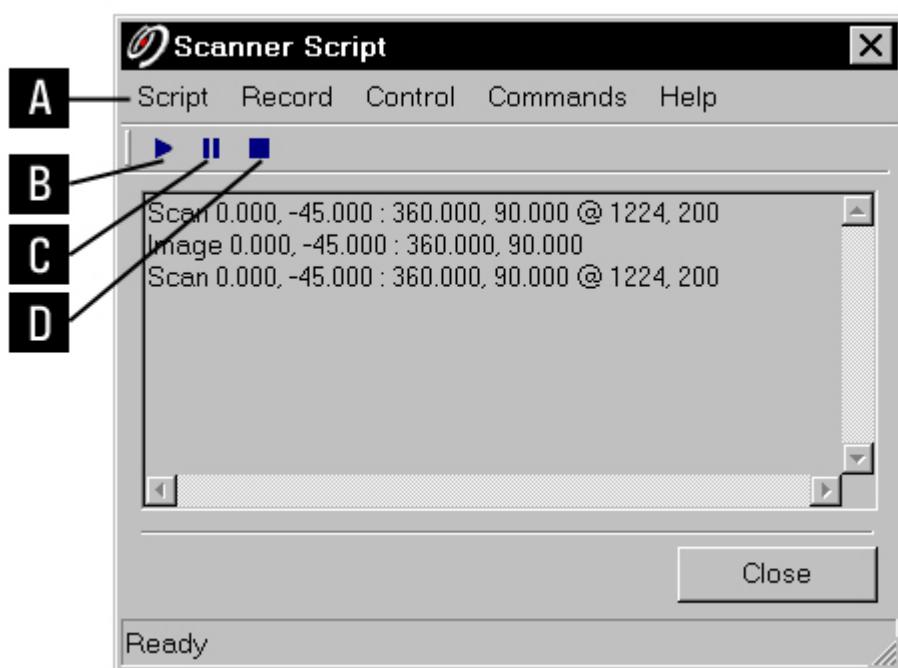
Run the current script.

Stop

Stop the currently running script.

Pause

Pause the currently running script.

Scanner Script Dialog

A. Script menus.

B. Run Script.

C. Pause Script.

D. Stop Script.

To add a targeted scan to the current script:

1. In the **Scan Control** window, set the target box and scan settings you want to add to the current script.
2. From the **Record** menu in the **Scanner Script** window, select **Targeted Scan**.

To insert a saved script into the current script:

1. From the **Script** menu in the **Scanner Script** window, select **Insert....** The **Select a Script** dialog appears, displaying a list of saved scripts.
2. Select the script you want to insert, and then click **OK**. The saved script is inserted into the **Scanner Script** window at the current cursor location.

To scan around a picked point:

1. From the **Scan Control** window, select **Open ModelSpace Viewer** or click **Open Viewer** to open a ModelSpace viewer.
2. In the ModelSpace, pick the point around which you want to scan.
3. From the **Record** menu in the **Scanner Script** window, select **Scan Around Pick Point**. The **Scan Around Pick Point** dialog appears.
4. Make the desired adjustments, and then click **OK**. The scan is added to the script.
 - The **Horizontal** and **Vertical Extent** settings represent the size of the area around the pick that you want to scan.
 - The **Horizontal** and **Vertical Points** settings represent the number of lines to be scanned and the number of points that you want in each line of the scan.

Elbow Connectors

Window: **ModelSpace**

Menu: **Tools | Piping**

Action: Executes command

Usage: The **Elbow Connectors** command connects two selected pipes that do not necessarily have intersecting centerlines.

To connect pipes with an elbow:

1. Pick a point on the first pipe near the end you want connected.
2. Multi-select the second pipe near the end that you want connected.
3. From the **Tools** menu, point to **Piping**, and then select **Elbow Connectors**. The pipes are

connected.

Elevation

Window: ModelSpace

Menu: Tools | Measure

Action: Executes command

Usage: The **Elevation** command calculates distance between picked points and the origin of the current coordinate system along the current “up” vector.

To measure elevation:

1. Pick the point you want to measure.
2. From the **Tools** menu, point to **Measure**, and then select **Elevation**. The elevation is displayed. The results are also displayed in the Output Box, if it is enabled.
 - If multiple points are being measured, the results are displayed in the order in which they were picked.

Elevation Check

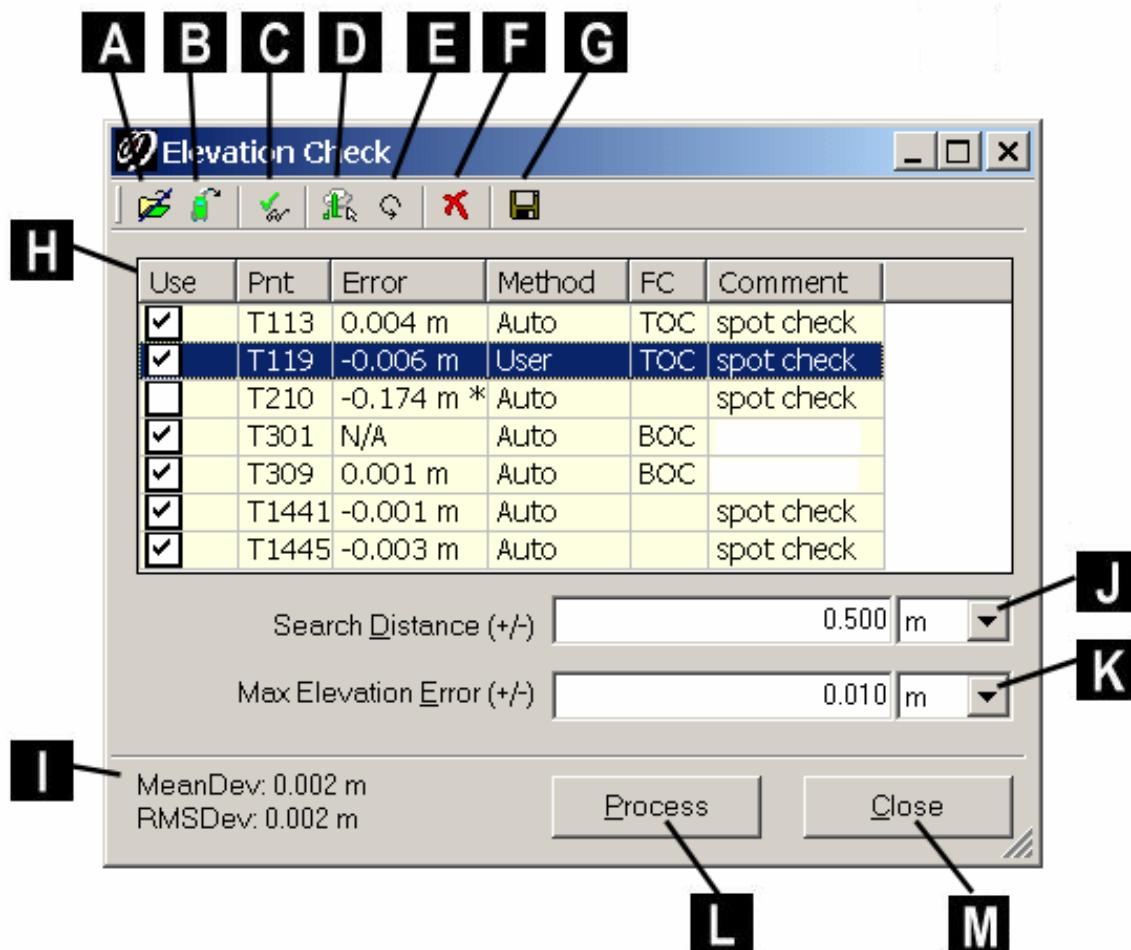
Window: ModelSpace

Menu: Tools | Registration

Action: Opens dialog

Usage: The **Elevation Check** dialog is used to measure the vertical distance between a QC (quality control) point and the cloud points in the vicinity. Use this command after Registration to verify that QC points (measured independently) on the ground are close in elevation to the neighboring cloud points.

Elevation Check Dialog



- A. Import the QC points to check from an ASCII or LandXML file.
- B. Import the QC points to check from a Leica System 1200 DB-X database.
- C. Hide QC points with **Error** less than the **Max Elevation Error**.
- D. Calculate the elevation for a selected QC point using a pick point. The **Method** becomes **User**.
- E. Reset the selected QC point's **Method** to **Auto**.
- F. Delete the selected QC point.
- G. Invoke the **Save Report** dialog to display information about the elevation check.
- H. Select **Use** to include the QC point in the statistics.

- I. The statistics for the elevation check.
- J. Measure the relative elevation of each QC point to the cloud points in the region around that QC point.
- K. A QC point with an **Error** greater in magnitude than the **Max Elevation Error** is marked with an asterisk in the **Error** column.
- M. Compute the difference in elevation between each QC point and the neighboring cloud points.
- N. Close the dialog.

Enable

Window: Registration

Menu: Constraint

Action: Executes command

Usage: The **Enable** command activates the selected constraint for use in the registration.

- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.

To enable a constraint:

1. In either the **ScanWorlds' Constraints** or the **Constraint List** tab, select the constraint you want to enable.
2. From the **Constraint** menu, select **Enable**. The constraint is enabled and its status is marked as "ON".

Enable Dual-Axis Compensator

Window: Scan Control (ScanStation, ScanStation 2)

Menu: Scanner

Action: Executes command

Usage: When **Enable Dual-Axis Compensator** is selected, the use of the dual-axis compensator is toggled.

Enable Output Box

Window: ModelSpace
Menu: View
Action: Executes command
Usage: When **Enable Output Box** is selected, all command output from the current ModelSpace (e.g., measurement results) is written to the Output Box.

- The output box can be viewed at any time by selecting **Show Output Box** from the **View** menu. If **Enable Output Box** was checked before you took the measurements, the measurement data is displayed in the output box in the order in which the measurements were taken.
- See the [Show Output Box](#) command for more information.

Enable Tilt Sensor

Window: Scan Control (HDS6000)
Menu: Scanner
Action: Executes command
Usage: When **Enable Tilt Sensor** is selected, the use of the tilt sensor is toggled.

End Caps

Window: ModelSpace
Menu: Edit Object
Action: Displays submenu
Usage: Pointing to **End Caps** displays submenu commands used to add and remove end caps of different types.

The following commands are available from the **End Caps** submenu:

[Add Flat Cap Closest to Pick](#), [Add Both Flat Caps](#), [Add Semi-Elliptical Head Closest to Pick](#), [Add Both Semi Elliptical Caps](#), [Remove Cap Closest to Pick](#), [Remove Both Caps](#)

Enter Datum...

Window: ModelSpace
Menu: Tools | Measure | Datum
Action: Opens dialog
Usage: The **New Datum** dialog is used to enter the X, Y, and Z coordinates for the point through which the datum passes.

To enter a new datum:

1. From the **Tools** menu, point to **Measure**, and then point to **Datum** and select **Enter Datum**. The **Set Datum Point** dialog appears.
2. Enter values for the X, Y, and Z coordinates, and then click **OK**. The datum is reset to pass through that point, perpendicular to the horizontal plane.

Exclusion Volume

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Exclusion Volume** displays submenu commands used to create and manipulate an exclusion volume.

- An exclusion volume is an object that encloses and replaces other objects in the ModelSpace. It is typically used to contain clouds of points for which you need only a rough approximation of the volume occupied.

The following commands are available from the **Exclusion Volume** submenu:

[Create](#), [Disassemble](#), [Collapse](#), [Minimize](#), [Keep Collapsed](#)

Exit

Window: Navigator

Menu: File

Action: Executes command

Usage: The **Exit** command shuts down the *Cyclone* program, including any open windows.

Explode

Window: ModelSpace

Menu: Create Object

Action: Executes command

Usage: The **Explode** command creates individual components from a selected object.

To explode an object:

1. Select the object that you want to explode.
 - You can multi-select objects for explosion.
2. From the **Create Object** menu, select **Explode**. The selected object is broken up into

components that can be selected and manipulated individually.

- Extrusions are broken up into patches.
- Polylines are broken up into line segments.
- Mesh contours are broken up into polylines.
- Merged point clouds are broken up into individual point clouds.

Export

Window: **ModelSpace, Navigator, Image Viewer**

Menu: **File**

Action: Opens dialog

Usage: The **Export to File** dialog is used to export images, objects, and point clouds to a variety of common graphical, binary, and text file formats, depending on the window from which the export is initiated, and on the content of the export.

Exporting Images

Images can be exported from *Cyclone* to a variety of image formats, including JPEG (jpg) and Windows bitmaps (bmp) files. Images can be exported from the **Navigator** and **Image Viewer** windows.

Exporting Geometric Objects and Point Clouds

Geometric objects and point clouds can be exported to a variety of text and binary file formats, including Leica Geosystems HDS's binary file format, *Cyclone* Object Exchange (COE), which preserves more object information during the import/export process than other file formats. For information on the COE import/export features, see the *COE for AutoCAD* and *COE for MicroStation* chapters. Geometric objects and point clouds can be exported from the ModelSpace and Navigator windows (ModelSpace or ModelSpace View only).

Exporting Ortho Images

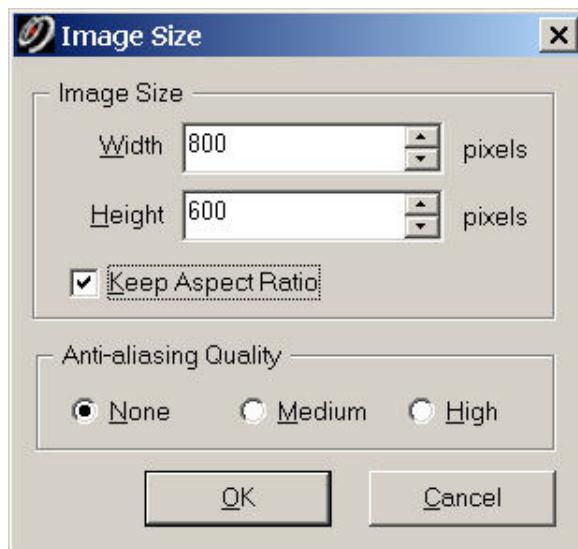
Ortho images can be exported from *Cyclone* as GeoTIFF files accompanied by TWF and TFW parameter files. Ortho images can be exported from the **ModelSpace** window.

To export an ortho image:

1. Using orthographic projection, set the viewpoint in the ModelSpace viewer. This viewpoint will be used for the ortho image.
- The ortho image will be centered in 3D on the viewer's focus point.
2. Adjust the visualization and graphics options of the ModelSpace View. The ortho image will use these settings.
3. In the **File** menu, select **Export....** The **Export to File** dialog appears.
4. In the **Export to File** dialog, set **Save as type** to **Ortho Image – Tagged Image File Format (*.tif)** and enter the filename.
5. Click **Save**. The **Image Size** dialog appears.
6. Enter the ortho image settings and click **OK**. The ortho image and corresponding TWF and TFW files are exported.

- Programs that can read and use the additional positional information in the GeoTIFF specification will be able to automatically position the ortho image in 2D space. Otherwise, refer to the contents of the TWF and TFW parameter files.
- Programs that can read the TWF and TFW parameter files directly will be able to automatically position the ortho image in 2D or 3D space. Otherwise, refer to the contents of the files for the parameters.

Image Size Dialog



A. Width, Height: The dimensions of the ortho image.

B. Keep Aspect Ratio: When checked, changing either the **Width** or the **Height** adjusts the other dimension proportionately.

C. Anti-aliasing Quality: Select the level of anti-aliasing in the ortho image. A higher level helps smoothen jagged edges in the image.

D. OK: Click to accept the settings and export the ortho image.

E. Cancel: Click to cancel this operation.

Export (Limit Box Manager)

Window: Limit Box Manager

Menu: File

Action: Executes command

Usage: The **Export** command exports limit boxes to an LBX file.

- If there are limit boxes selected, only the selected limit boxes are exported.

- If no limit boxes are selected, all limit boxes in the ModelSpace View are exported.

Export Calibration Info

Window: Scan Control, Image Viewer
Menu: Scan Control: Scanner Control | Camera Calibration
Image Viewer: Image | Calibration
Action: Opens dialog
Usage: The **Export Calibration** dialog is used to export camera calibration data to a text file.

- For a general overview of camera calibration, see the [Camera Calibration](#) entry in the *Scanning with the HDS2500* chapter.

To export camera calibration info:

1. The **Scanner Control** menu, point to **Camera Calibration** and select **Export Calibration Info**.... The **Export Calibration** dialog appears.
2. Type in a file name in the **File Name** field.
3. Select a destination folder for the file, and click **Save**.

Export Customizations

Window: Navigator
Menu: Edit
Action: Executes command
Usage: The **Export Customizations** command saves *Cyclone* customizations as an XML file.

- Customizations are modifications you have made to *Cyclone*'s default toolbars and hotkeys.

Export Leica System 1200

Window: ModelSpace
Menu: File
Action: Opens dialog
Usage: The **Export Leica System 1200** command is used to export points, lines, and meshes to Leica's System 1200 format.

- This command can be used to transfer existing conditions (e.g., a decimated TIN of a roadway scan) to a Total Station as a final "design" for use in stake-out.

To export data to Leica's System 1200 format:

1. From the **File** menu, select **Export Leica System 1200**. A folder browser appears.
2. Select the folder that will contain the exported Leica System 1200 files and click **OK**. The **Export Leica System 1200: Data Type** dialog appears.
 - Click **Make New Folder** to create a new folder.
3. Select the type of data that will be exported.
 - If exporting a mesh, the mesh should have TIN properties relative to the +Z axis.
4. Click **Export**. The **Export Options Dialog** appears.
5. Configure the Export Options, then click **Export**. The Leica System 1200 files are written to the selected folder.

Export PCF

Window: ModelSpace

Menu: File

Action: Opens dialog

Usage: The **Export PCF** dialog is used to specify the parameters used in the creation of Alias PCF (Piping Component File) files that can be imported by Alias Piping Solutions software.

- For more information about Alias Piping Solutions, visit: <http://www.alias.ltd.uk>

To export selected objects to PCF:

1. Multi-select the piping objects to be exported to PCF.
Select at least one piping component (e.g., cylinder, flange, valve, etc) in each piping line to be exported. Each piping line is exported to a separate PCF file.
The components in the selected piping line(s) must have proper Spec, LineID, and SKEY values. (See [Piping Mode](#) in the Command Reference.)
2. From the **File** menu, select **Export PCF**. The **Export PCF** dialog appears.
3. Select the folder that will contain the exported PCF files.
4. Select the assumed **Gasket Thickness**. This is used during the export and import to introduce a space between adjacent flanges.
5. Click **OK**. The selected piping lines are exported to PCF.

Export Template

Window: ModelSpace

Menu: Tools | Drawing

Action: Opens dialog

Usage: The **Save Edge Template** dialog is used to export a 2D drawing for use with the **Fit Edge** command.

- See the [Fit Edge](#) entry in the *Modeling* chapter for more information.

To export a cross section template:

1. Using the 2D drawing tools, draw the desired template. Alternately, select an existing (or imported) line drawing and use the **Create Drawing from Object** command to convert the object into a 2D drawing (see the *Create Drawing from Object* entry in this chapter for more information).
2. Before completing the drawing (via the **Create Drawing** command), point to **Tools**, and then point to **Drawing** and select **Export Template....** The **Save Edge Template** dialog appears.
3. Navigate to the desired destination directory, enter a file name in the **File name** box, and click **Save**. The template is saved. See the *Fit Edge* command entry in this chapter for information on using the saved template.

Extend

Window: **ModelSpace**

Menu: **Edit Object**

Action: Displays submenu

Usage: Pointing to **Extend** displays submenu commands used to extend one or more objects using other objects as references.

The following commands are available from the **Extend** submenu:

[Extend to Last Selection](#), [Extend to Ref Plane](#), [Extend All Objects](#)

Extend All Objects

Window: **ModelSpace**

Menu: **Edit Object | Extend**

Action: Executes command

Usage: The **Extend All Objects** command extends the boundary of all selected objects to the planar or linear intersection with the other selected objects.

To extend all objects:

1. Multi-select the objects that you want to extend.
2. From the **Edit Object** menu, point to **Extend**, and then select **Extend All Objects**. The boundaries of the selected objects are extended to the planar or linear intersection with the other selected objects.
 - If they are already overlapping, the boundaries of the selected objects can be retracted to the planar or linear intersection with the other selected objects.

Extend TIN to Polyline

Window: ModelSpace

Menu: Tools | Mesh | Polylines

Tools | Mesh | Breaklines

Action: Executes command

Usage: The **Extend TIN to Polyline** command re-shapes a mesh object to include the vertices contained in a selected polyline.

- The mesh to be extended must qualify to be a TIN.

To extend a TIN to a polyline:

1. Multi-select a polyline and the TIN to be extended.
2. From the **Tools** menu, point to **Mesh**, then to **Polyline**, and then select **Extend TIN to Polyline**. The TIN is redrawn to include the vertices in the polyline, and is moved along the “up” direction to create new edges along the polyline.

To extend a TIN to a breakline:

1. Multi-select a breakline and the TIN to be extended.
 - Mesh editing can be performed only on a *mesh object*, as opposed to a point cloud that is viewed as a mesh. To create a mesh object from a point cloud, see the *Create Mesh* entry in this chapter.
2. From the **Tools** menu, point to **Mesh**, then to **Breakline**, and then select **Extend TIN to Polyline**. The TIN is redrawn to include the vertices in the breakline.

Extend to Last Selection

Window: ModelSpace

Menu: Edit Object | Extend

Action: Executes command

Usage: The **Extend to Last Selection** command extends selected objects to intersect with a reference object, which is the last object in the multi-selection.

- This command is useful for creating piping tees or to extend a wall to a floor or ceiling.

To extend one object to another:

1. Multi-select the objects that you want to extend, ending with the object that you want to serve as the reference object.
2. From the **Extend** submenu, select **Extend to Last Selection**. The selected objects are extended to the planar or linear intersection with the reference object.

Extend to Ref Plane

Window: ModelSpace

Menu: Edit Object | Extend

Action: Executes command

Usage: The **Extend to Ref Plane** command extends selected objects to intersect with the active Reference Plane.

To extend objects to the Reference Plane:

1. Multi-select the objects that you want to extend.
2. From the **Extend** submenu, select **Extend to Ref Plane**. The selected objects are extended to intersect with the Reference Plane.

Extract Cloud from Selection

Window: ModelSpace

Menu: Create Object

Action: Executes command

Usage: The **Extract Cloud from Selection** command creates a point cloud from a mesh object using the mesh vertices as points.

- This command is active only for mesh objects that have been created from a point cloud as opposed to point clouds that are merely viewed as a mesh. See the [Create Mesh](#) command for more information on creating a mesh object.
- This command does not retrieve the original point cloud used to create the selected mesh object; rather, it creates a new point cloud which may differ significantly from the original point cloud used to create the mesh object.
- To retrieve the original cloud used to create a mesh object, use the [Insert Copy of Object's Points](#) command.

Extrude

Window: ModelSpace

Menu: Edit Object

Action: Displays submenu

Usage: Pointing to **Extrude** displays submenu commands used to create an extrusion object from a patch or from a face of an extrusion.

- Extrusions are created with a specified thickness along a user-specified axis or along the patch's normal.

The following commands are available from the **Extrude** submenu:

[Extrude Perpendicular...](#), [Extrude to Last Pick](#), [Extrude Along Axis...](#), [Unextrude](#)

Extrude Along Axis...

Window: ModelSpace

Menu: Edit Object | Extrude

Action: Opens dialog

Usage: The **Extrude Object** dialog is used to create an extrusion object along a specified axis from a selected patch or a picked face on an extruded object.

Extrude Along Axis Dialog Options

AXIS SELECTION

Select the current **X**, **Y**, **Z**, or **Customized** axis along which you want to create an extrusion object. Selecting **Customize** uses the axis defined in the **Axis Value** setting.

AXIS VALUE

Define a custom axis along which you want to create an extrusion object. (Changing a value here selects **Customized** in the **Axis Selection** option.)

MULTIPLIER

Enter a factor by which the **Axis Value** is multiplied to arrive at the actual extrusion.

To create an extrusion object along a specified axis:

1. Select the patch or pick the extrusion object face from which you want to create an extrusion object.
2. From the **Edit Object** menu, point to **Extrude**, and then select **Extrude Along Axis....** The **Extrude Object** dialog appears.
3. Choose the axis and thickness for the extrusion, and then click **OK**. The selected patch or face is extruded.

Extrude Perpendicular...

Window: ModelSpace

Menu: Edit Object | Extrude

Action: Opens dialog

Usage: The **Extrude Perpendicular** command is used to create an extrusion object towards the viewpoint and along an axis perpendicular to a patch or a picked face on an extruded object.

- This command works with multiple selected patches or extrusion object faces.

To create an extrusion object along a perpendicular axis:

1. Select the patch or pick the extrusion object face from which you want to create an extrusion object.
2. From the **Edit Object** menu, point to **Extrude**, and then select **Extrude Perpendicular....** The **Set Extrusion Thickness** dialog appears.
3. Enter a value for the thickness of the extrusion object, and then click **OK**. The extrusion object is created along a perpendicular axis and towards the viewpoint.

Extrude to Last Pick

Window: ModelSpace

Menu: Edit Object | Extrude

Action: Executes command

Usage: The **Extrude to Last Pick** command adds sufficient thickness to a patch or a face of an extruded object to reach the last picked point.

- This command works with multiple selected patches or extruded object faces.

To create an extrusion object to a picked point:

1. Select the patch or extruded object face from which you want to create an extruded object.
2. Pick a point to define the thickness of the extrusion object.
3. From the **Edit Object** menu, point to **Extrude**, and then select **Extrude to Last Pick**. The extrusion object is created by extruding the selected patch or extrusion object face until the “far” face of the extrusion object lies in the same plane as the last picked point.

Fence

Window: ModelSpace

Menu: Edit

Action: Displays submenu

Usage: Pointing to **Fence** displays submenu commands used to clear a fence and to segment and delete portions of objects relative to the fence.

The following commands are available from the **Fence** submenu:

[Clear](#), [Delete Inside](#), [Delete Outside](#), [Delete Selected Inside](#), [Delete Selected Outside](#), [Set From Selection](#).

- For more information on working with fences, see the [Using a Fence](#) entry in the *Modeling* chapter.
-

Fence/View Toggle

Window: Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)
Menu: Edit | Modes
Action: Executes command
Usage: The **Fence/View Toggle** command switches between **Fence** mode and **View** mode.

- The main purpose of this command is to allow for a single hotkey to toggle between the two modes. To assign hotkeys, see the *Customize Hotkeys* entry in this chapter.

Field-of-View

Window: Scan Control
Menu: Window
Action: Expands Tab
Usage: The **Field-of-View** command expands the **Field-of-View** tab in the Scanner Control Panel.

Field-of-View Tab Options

QUICKSCAN

Click to aim your scanner manually using the scanner's **QuickScan** button.

PRESETS

Select a preset rectangular or panoramic scan mode from the dropdown. If target settings do not match the presets, then it displays "Custom".

HZ WINDOW

Displays left and right targeting extents.

V WINDOW

Displays top and bottom targeting extents (relative to the scanner's horizontal plane).

SCANNER HZ

Displays the current horizontal orientation of the scanner's front window .

Field Setup

Window: Scan Control (ScanStation, ScanStation 2, HDS6000)
Menu: Window
Action: Expands Tab
Usage: The **Field Setup** command expands the **Field Setup** tab in the Scanner Control Panel.

- For an overview of the Field Setup process, see the [Field Setup \(ScanStation, ScanStation 2\)](#) section in the *Scanning with the ScanStation, ScanStation 2 and HDS3000* chapter, or the [Field Setup \(HDS6000\)](#) section in the *Scanning with the HDS6000 and HDS4500* chapter.
- The Field Setup procedure requires an enabled dual-axis compensator (ScanStation, ScanStation 2) or enabled tilt sensor (HDS6000). The scanner must be close to level.

Field Setup Tab Options

METHOD

Displays the current method used to determine the position and orientation of the scanner in a known or assumed coordinate system.

RESECTION

Two or more targets with known or assumed coordinates are acquired to calculate the position and orientation of the scanner.

KNOWN BACKSIGHT

A target at a known or assumed coordinate is acquired from a scanner position over a known or assumed coordinate to calculate the actual position of the scanner.

KNOWN AZIMUTH (3D POINT)

The direction to an arbitrary target or a pick point from a scanner position over a known or assumed coordinate is entered to calculate the actual orientation of the scanner.

KNOWN AZIMUTH (AIM)

The azimuth of the direction towards which the user turns the scanner is entered to calculate the actual orientation of the scanner.

OPTIONS

Displays a pop-up menu with additional Field Setup operations.

RECHECK BACKSIGHT...

Re-acquires the current backsight target to see if the scanner has drifted since the backsight was first acquired.

START TRAVERSE

Copies the current Field Setup values (e.g., Station ID, Backsight ID, etc.) to the Traverse panel. A Traverse can then be performed to extend the initial Field Setup.

STATION ID

Displays the ID of the current location of the scanner.

HI

The height of the scanner above the ground.

<, TARGET ID, >

Selects the target settings to display in the panel.

HT

The height of the target above the known point.

AZIMUTH

For **Method: Known Azimuth (3D Point)**, the horizontal angle towards the 3D Point. For **Method: Known Azimuth (Aim)**, the direction in which the scanner is aimed.

X Y Z / N E EL

The position of the known or assumed coordinate with the matching ID in the **Known Coordinates** ScanWorld.

TYPE

The type of the target to be acquired.

ACQUIRE

Acquires the target at the current pick point.

ADD

For **Method: Resection**, adds the current target parameters to the Resection (see the **Resection: Target List** dialog below).

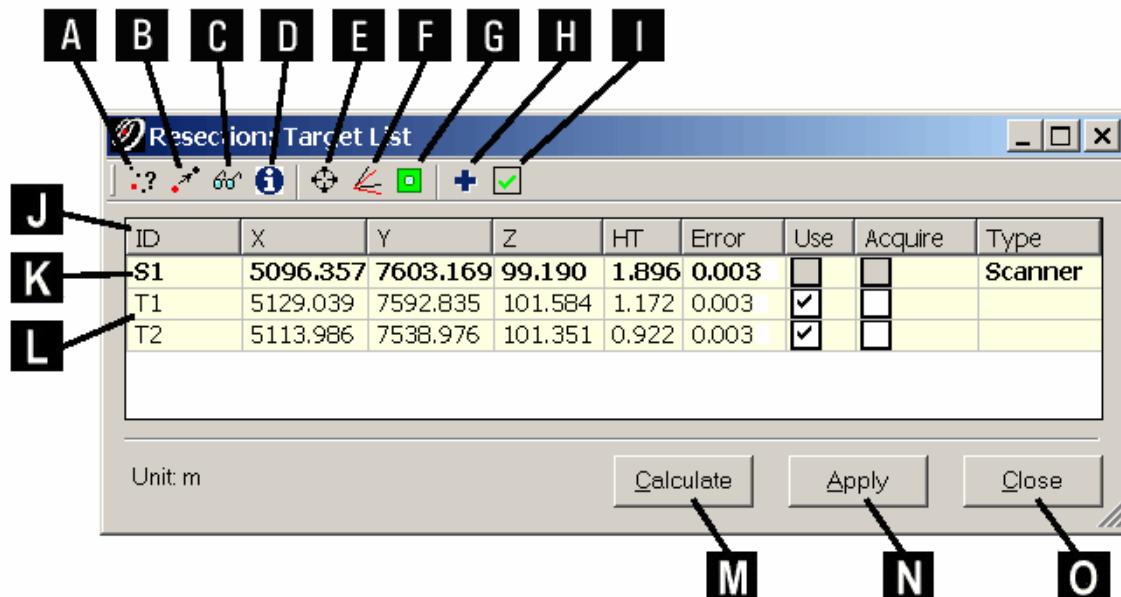
CALCULATE

Calculates the position and orientation of the scanner based on the entered parameters. For **Method: Resection**, the **Resection: Target List** dialog appears (see below). For **Method: Known Azimuth (Aim)**, the user is first prompted to aim the scanner at the real-world azimuth corresponding to the **Azimuth**.

APPLY

For **Method: Known Backsight**, **Known Azimuth (3D Point)**, and **Known Azimuth (Aim)**, sets the ScanWorld Default Coordinate System based on the calculated position and orientation of the scanner.

Resection: Target List Dialog



- A. Calculate the position and orientation of the scanner.
- B. Apply the calculated scanner position to the ScanWorld Default Coordinate System.
- C. Display the **Resection Result ModelSpace** to visualize the Resection.
- D. Invoke the **Resection Report**, which lists the results of the calculated Resection.
- E. Set the viewpoint in the Scan Control on the selected target.
- F. Invoke the **Resection Compare Report**, which lists differences between the acquired target positions relative to the known or assumed coordinates.
- G. Acquire targets with a check-mark in the **Acquire** column. With at least two acquired targets, the remaining targets can be automatically acquired without requiring a pick point.
- H. Create a new candidate target in the list.
- I. Add the new target to the list.
- J. The ID, position, and height of the scanner and targets. **Error** is the difference between the calculated positions and the known or assumed coordinates. Select **Use** to include the target in the calculation. Select **Acquire** to mark the target for acquisition, then push the **Acquire** toolbar button. Select the **Type** of target to be acquired.
- K. The scanner parameters are displayed at the top of the list.

- L. The targets used by the Resection.
- M. Calculate the position and orientation of the scanner.
- N. Apply the calculated scanner position to the ScanWorld Default Coordinate System.
- O. Close the dialog.

To perform a field setup using resection:

1. Set up the scanner. Set up at least two targets over known points.
2. Connect to the ScanStation/ScanStation 2 and enable the dual-axis compensator, or connect to the HDS6000 and enable the tilt sensor.
3. Import or add the known or assumed coordinates, using **Add/Replace Coordinate**, **Import Coordinate List**, or **Import Coordinate List (Leica System 1200)**.
4. From the **Window** menu, select **Field Setup**. The **Field Setup** panel appears in the control panel.
5. Select **Resection** from the **Method** control.
6. Select the **Station ID** from the list, or enter a new ID.
7. Measure and enter the **HI**.
8. Select the **Target ID** from the list of known coordinates.
9. Measure and enter the **HT**.
10. Select the target **Type**.
11. Pick a cloud point on the target over the known point and push **Acquire**. The backsight is acquired. If the target already exists in the ControlSpace, the target need not be re-acquired.
12. Push the **Add** button to add the target to the resection target list.
13. Add at least one other target.
14. Push the **Calculate** button to invoke the **Resection: Target List** dialog.
15. In the **Resection: Target List** dialog, push the **Calculate** button. The position of the scanner is calculated.
16. Optionally, **Create** and **Add** additional targets over known points for a more robust resection. Re-acquire targets with high **Error**, or exclude them from the calculation by turning OFF the **Use** checkbox.
17. Push the **Apply** button to set the ScanWorld Default Coordinate System based on the resection calculation.

To perform a field setup using a known backsight:

1. Set up the scanner over a known point. Set up a target over another known point.
2. Connect to the ScanStation/ScanStation 2 and enable the dual-axis compensator, or connect to the HDS6000 and enable the tilt sensor.
3. Import or add the known or assumed coordinates, using **Add/Replace Coordinate**, **Import Coordinate List**, or **Import Coordinate List (Leica System 1200)**.
4. From the **Window** menu, select **Field Setup**. The **Field Setup** panel appears in the control panel.
5. Select **Known Backsight** from the **Method** control.
6. Select the **Station ID** from the list.

7. Measure and enter the **HI**.
8. Select the **Target ID** from the list of known coordinates.
9. Measure and enter the **HT**.
10. Select the target **Type**.
11. Pick a cloud point on the target over the known point and push **Acquire**. The backsight is acquired. If the target already exists in the ControlSpace, the target need not be re-acquired.
12. Push the **Calculate** button to compute the actual relation of the current setup to the known or assumed coordinates.
13. Push the **Apply** button to set the ScanWorld Default Coordinate System based on the calculation.

To perform a field setup using a known azimuth or bearing to a 3D point:

1. Set up the scanner over a known point. Set up a target.
2. Connect to the ScanStation/ScanStation 2 and enable the dual-axis compensator, or connect to the HDS6000 and enable the tilt sensor.
3. Import or add the known or assumed coordinates, using **Add/Replace Coordinate**, **Import Coordinate List**, or **Import Coordinate List (Leica System 1200)**.
4. From the **Window** menu, select **Field Setup**. The **Field Setup** panel appears in the control panel.
5. Select **Known Azimuth (3D Point)** from the **Method** control.
6. Select the **Station ID** from the list.
7. Measure and enter the **HI**.
8. Select the **Target ID** from the list, or enter a new ID.
9. Measure and enter the **HT**.
10. Measure and enter the **Azimuth** (or bearing) from the scanner to the target.
11. Select the target **Type**.
11. Pick a cloud point on the target over the known point and push **Acquire**. The backsight is acquired. If the target already exists in the ControlSpace, the target need not be re-acquired.
 - Optionally, a pick point may be used, without acquiring a target.
12. Push the **Calculate** button to compute the actual orientation of the scanner towards the target's (or pick point's) position.
13. Push the **Apply** button to set the ScanWorld Default Coordinate System based on the calculation.

To perform a field setup using a known azimuth or bearing of the scanner:

1. Set up the scanner over a known point. Set up a target.
2. Connect to the ScanStation/ScanStation 2 and enable the dual-axis compensator, or connect to the HDS6000 and enable the tilt sensor.
3. Import or add the known or assumed coordinates, using **Add/Replace Coordinate**, **Import Coordinate List**, or **Import Coordinate List (Leica System 1200)**.
4. From the **Window** menu, select **Field Setup**. The **Field Setup** panel appears in the control panel.
5. Select **Known Azimuth (Aim)** from the **Method** control.
6. Select the **Station ID** from the list.
7. Measure and enter the **HI**.

8. Measure and enter the **Azimuth** (or bearing) of the scanner to a direction in the world.
9. Push the **Calculate** button. Turn the scanner in the correct direction and push the QuickScan button on the scanner. The actual orientation of the scanner towards the target's (or pick point's) position is computed.
10. Push the **Apply** button to set the ScanWorld Default Coordinate System based on the calculation.

Fill Selected Hole (Mesh)

Window: ModelSpace

Menu: Tools | Mesh

Action: Executes command

Usage: The **Fill Selected Hole** command replaces gaps in a mesh with triangles.

No new vertices are added. Triangles are created using the existing vertices that form the boundaries of the hole.

- Mesh editing can be performed only on a *mesh object*, as opposed to a point cloud that is viewed as a mesh. To create a mesh object from a point cloud, see the [Create Mesh](#) command.

To fill selected holes:

1. Select a hole in the mesh object to be filled.
▪ To select a hole, click the border of the hole.
2. From the **Tools** menu, point to **Mesh**, and then select **Fill Selected Hole**. The selected hole is filled in with the least number of triangles using existing vertices along the border of the hole.

Fill Selected Hole (Patch)

Window: ModelSpace

Menu: Edit Object | Patch

Action: Executes command

Usage: The **Fill Selected Hole** command fills a hole in a selected patch.

To fill a hole in a selected patch:

1. Pick a point on the patch near the hole you wish to fill. You may also draw a fence around the hole.
2. From the **Edit Object** menu, point to **Patch**, and then select **Fill Selected Hole**. The selected hole is filled.

Find High/Low Point

Window: ModelSpace

Menu: Tools | Measure | Find High/Low Point

Action: Opens dialog

Usage: The **Find High/Low Point** dialog is used to locate the highest or lowest point in a point cloud within a specified area around a picked point. Vertices can be created when high/low points are found.

To find a high/low point:

1. In a point cloud, pick a point near which you want to find a high or low point.
2. From the **Tools** menu, point to **Measure**, and then select **Find High/Low Point**. The **Find High/Low Point** dialog appears. The points within the search range are highlighted, and the highest point along the “up” axis is displayed by default.
3. Continue searching for high/low points. When finished, click **Close**.
 - To search for the low point within the current search range, select **Low Point**.
 - To increase/decrease the search range (an invisible sphere around the pick point), use the **Search Range** slider or enter a value directly.
 - To reset the search using the current high/low point as the center of the search range, click **Pick Point**.
 - To create a vertex at the current high/low point, click **Create Vertex**.

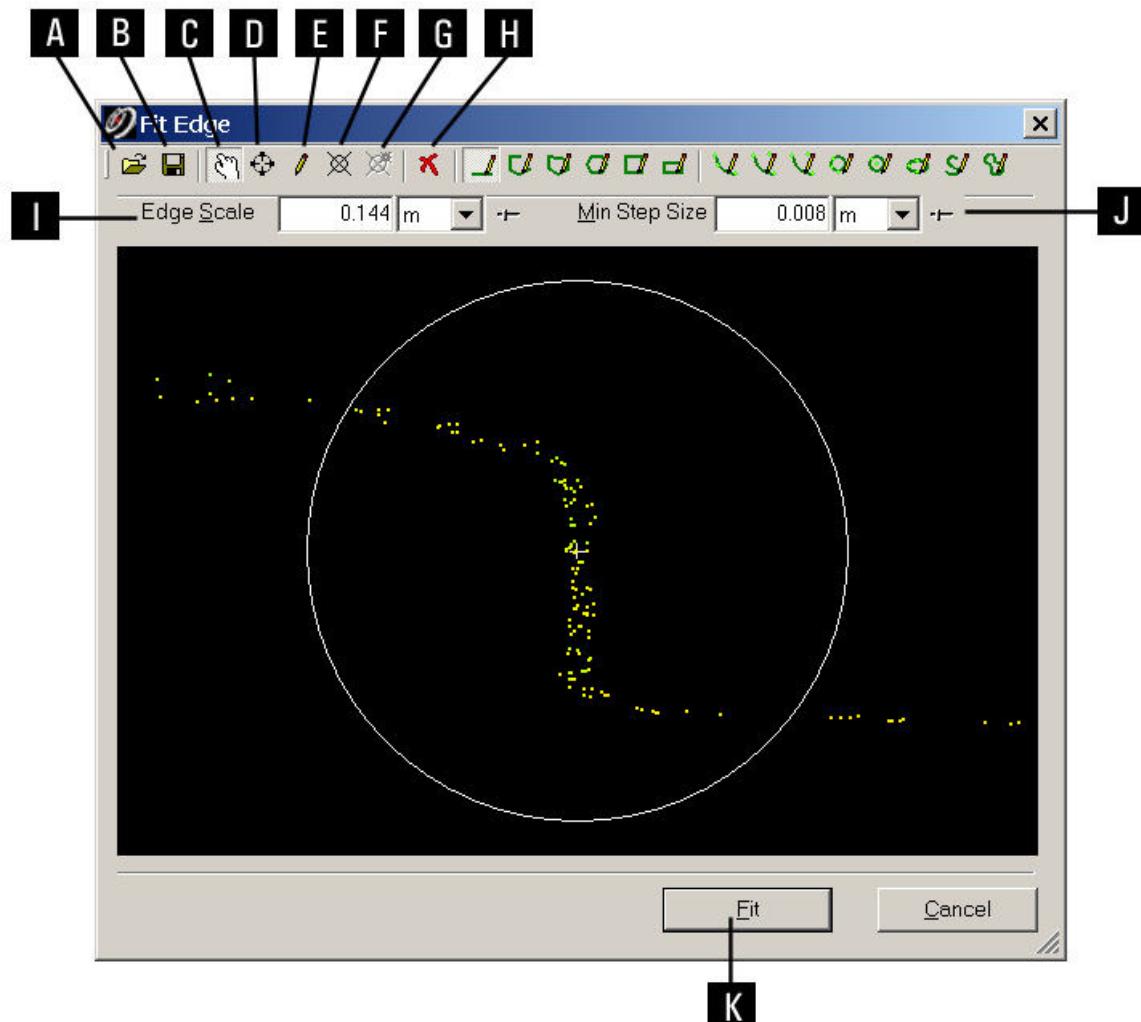
Fit Edge

Window: ModelSpace

Menu: Object

Action: Opens dialog

Usage: The Fit Edge dialog is used to generate polylines describing a swept edge best-fit to a point cloud.

Fit Edge Dialog

- A.** Open a saved template.
- B.** Save the current template to a file.
- C.** Enter View Mode.
- D.** Enter Seek Mode.
- E.** Enter Drawing Mode.
- F.** Enter Nodal Points Mode.
- G.** Add nodal points to each vertex in the current template.
- H.** Clear the current template and all nodal points.

- I. Adjust the radius of the circle drawn to show the scale of the edge.
- J. Adjust the minimum interval between consecutive samples along the edge.
- K. Fit the template.

To fit an edge:

1. Select a point cloud along which you want to fit an edge.
 - For improved performance, first subselect or segment the points around the curb from the rest of the point cloud.
2. From the **Create Object** menu, select **Fit Edge....** The **Fit Edge** dialog appears and displays the selected point cloud.
3. Manipulate the viewpoint so that the cross-section is apparent.
4. Select the **Draw Mode** icon to draw a 2D template using the selected point cloud as a reference.
 - The template is later fit to the surface of the point cloud at intervals and used to determine the location of the polyline vertices. The **Min Step Size** slider determines the minimum interval between consecutive samples. A smaller **Min Step Size** will result in more samples, but may take longer to compute. A larger **Min Step Size** may skip some details, but may be able to jump over gaps in the data.
 - For 2D drawing procedures, see the *Drawing* entry in the *Modeling* chapter. Also, a complete list of the individual drawing commands can be found in the *Drawing* entry in the *Commands* chapter.
 - Panning and rotating the viewpoint does not move the template, to make it easier to align the data with the template.
 - To save a template, click the **Save Template** icon. (Templates can also be created in the ModelSpace using 2D drawing tools and saved via the **Export Template** command.)
 - To load a saved template, click the **Open Template** icon.
5. Click the **Nodal Points Mode** icon and add nodal points to the 2D template.
 - The nodal points indicate the position at which polylines are created and swept across the point cloud in the form of the template.
 - To add a nodal point at each vertex in the template drawing, click the **Add Nodal Points** icon.
 - Hold the Alt key and click on a nodal point to delete it.
 - To clear the current template including nodal points, click the **Clear All** icon.
6. Click **Fit** to fit the template to the selected point cloud. Preview results are displayed in the **ModelSpace Viewer**, and then the **Fit Edge Results** dialog appears.
 - To adjust the maximum distance between the polyline and the nearest point on the original point cloud, use the sliders.
 - If the preview results are not satisfactory, click **Back**. The **Fit Edge** dialog reappears. Adjust the template and nodal points as needed.
7. Click **Create**. The polylines are created.

Fit Fenced

Window: ModelSpace

Menu: Create Object

Action: Displays submenu

Usage: Pointing to **Fit Fenced** displays submenu commands to fit a geometric object to a fenced point cloud.

- The **Fit Fenced** commands function identically to [Fit to Cloud](#) except that only the fenced portion of the point cloud is used to calculate the fit (bypassing the need to first segment the point cloud). Also, it cannot be used to fit a HDS Target or sphere target.
- If any clouds are selected, only the points from selected clouds that lie within the fence are used. If no clouds are selected, then any points that lie within the fence are used.
- Fitting steel sections additionally requires at least one pick point. See the [Steel Section](#) command for details.

Fit to Cloud

Window: ModelSpace

Menu: Create Object

Action: Displays submenu

Usage: Pointing to **Fit to Cloud** displays submenu commands to fit a geometric object to a point cloud.

- To adjust fitting quality tolerances, specify constraints, or fit to parts tables, see the *Object Properties* entry in this chapter.

The types of objects that can be fit are:

Patch, Box, Cylinder, Sphere, Cone, Corner, Line Segment, Elbow, Steel Section (For steel section details, see the [Steel Section](#) command.), **HDS Target, Sphere Target** (For details on HDS Targets or sphere targets, see the [HDS Target](#) and [Sphere Target](#) commands).

To fit an object:

1. From the **Object Preferences** dialog, enable and set the fitting preferences for the object type that you want to fit. See the *Object Preferences* dialog for details. If no preferences are set, objects are simply fit as closely as possible to the selected cloud.
- Tolerances and constraints must be enabled and set to non-zero values in the *Object Preferences* dialog in order to be active for fitting.
2. From the **Create Object** menu, point to **Fit to Cloud Options**, and toggle ON the type of fitting you want to use for the current fit.
- Tolerances, constraints, and parts tables (see *To fit using parts tables* following) can be used

in different combinations depending on the object type. Some objects do not support one or more of the cloud fitting options.

3. Select the point cloud(s) you want to use to fit a geometric shape.
 - For information on preparing point clouds for modeling, see the *Point Cloud Segmentation* entry in the *Modeling* chapter.
 - When multiple point clouds are selected, the **Multiple Point Cloud Style** setting in the **Fitting** tab of the **Edit Preferences** dialog determines the results. If **Fit Single Object** (default) is selected, the clouds are first merged and then a single object is fit to the resulting merged point cloud. If **Fit Multiple Objects** is selected, an object is fit to each cloud individually.
4. From the **Create Object** menu, point to **Fit to Cloud**, and then select the type of object you want to fit. After some computation, the object appears.
 - If the point cloud varies too much from the shape of the object you are trying to create (see the *Fit Tolerances* area of the *Object Preferences* entry in this chapter), a **Fit Quality** warning message appears and gives you an option to keep or cancel the fit. In this case it may help to further refine the point cloud (see the *Point Cloud Segmentation* entry in the *Modeling* chapter) and retry the fit.
 - Each object has a set of editable constraints (see the *Fit Constraints* area of the *Object Preferences* entry in this chapter). When used, these constraints define various attributes of the fit object.
 - If the shape being fit is a corner, the result is three patches with a vertex at their common intersection. When using the **Corner** command, it is not necessary to segment and fit the three planes individually.
 - If the shape being fit is a patch, the **Adjust Patch Boundary** appears. Use the slider to adjust the patch's polygonal boundary. The closer the slider moves towards **Tight Boundary**, the more rigidly the boundary of the new patch adheres to the points on the edge of the point cloud. In some cases this adjustment may necessitate the creation of more than one patch.

To fit an object using parts tables:

1. From the **Object Preferences** dialog, enable and set the constraint and tolerance options for the object type that you want to fit. See the *Object Preferences* dialog for details.
2. From the **Object Preferences** dialog, select the **Catalog/Parts Table** check box, and then make a **Table** selection for each object type that you want to fit.
 - Some objects do not support fitting to parts tables.
3. From the **Object Preferences** dialog, enter a value in the **Match Table Item Within** field. This value determines the range of possible table matches that are checked when the item is fit.
4. From the **Create Object** menu, point to **Fit to Cloud Options**, and toggle **Show Table Matches**, **Use Fit Constraint**, and **Check Fit Tolerances** ON if you are using those options).
 - Tolerances and constraints can be used along with parts tables in different combinations, depending on the object type. Some objects do not support one or more of the cloud fitting options.
5. Select the point cloud(s) you want to use to fit a geometric shape.
 - For information on preparing point clouds for modeling, see the *Cloud Segmentation* entry in the *Modeling* chapter.
 - When multiple point clouds are selected, the **Multiple Point Cloud Style** setting in the **Fitting** tab of the **Edit Preferences** dialog determines the results. If **Fit Single Object** (default) is selected, the clouds are first merged and then a single object is fit to the resulting merged

- point cloud. If **Fit Multiple Objects** is selected, an object is fit to each cloud individually.
6. From the **Create Object** menu, point to **Fit to Cloud**, and then select the type of object you want to fit. The **Select Table Item** dialog appears, displaying possible matches. The best match is temporarily drawn in the ModelSpace and marked with an asterisk in the dialog.
 - Matches within the **Match Table Item Within** value are marked with a "+". The **Match Table Item Within** value is initially set to the corresponding value in the **Object Preferences**.
 - The **vs Table** column displays the RMS (root mean squared) error of the object's dimensions (from the initial fit) compared to the dimension in the table. The **vs Cloud** column displays the standard deviation of point errors relative to the object best-fit using the individual table item's dimensions.
 - Any enabled fitting constraints corresponding to parameters from the table are used in the initial fitting, but only the selected table entry is used when creating the actual object.
 - To update the dialog's values, click **Refresh**. Clicking **Refresh** also calculates any missing **vs Cloud** values.
 - To edit the range of potential matches, enter a new value in the **Match Table Items Within** field, and then click **Refresh**. The new range of potential matches is shown.
 7. Select the appropriate part from the list and click **OK**. The selected parts table item is fit. Note that a dimensional Fit Constraint may be overridden by the selected part.
 - If the point cloud varies too much from the shape of the object you are trying to create (see the *Fit Tolerances* area of the *Object Preferences* entry in this chapter), a **Fit Quality** warning message appears and gives you an option to keep or cancel the fit. In this case it may help to further refine the point cloud (see the *Cloud Segmentation* entry in the *Modeling* chapter) and retry the fit.

Fit to Cloud Options

- Window:** ModelSpace
Menu: Create Object
Action: Displays submenu
Usage: Pointing to **Fit to Cloud Options** displays submenu commands used to toggle various fitting options.

The following objects and dialogs are available from the **Fit to Cloud Options** submenu:

Show Table Matches, **Use Fit Constraints**, **Check Fit Tolerances**

- For details, see the individual command entries.

Flip Selected Edge

Window: ModelSpace

Menu: Tools | Mesh

Action: Executes command

Usage: When the **Flip Selected Edge** command is activated, the bisecting line (selected triangle edge) of a four-sided polygon (with vertices A-B-C-D) is flipped from A-D to B-C, connecting along the opposite diagonal.

- Mesh editing can be performed only on a *mesh object* as opposed to a point cloud that is viewed as a mesh. To create a mesh object from a point cloud, see the [Create Mesh](#) command.

To flip the edge of a triangle:

1. Select the triangle edge to be flipped by clicking on the edge.
- Notice that the selected line is a “bisector” of a four-sided polygon. By identifying the four points of the polygon bisected by the selected line, you can know beforehand what the results will be.
- The edge may be more obvious if viewing the mesh as wireframe.
2. From the **Tools** menu, point to **Mesh**, and then select **Flip Selected Edge**. The selected edge is redrawn.

Fog

Window: ModelSpace

Menu: Tools | Lighting

Action: Toggles menu item ON/OFF

Usage: The **Fog** command toggles the effect of more distant objects appearing dimmer.

- When ON, objects that are farther away appear darker, producing a sense of depth.

Note: Having **Fog** ON while modeling may make distant points difficult to see and lead to modeling errors.

From Curves

Window: ModelSpace

Menu: Create Object

Action: Displays submenu

Usage: Pointing to **From Curves** displays submenu commands used to create polylines and patches from the intersections of selected objects.

The following commands are available from the **From Curves** submenu:

[Polyline](#), [Patch](#)

From Fence Contents...

Window: ModelSpace

Menu: Tools | 2D Extraction

Action: Executes command

Usage: The **From Fence Contents...** command creates a 2D line drawing from the contents of a fence.

To extract 2D lines from the contents of a fence:

1. Make sure that the ModelSpace is in orthographic projection. (See *Orthographic/Perspective* entry in the *Commands* chapter.)
2. Using the rectangular **Fence** tool, fence the area from which you want to extract 2D lines.
3. From the **Tools** menu, point to **2D Extraction**, and then select **From Fence Contents....** The 2D drawing is created and displayed in a new window.
4. Save the new 2D drawing as a CAD file.
 - To save the 2D drawing as a CAD file, select **Save as CAD file...** from the **File** menu, and then use the browser that appears to select a name and location.
 - The unit of measure in the exported file is scaled to match the current unit of measure in *Cyclone*.

From Intersections

Window: ModelSpace

Menu: Create Object

Action: Displays submenu

Usage: Pointing to **From Intersections** displays submenu commands used to create lines and vertices from the intersections of selected objects.

The following commands are available from the **From Intersections** submenu:

[Curve](#), [Vertex](#)

From Pick Points

Window: ModelSpace

Menu: Create Object

Action: Displays submenu

Usage: Pointing to **From Pick Points** displays submenu commands used to connect a series of picked points with a variety of lines.

The following commands are available from the **From Pick Points** submenu:

[Vertex](#), [Line Segment](#), [Polyline](#), [Arc \(3 Points on Arc\)](#), [Arc \(Start, Center, End\)](#), [Cubic Spline](#)

From Selection in Fence...

Window: ModelSpace

Menu: Tools | 2D Extraction

Action: Executes command

Usage: The **From Selection in Fence...** command creates a 2D line drawing from the current selection within the rectangular fence.

To extract 2D lines from a selection:

1. Make sure that the ModelSpace is in orthographic projection. (See the *Orthographic/Perspective* entry in this chapter.)
2. Select the object(s) from which you want to extract 2D lines.
3. Using the rectangular Fence tool, fence the area from which you want to extract 2D lines. The fence defines the boundaries of the 2D drawing.
4. From the **Tools** menu, point to **2D Extraction**, and then select **From Selection in Fence....** The 2D drawing is created and displayed in a new window.
5. Save the new 2D drawing as a CAD file or as a new ModelSpace.
 - To save the 2D drawing as a CAD file, select **Save as CAD file...** from the **File** menu, and then use the browser that appears to select a name and location.
 - The unit of measure in the exported file is scaled to match the current unit of measure in *Cyclone*.

Front

Window: ControlSpace, ModelSpace, Registration, Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)

Menu: Point Cloud Rendering

Action: Toggles menu item ON

Usage: The **Front** command determines the active side of the estimated surfaces in point clouds.

- The command affects the **Shaded** and **One-Sided Point Cloud Rendering** modes.

Get Image

Window: Scan Control

Menu: Scanner Control

Action: Executes command

Usage: The **Get Image** command captures an image of the scene from the point of view of the currently connected scanner and displays it in the **Scan Control** window.

- This image is useful for determining the field of view of the scanner, as well as for specifying the areas of the scene that you want to scan.

- Having a valid image in the **Scan Control** window is not required for targeting the scanner or for scanning.

To capture an image of the scanner's point of view:

- When using the HDS3000, ScanStation, or ScanStation 2 scanner, specify a target region from which to capture an image. For all other scanners, start at step 2.
- From the **Image** menu, select **Get Image** (or click the **Get Image** icon). The image appears in the **Scan Control** window and is saved in the **Images** folder under the parent ScanWorld.
- It may take the scanner several seconds to acquire and transfer the image before it is displayed.

Get Preview

Window: **Scan Control (HDS4500, HDS6000)**

Menu: **Scanner Control**

Action: Executes command

Usage: The **Get Preview** command captures a preview scan of the scene and displays it in the **Scan Control** window.

- This preview scan is useful for determining the field of view of the scanner, as well as for specifying the areas of the scene that you want to scan.

Global Color Map

Window: **ModelSpace**

Menu: **Edit Object | Appearance**

Action: Toggles menu item ON/OFF

Usage: The **Global Color Map** command toggles the use of global color map settings or the current individual color map settings.

- This command affects all point clouds and meshes in the current **ModelSpace**.
- For information on adjusting global color map override settings, see the [Edit Global Color Map](#) command.
- For an overview of color mapping in *Cyclone*, see the [Color Mapping](#) entry in the *Modeling* chapter.

Global Texture Map

Window: **ControlSpace, ModelSpace**

Menu: **View**

Action: Toggles menu item ON

Usage: The **Global Texture Map** command enables the application of texture mapping.

- See the **Edit Object | Appearance | Apply Color Map | Image Texture Map** command.

Grid Settings

Window: Scan Control
Menu: Image
Action: Opens dialog
Usage: The **Grid Settings** dialog is used to set horizontal and vertical spacing for grid lines as well as the font size for grid line markers.

Group (Command)

Window: ModelSpace
Menu: Edit | Group
Action: Executes command
Usage: The **Group** command creates a link between two or more objects so they can be manipulated simultaneously.

- *Cyclone* can group objects of any type. Selecting one object in the group selects the entire group.

Group (Menu)

Window: ModelSpace
Menu: Edit
Action: Displays submenu
Usage: Pointing to **Group** displays submenu commands used to add or remove items to and from a group.

The following commands are available from the **Group** submenu:

[Group](#), [Ungroup](#), [Remove from Group](#)

Handles

Window: ModelSpace
Menu: Edit Object
Action: Displays submenu
Usage: Pointing to **Handles** displays submenu commands used to set display and constraint options for handles.

The following commands, dialogs, and submenus are available from the **Handles** submenu:

[Show Handles](#), [Handles Always Visible](#), [Show Rotation Handles](#), [Edit Snap to Object Threshold...](#), [Constrain Motion to](#)

- For **Handle** submenu command details, see their individual entries.
- For more information on using handles, see the [Handles](#) entry in the *Modeling* chapter.

Handles Always Visible

Window: ModelSpace

Menu: Edit Object | Handles

Action: Toggles menu item ON/OFF

Usage: The **Handles Always Visible** command toggles the constant visibility of handles even when hidden from view by an object.

- When this command is checked, it allows convenient access to handles that would otherwise be hidden from view (behind or inside of objects).
- When this command is unchecked, handles that are inside or behind an object are not visible and cannot be selected for manipulation.
- This command has no effect if [Show Handles](#) is OFF.
- For more information on using handles, see the [Handles](#) entry in the *Modeling* chapter.

Hide Point Clouds

Window: ModelSpace

Menu: View

Action: Executes command

Usage: The **Hide Point Clouds** command toggles visibility of point clouds. This is equivalent to toggling the visibility of point clouds in the **Selectable/Visible** tab of the **View Properties** dialog.

Home Position

Window: Scan Control (HDS6000)

Menu: Scanner

Action: Executes command

Usage: The **Home Position** command turns the scanner back to its 0-azimuth position.

HDS Target

Window: ModelSpace

Menu: Create Object | Fit to Cloud

Action: Opens dialog

Usage: The **HDS Target** command fits a vertex at the center of an HDS target. When a HDS Target is created, it is added automatically to the ControlSpace as well.

To fit an HDS target

1. Segment the point cloud to which you want to fit an HDS tie-point from the surrounding cloud points (if any).
2. Pick near the bright center of the point cloud subset to which you want to fit a HDS Target.
3. From the **Create Object** menu, point to **Fit to Cloud**, and then select **Tie-Point**. The **HDS Target** dialog appears.
4. Enter an identifier and a comment in the appropriate fields, and then click **OK**. A vertex is fit to the HDS Target.

Hide Selected Clouds

Window: ModelSpace

Menu: View

Action: Executes command

Usage: The **Hide Selected Clouds** command hides all selected cloud points.

- Points hidden using this command can be recovered with the [Show Hidden Clouds](#) command.

Hide Unselected Clouds

Window: ModelSpace

Menu: View

Action: Executes command

Usage: The **Hide Unselected Clouds** command hides all cloud points in the ModelSpace except those that have been selected.

- Points hidden using this command can be recovered with the [Show Hidden Clouds](#) command.

Image Info...

Window: ModelSpace

Menu: Tools | Info

Action: Opens dialog

Usage: The **Object Info** dialog is used to view available information on the current image.

Image Resolution

Window: Scan Control

Menu: Image

Action: Displays submenu

Usage: Pointing to **Image Resolution** displays submenu commands to select **High** (1024x1024), **Medium** (512x512), or **Low** (256x256) resolution for image acquisition.

- This command does not apply to the HDS2500.

Import...

Window: ModelSpace, Navigator

Menu: File

Action: Opens dialog

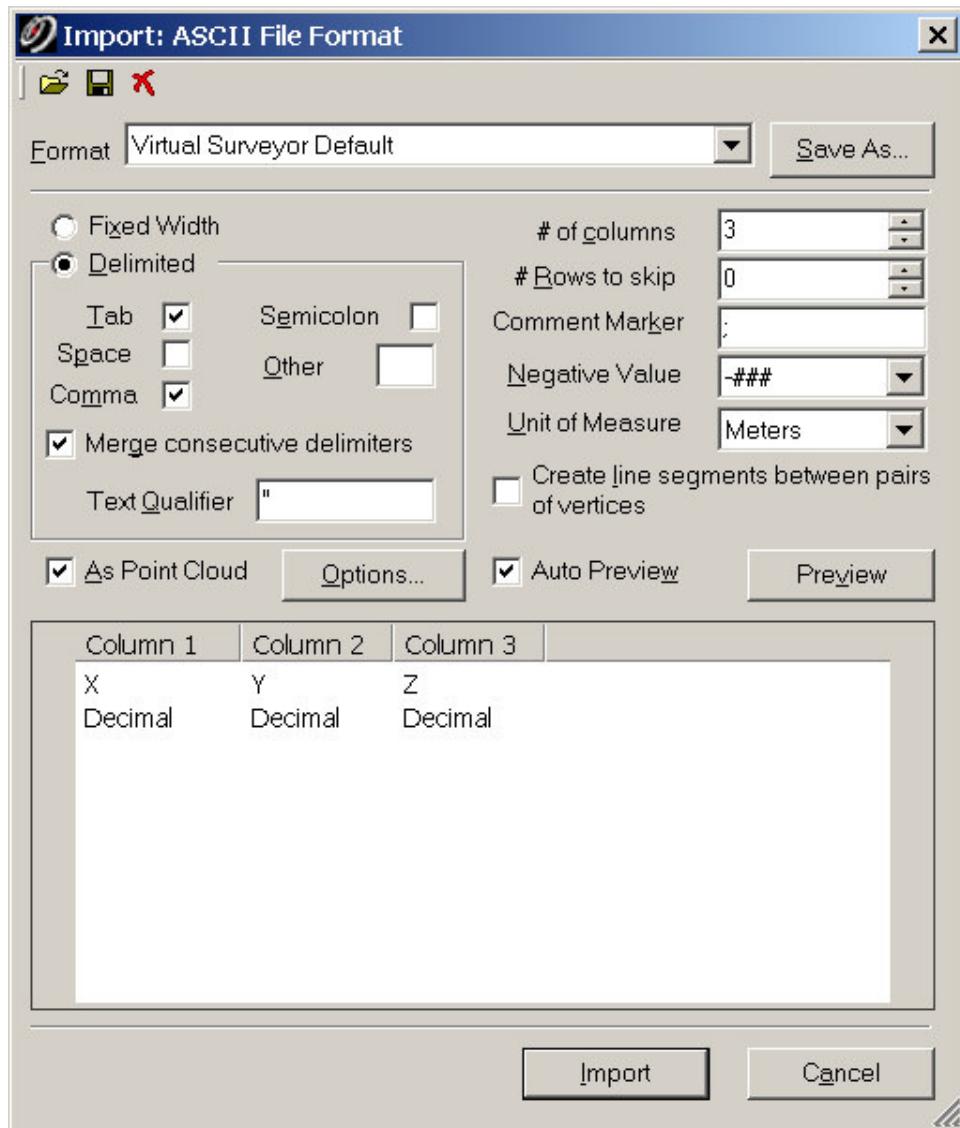
Usage: The **Import From File** dialog is used to import images, objects, and point data into *Cyclone* from a variety of common image and text file formats.

▪▪▪For instructions on importing COE files into *Cyclone*, see *COE for AutoCAD* or *COE for MicroStation* of this manual.

Note: Importing a file into a project via the **Navigator** window generally reconstructs the most objects (e.g., ScanWorld with Registration, Scans, ModelSpace with ModelSpace View). Importing a file into a ModelSpace recovers only the geometry from that file.

- Some ASCII files have pre-established formats and are imported directly into *Cyclone*; the data contained in the files is represented by point clouds or vertices in the resulting ModelSpace. Other text files do not import directly. Instead, an **Import: ASCII File Format** dialog is displayed, prompting you to define/map the format of the file so that *Cyclone* can read (or “parse”) it correctly and extract the relevant information. You can either select a pre-defined file format from the **Format** pull-down list, modify an existing file format, or define a new format.

Import: ASCII File Format Dialog



Click the **Import** icon to imports an ASCII rules file.



Click the **Export** icon to export a rules format to an external ASCII rules file (*.afr).



Click the **Delete** icon to remove a previously saved rule from the **Format** list.

Format

Displays previously created ASCII rules formats.

Click **Save As** to save the current settings as a format. The newly saved format will be displayed in the **Format** drop-down list box.

Fixed Width

Select this check box to specify that the columns in the file to be imported have a fixed width; clear this check box if delimiters are used to represent a column break.

Delimited

Select **Delimited**, and then complete the available options as appropriate. You can select one or more delimiters. Type a **Text**

	Qualifier (default is quotation mark) which parses as one item anything enclosed in it.
As Point Cloud	Select As Point Cloud to import the coordinates in the file as points in a point cloud. The reserved point cloud-specific columns "Red", "Green", "Blue", "Gray", "Intensity", "NormalX", "NormalY", and "NormalZ" become available.
Options	Click the Options button to customize the ranges used for color and intensity when importing as a point cloud.
# of columns	Type how many columns of data are in the file to be imported.
# Rows to skip	Type the number of rows to ignore beginning with row 1 (such as header rows).
Comment Marker	Type a comment marker. When the marker entered into this field is encountered, everything else after the indicator on that row is ignored (not imported). The default is a semi-colon.
Negative Value	Select the format used to specify negative values in the import file.
Unit of Measure	Select the unit of measure used in the import file.
Create line segments between pairs of vertices	Select this check box to tell <i>Cyclone</i> to draw lines between pairs of vertices, which some users may use to aid in the registration process.
Auto Preview	Select this check box to automatically update the contents of the preview window when settings are changed. Click Preview to update the contents of the preview window.
Preview window	Displays the contents of the file to be imported and the impact of the different rules imposed upon it.

To import an ASCII file as vertices:

1. If you are in the **Navigator** window, select the Database, Project, ScanWorld, or ModelSpace into which you want to import the file.
2. From the **File** menu, select **Import** (for both the **Navigator** and **ModelSpace** windows).
3. In the **Files of type** field, select **All ASCII files** (or a specific text file format), then navigate to the location of the desired text file, and select the file to be imported. If the text file has a TXT, SVY, or XYZ extension, the **Import: ASCII File Format** dialog is displayed.
4. If you have pre-defined ASCII file formats available, select the desired format from the **Format** pull-down list and click **Import**. Otherwise, follow the steps outlined below.
 - Use the **Import Rules to a File** icon to import *afr files if available.
5. Select either **Fixed Width** or **Delimited** display as appropriate.
 - If you select **Delimited**, you are offered a variety of delimiter options. Select the delimiter(s) used in your file. A delimiter separates one column of data from the next.
 - If you select **Fixed Width**, an extra row of numbers appears beneath the column headers (e.g., 1-10 11-16 17-24), which represents the extents of each column. Click a pair of numbers to set the left-most border of that column (note that the first column is adjusted by selecting the second column and editing the beginning number).
 - The **Number of columns** field is populated automatically, based on your display choices in steps 3 and 4. You can change this value as necessary.
6. If you selected **Delimited** display, enter a value in the **Text Qualifier** field (default is a quotation mark). Anything enclosed within the defined text qualifiers will be parsed as one

item. For example, you may have commas set as delimiters, but in certain cases, you may want commas to be imported; enclosing the comma in quotation marks will qualify it as “text” and when encountered, it is imported as text rather than as a qualifier.

7. Deselect the **As Point Cloud** checkbox. The column names reserved for use by point cloud import (e.g., “Intensity”) become regular column headings.
8. Indicate any initial rows (e.g., headers) to be skipped during the import by entering a number in the **# Rows to skip** field.
9. If you have text in your file to be treated as commentary (hence, to be ignored upon import), enter the qualifier in the **Treat as Comment** field. As soon as the comment qualifier is encountered on a line in the file, the rest of the line is ignored.
10. Select the unit of measure represented by the numbers in your data table using the **Unit of Measure** field.
11. Check the **Create line segments between pairs of vertices** option to create line segments between consecutive pairs of vertices.

Note: The bottom half of the screen allows you to see the results of your settings/selections immediately. Columns can be resized as needed, and the dialog box can be widened as necessary to view additional columns.

12. When you are satisfied with the settings, click the **Save as** button next to the **Format** field and enter a name for this ASCII file format. This makes the format available for future use.
 - To save this format to an external file for use on another machine, click the **Export Format to a File** icon.
13. To complete the import process, click the **Import** button. Each coordinate is imported as a vertex into the created ModelSpace. The newly created vertices are also placed automatically into the ScanWorld's ControlSpace (if importing into an existing or import-created ScanWorld).

To import an ASCII file as a point cloud:

See *To import an ASCII file as vertices* for a description of the process. Additionally, select the **As Point Cloud** checkbox to enable the import of the coordinates in the ASCII file as a single point cloud, instead of as individual vertices.

1. Configure the column headings based on the contents of the ASCII file.
 - If the file specifies a red-green-blue color for each point, assign the “Red”, “Green”, and “Blue” headings to the corresponding columns.
 - If the file specifies a grayscale value for each point, assign the “Gray” heading to the corresponding column.
 - If the file specifies an intensity for each point, assign the “Intensity” heading to the corresponding column.
 - If the file specifies a normal vector for each point, assign the “NormalX”, “NormalY”, and “NormalZ” headings to the corresponding columns.
2. If needed, click the **Options** button to invoke the **Range Options** dialog. Configure the minimum and maximum valid values for the Color and Intensity (if needed). Values beyond the valid ranges are clamped to the nearest minimum or maximum value. Typical values for Color are 0-255. Typical values for Intensity can vary by scanner manufacturer.

To import an image (graphic) file into *Cyclone*:

1. Select the Database or Project into which you want to import the image file, and then select **Import...** from the **File** menu.
2. In the **Files of type** field, select **All Image Formats** (or select a specific graphical file format), then navigate to the location of the file to be imported, and select the file to be imported.
3. Click **Open**. The file is imported into *Cyclone*.

Import (Limit Box Manager)

Window: Limit Box Manager

Menu: File

Action: Executes command

Usage: The **Import** command imports saved limit boxes from an LBX file.

- The imported limit boxes are appended to the end of the current list.
- Limit box names and parameters may be duplicated.

Import Calibration Info

Window: Scan Control, Image Viewer

Menu: Scan Control: Scanner Control | Camera Calibration

Image Viewer: Image | Calibration

Action: Opens dialog

Usage: The **Import Calibration** dialog is used to import camera calibration data from a text file.

- For a general overview of camera calibration, see the [Camera Calibration](#) entry in the *Scanning with the HDS2500* chapter.

To import camera calibration info:

1. The **Scanner Control** menu, point to **Camera Calibration** and select **Import Calibration Info....** The **Import Calibration** dialog appears.
2. Browse to and select the calibration file.
3. Click **Open**. The calibration information is imported and overwrites any existing camera calibration information.

Import Coordinate List...

Window: Scan Control (ScanStation, ScanStation 2, HDS6000)

Menu: Project

Action: Opens dialog

Usage: The Import Coordinate List dialog is used to select an ASCII file to import as known or assumed coordinates that will be used in the Field Setup or Traverse procedures.

- The known or assumed coordinates are saved in the Known Coordinates ScanWorld under the current Project.
- For more information on Field Setup and Traverse, see the [Field Setup \(ScanStation, ScanStation 2\)](#) and [Traverse \(ScanStation, ScanStation 2\)](#) entries in the *Scanning with the ScanStation, ScanStation 2, and HDS3000* chapter, or the *Field Setup (HDS6000) Traverse (HDS6000)* section in the *Scanning with the HDS6000 and HDS4500* chapter.

To import known or assumed coordinates:

1. From the Project menu, select Import Coordinate List. The Import Coordinate List dialog appears.
2. Select the text file and click Open.
 - If importing a custom file format, the Import: ASCII File Format dialog appears. Configure the correct format and click Import.
 - Each coordinate from the file is added as a vertex with an ID and that position to the Known Coordinates ScanWorld.

Import Coordinate List (Leica System 1200)...

Window: Scan Control (ScanStation, ScanStation 2, HDS6000)

Menu: Project

Action: Opens dialog

Usage: The Browse For Folder dialog is used to select a folder with a Leica System 1200 DB-X database to import as known or assumed coordinates that will be used in the Field Setup or Traverse procedures.

- The known or assumed coordinates are saved in the Known Coordinates ScanWorld under the current Project.
- For more information on Field Setup and Traverse, see the [Field Setup \(ScanStation, ScanStation 2\)](#) and [Traverse \(ScanStation, ScanStation 2\)](#) entries in the *Scanning with the ScanStation, ScanStation 2, and HDS3000* chapter, or the *Field Setup (HDS6000) and Traverse (HDS6000)* section in the *Scanning with the HDS6000 and HDS4500* chapter.

To import known or assumed coordinates from Leica System 1200 DB-X:

1. From the Project menu, select Import Coordinate List (Leica System 1200). The Browse For Folder dialog appears.
2. Select the folder containing a Leica System 1200 DB-X database and click OK.
 - Each coordinate in the Leica System 1200 DB-X database is added as a vertex with an ID and that position to the Known Coordinates ScanWorld.

Import Customizations

Window: Navigator

Menu: Edit

Action: Executes command

Usage: The **Import Customizations** command imports *Cyclone* customizations that have been saved as an XML file.

- Customizations are modifications you have made to *Cyclone*'s default toolbars and hotkeys.

Import Leica System 1200

Window: Navigator

Menu: File

Action: Opens dialog

Usage: The **Browse For Folder** dialog opened by this command is used to select a folder containing the Leica System 1200 DB-X database to be imported into a *Cyclone* Project or Database.

- Each Leica System 1200 job in the DB-X database will be imported as a ModelSpace. Points in the job will be imported as individual vertices.

Import Registration...

Window: Registration

Menu: Registration

Action: Opens dialog

Usage: The **Import Registration** dialog is used to import one or more Registrations into the current Registration for the purpose of constructing a single inclusive Registration.

- The import process brings in all the ScanWorlds and constraints of the imported Registrations without duplication.

To import a registration:

- From the **Registration** menu, select **Import Registration....** The **Select Registration(s) to Import** dialog appears. The left side of the dialog contains a Navigator pane.
- In the Navigator pane, select the existing Registration that you want to add to the new Registration-in-progress, and then click the **Add** icon. The selected Registration appears in the right-hand pane.
 - All Registrations must be in the same database.
- Continue adding until all of the Registrations that you want to add appear in the right-hand window, and then click **OK**. Each unique ScanWorld from all of the added Registrations is added once, and each unique constraint is copied to the current Registration.

Increase Point Width

Window: ControlSpace, ModelSpace, Registration, Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)
Menu: Point Cloud Rendering
Action: Toggles menu item ON
Usage: The **Increase Point Width** command increments by one pixel the thickness of each cloud point displayed in the ModelSpace View.

Info (Limit Box Manager)

Window: Limit Box Manager
Menu: Limit Box
Action: Executes command
Usage: The **Info** command displays the parameters of the selected limit box.

Info (ModelSpace viewer)

Window: ModelSpace
Menu: Tools
Action: Displays submenu
Usage: Pointing to **Info** displays submenu commands used to view information for selected objects and windows.

The following commands and dialogs are available from the **Info** submenu:

[Object Info](#), [ModelSpace Info](#), [ScanWorld Info](#)

Info (Scan Control viewer)

Window: Scan Control
Menu: Scanner
Action: Opens dialog
Usage: The **Info** dialog displays the current scanner's name and address.

Insert

Window: ModelSpace
Menu: Create Object
Action: Displays submenu
Usage: Pointing to **Insert** displays submenu commands used to insert a variety of objects.

The following objects and dialogs are available from the **Insert** submenu:

Vertex, Line Segment, Patch, Box, various **Steel Sections, Cone, Cylinder, Sphere, Torus, Elbow, Reducing Elbow, Welding Neck Flange, Blind Flange, Concentric Reducer, Eccentric Reducer, Valve, Tee, Camera, Info Marker**

- To edit default insertion parameters or select a default parts table, see the [Object Preferences](#) command.

Object Insertion Rules

- **The first object inserted** into an empty scene is inserted at (0, 0, 0).
- If there is no selected object and no pick point when an object is inserted, the size and alignment of the new object are defined in the **Creation/Initial Values** area of the [Object Preferences](#) dialog. The object is located in front of the current view at the focal point.
- When an existing object is picked, the picked object provides a plane, direction, and/or vertex for the inserted object to base its initial position, orientation, and dimensions. For example, a patch readily defines a plane, a direction (plane normal), and a point (origin). The rules can be sophisticated (e.g., when inserting an object on a cylinder, picking near the endcap inserts the object aligned with the endcap, but picking away from the endcap inserts the object perpendicular to the cylinder's surface at the picked point). Exception: cameras are always inserted at the current viewpoint regardless of picks.
- When inserting a series of objects, the previous object provides a plane, direction, and/or vertex for the inserted object to base its initial position, orientation, and dimensions. Using this method, it is possible to build a chain of objects, such as a piping run, from scratch.
- The inserted object is initially selected with a pick point based on the previous selection. If there was no previous selection, a default pick point is provided.

To insert an object:

1. Pick an existing object if you want to use it as a reference, or position the ModelSpace viewpoint's focal point where you want the object to appear.
- To insert parts from a table, enable and specify the part in the **Object Preferences** dialog (see the *Object Preferences* entry in this chapter), and then toggle the **Use Table Parts** command ON from the **Create Object** menu. For more information on parts tables, see the *Using Parts Tables* section in the *Modeling* chapter.
2. From the **Create Object** menu, point to **Insert**, and then select the type of object you want to insert. The **Insertion Properties** dialog appears, and a transparent preview object appears in the ModelSpace View.
3. Make desired changes to the default size, position, color, or other properties of the selected object. The preview object is updated to reflect the current settings.
- If parts tables are enabled for the type of object being inserted, select the desired selections from the **Table** and **Item** lists.

- To move or rotate the object from its current position, use the **Move** or **Rotate** commands. See the *Move/Rotate* command entry in this chapter for detailed instructions.
 - To cancel the insertion and close the dialog, click **Cancel**.
4. Click **Create**. The object is inserted.

Insert Copy of Object's Points

Window: ModelSpace
Menu: Create Object
Action: Executes command
Usage: The **Insert Copy of Object's Points** command inserts a copy of the points used in fitting the selected object.

- This command creates new points that are selectable and editable and have no relation to the object.
- If you only need to view an object's points, use the [Show Object's Cloud](#) command.

Insert Point Light...

Window: ModelSpace
Menu: Tools | Lighting
Action: Executes command
Usage: The **Insert Point Light...** command inserts a point light in your current ModelSpace View.

- A point light source emits light in all directions from a 3D point. By default, the light does not dim with distance.
- The point light is inserted at the current pick point. If there is no pick point, the point light is inserted at the current viewpoint.
- To customize point light settings, see the [Edit Point/Spot Lights](#) command.
- For more information about lighting, see the [Lighting](#) entry in the *Modeling* chapter.

Insert Spot Light...

Window: ModelSpace
Menu: Tools | Lighting
Action: Executes command
Usage: The **Insert Spot Light...** command inserts a spot light in the center of your current ModelSpace View.

- A spot light emits light within a cone from a 3D point. The centerline of the cone determines the direction of the spot light source, and the tip determines the origin. The angle of the cone determines the spread of the spot light.
- If there is a pick point on an object, the object provides the location, and in most cases, the orientation for your spot light. If there is no pick point, the spot light is inserted at the current viewpoint.
- By default, the light does not dim with distance; the fall-off at the edge of the cone is immediate.
- To customize spot light settings, see the [Edit Point/Spot Lights](#) command.
- For more information about lighting, see the [Lighting](#) entry in the *Modeling* chapter.

Interfering Points...

Window: ModelSpace

Menu: Tools | Measure

Action: Opens dialog

Usage: The **Interfering Points** dialog is used to highlight cloud points that are within a user-specified distance of a selected object; those points may be segmented from the rest of the point cloud.

To segment interfering points:

1. From the **Tools** menu, point to **Measure**, and then select **Interfering Points**. The **Interfering Points** dialog appears.
2. Select the object and any nearby point clouds for which you want to indicate interferences.
3. Enter a value in the **Tolerance Value** box, or use the **Interference Tolerance** slider to interactively adjust the distance from the selected object within which points are highlighted. The points within the specified tolerance are highlighted in white.
 - If the selected object defines a volume, the points contained inside the object are also highlighted.
 - The total number of points within the specified tolerance is displayed in the **Interference Tolerance** field.
4. Click **Segment**. The highlighted points are segmented from the rest of the point cloud.
5. Click **Close**. The **Interfering Points** dialog is closed.

Invert

Window: ModelSpace

Menu: Selection | Point Cloud Sub-Selection

Action: Executes command

Usage: The **Invert** command inverts the point cloud selection set. All selected points become unselected; all unselected points become selected.

Invert Selection

Window: ModelSpace

Menu: Selection

Action: Executes command

Usage: The **Invert Selection** command toggles the selection status of every loaded object in the ModelSpace.

- This command is particularly useful when you want to select all but a few objects, because it allows you to first select the objects that you ultimately do *not* want selected, and then invert the selection.

Isometric

Window: ModelSpace

Menu: Viewpoint | Standard Views

Action: Executes command

Usage: The **Isometric** command moves the current viewpoint to the standard isometric viewing position vector (-1, -1, 1) at a distance where the entire scene is visible.

- The **Right Isometric** command moves the current viewpoint to the standard isometric viewing position vector (1, -1, 1).

Keep Collapsed

Window: ModelSpace

Menu: Tools | Exclusion Volume

Action: Toggles setting ON/OFF

Usage: The **Keep Collapsed** command toggles the setting to keep the sides of the selected exclusion volume collapsed to the minimum dimensions needed to enclose its contents.

Keep Viewpoint Upright

Window: ModelSpace, Registration

Menu: ModelSpace: Viewpoint

Registration: Viewers

Action: Toggles menu item ON/OFF

Usage: The **Keep Viewpoint Upright** command toggles the setting to keep the viewpoint in the ModelSpace window upright.

- Toggling this option OFF allows you to rotate the viewpoint in any direction.

- *Cyclone* lets you pan, rotate, and zoom the viewpoint in 3D. However, some users prefer not to rotate the view with all three degrees of freedom (roll, yaw, and pitch) turned ON, because it is not always clear which way is "up". **Keep Viewpoint Upright** disables the roll component of the viewer. That is, you can rotate left and right, and up and down, but you can't view a scene upside-down.
- A scene's "up" and "down" are defined by the current coordinate system and the [Set Up Direction](#) submenu.
- To reset the "up" direction, see the [Set Up Direction](#) command.

Launch

Window: ModelSpace

Menu: File

Action: Displays submenu

Usage: Pointing to **Launch** displays submenu commands used to: open a second viewer for the current ModelSpace, copy selected objects to a new ModelSpace, and export and view the ModelSpace in a supported 3rd-party application.

The following commands are available from the **Launch** submenu:

[Open New Viewer](#), [Copy Selection to New ModelSpace](#), [Copy Fenced to New ModelSpace](#),
[Update Original ModelSpace](#), [AutoCAD...](#), [MicroStation...](#)

Layers...

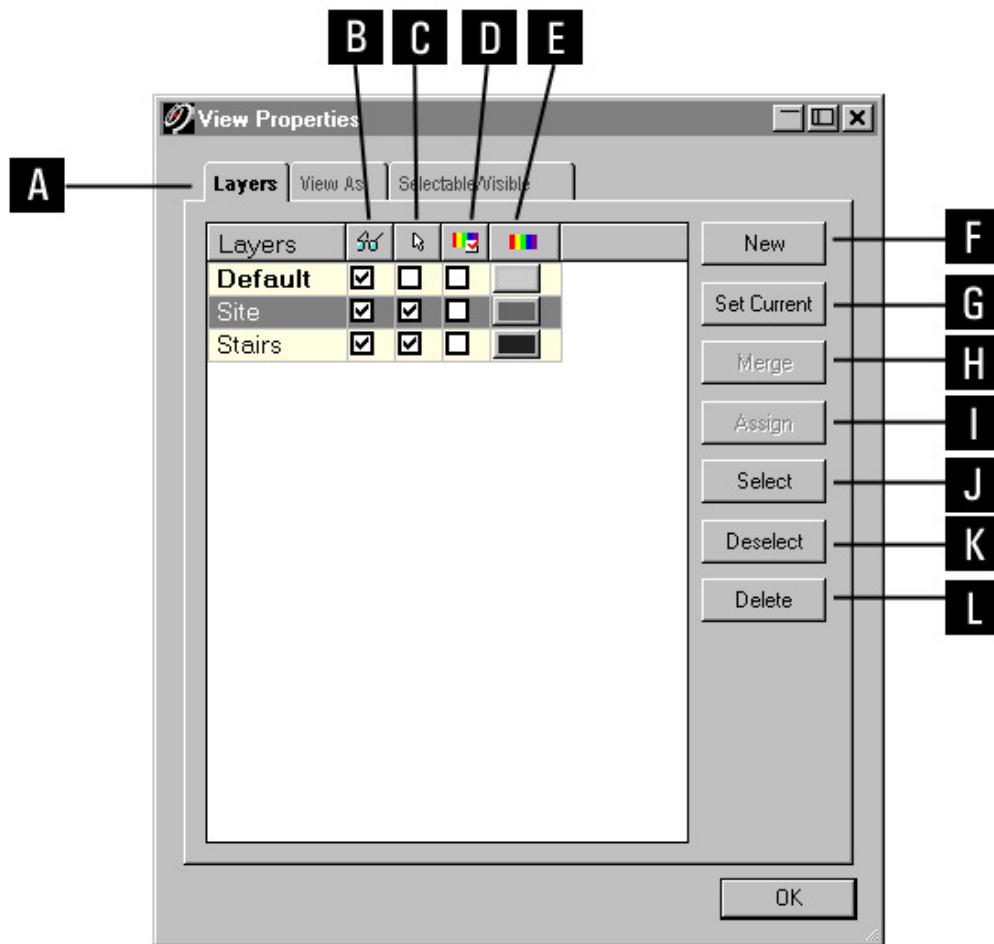
Window: ModelSpace

Menu: View

Action: Opens dialog

Usage: The **Layers...** command opens the **Layers** tab of the **View Properties** dialog, which is used to organize objects into layers.

- Layers are user-defined logical groupings of objects that can be used to simplify tasks in a crowded ModelSpace.
- For an overview of layers, see the [Layers](#) entry in the *Modeling* chapter.

View Properties Dialog – Layers Tab

- A. Click to access the other View Properties tabs.
- B. Check to make the layer's objects visible.
- C. Check to make the layer's objects selectable.
- D. Check to use the layer's color for all the layer's objects.
- E. Click to use the color picker to choose a color for the layer.
- F. Create a new layer.
- G. Designate the selected layer as the current layer. Any objects created in the ModelSpace are added to the current layer.

H. Create a single new layer from the selected layers. (If one of the layers is the current layer, the new merged layer becomes the current layer.)

I. Assign the selected objects to the selected layer.

J. Select all objects in the selected layer.

K. Deselect all objects in the selected layer.

L. Delete the selected layer.

License Manager

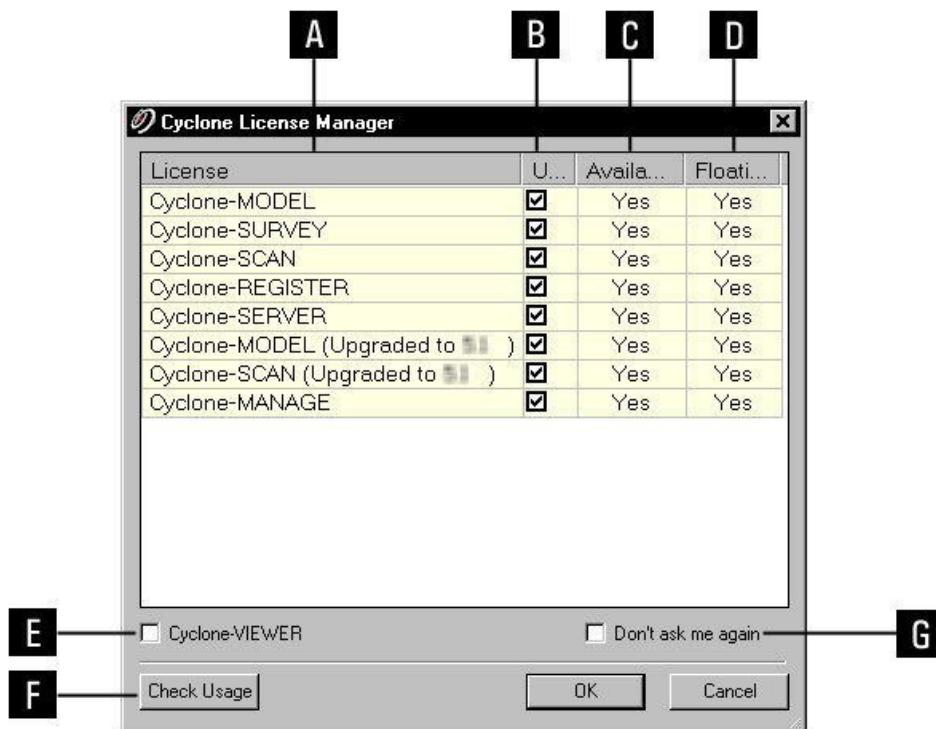
Window: Navigator

Menu: Help

Action: Opens dialog

Usage: This **License Manager** dialog is used to configure floating licenses for the current client.

Cyclone License Manager Dialog



A. List of known licenses.

- B.** *Cyclone* will attempt to acquire the selected licenses when launched, regardless of the Use selections or licenses available.
- C.** Displays the availability of each license. Yes means that there is currently a (valid) license available. No means that the license is either invalid or in use by some other client.
- D.** Licenses marked as Yes are floating licenses obtained from the floating license server. No means that the license was acquired as a node-locked (non-floating) license. N/A means that the license could not be acquired, so it is unknown if the license is floating or not.
- E.** Select to open the next *Cyclone* session with the *Cyclone* Viewer only (acquires no licenses),
- F.** Displays a list of current users for the selected licenses.
- G.** When selected, the License Manager does not automatically appear when a new floating license is available.

Lighting

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Lighting** displays submenu commands used to insert, edit, show, or hide lights.

The following commands and dialogs are available from the **Lighting** submenu:

[Edit Point/Spot Light](#), [Edit Environmental Lights](#), [Insert Point Light](#), [Insert Spot Light](#), [Fog](#)

- For more information about lighting, see the [Lighting](#) entry in the *Modeling* chapter.

Limit Box

Window: ModelSpace

Menu: View

Action: Executes command

Usage: The **Limit Box** command creates a boundary beyond which nothing is loaded into the ModelSpace View.

Note: A limit box can be helpful when you want to work on a subset of objects in ModelSpace. It can also improve performance because *Cyclone* does not need to load or draw objects outside of the limit box.

Tip: To resize the limit box, click an edge of the displayed limit box to display resizing handles. Use the blue handles to move the limit box. Use the orange handles to resize the limit box.

Note: Limit box parameters may also be edited from the [Edit Properties](#) dialog.

Limit Box On/Off (Limit Box Manager)

Window: Limit Box Manager

Menu: View

Action: Executes command

Usage: The **Limit Box On/Off** command toggles the use of the limit box in the ModelSpace View on or off.

Limit Box on Viewer 1

Window: Registration

Menu: Viewers

Action: Executes command

Usage: The **Limit Box on Viewer 1** command toggles the use of the limit box in Constraint Viewer 1 on or off.

Limit Box on Viewer 2

Window: Registration

Menu: Viewers

Action: Executes command

Usage: The **Limit Box on Viewer 2** command toggles the use of the limit box in Constraint Viewer 2 on or off.

Line Segment

Window: ModelSpace

Menu: Create Object | From Pick Points

Action: Executes command

Usage: The **Line Segment** command creates line segments that connect a series of picked points.

To create a line segment

1. Pick two points.
2. From the **Create Object** menu, point to **From Pick Points**, and then select **Line Segment**.
The line segment is created.

Load Image

Window: Scan Control

Menu: Image

Action: Executes command

Usage: The **Load Images** command is used to load previously captured camera images into the current **Scan Control** window.

- This command does not apply to the HDS2500.

Load Points within Fence

Window: Scan Control (ScanStation 2, HDS4500, HDS6000)

Menu: Scanner Control

Action: Executes command

Usage: The **Load Points within Fence** command is used to load only those points in the current ScanWorld's partially-imported scans that fall within the fence.

- The command can be used to selectively view a subset of the entire scan without having to load the entire scan, which can take a significant amount of time. For example, use this command to load only the region of the partially-imported scan that includes a black/white target.

To load points within the fence:

1. Perform a scan that is only partially imported (see the **ScanStation 2: Max # Points Auto-Imported** and **HDS4500/HDS6000: Max # Points Auto-Imported** preferences in the **Scan** tab of the **Edit Preferences** dialog).
2. Set the fence to cover the area to be loaded. A larger region may take more time to load.
3. From the **Scanner Control** menu, select **Load Points within Fence**. The points in the fenced region are loaded.
 - A new scan is created from each partially-imported scan that overlaps the fence.
 - The points within the fence are fully loaded without subsampling.

Load Points within Fenced Selection

Window: Scan Control (ScanStation 2, HDS4500, HDS6000)

Menu: Scanner Control

Action: Executes command

Usage: The **Load Points within Fenced Selection** command is used to load only those points in the current ScanWorld's partially-imported scans that fall within the fence and that are also selected in the Scan Control.

- The command can be used to selectively view a subset of the entire scan without having to load the entire scan or parts of other scans, which can take a significant amount of time. For

example, use this command to load only the region of the partially-imported scan that includes a black/white target.

To load points from selected clouds within the fence:

1. Perform a scan that is only partially imported (see the **ScanStation 2: Max # Points Auto-Imported** and **HDS4500/HDS6000: Max # Points Auto-Imported** preferences in the **Scan** tab of the **Edit Preferences** dialog).
2. Set the fence to cover the area to be loaded. A larger region may take more time to load.
3. Select the point cloud(s) to be considered for loading.
4. From the **Scanner Control** menu, select **Load Points within Fenced Selection**. The points in the fenced region are loaded from the selected scans.
 - A new scan is created from each selected, partially-imported scan that overlaps the fence.
 - The points within the fence are fully loaded without subsampling.

Lower Active Cutplane

Window: ModelSpace

Menu: Tools | Cutplane

Action: Executes command

Usage: The **Lower Active Cutplane** command lowers the Cutplane by the amount specified in the [Cutplanes](#) dialog offset parameter (see the [Add/Edit Cutplanes](#) command).

Make Available Offline

Window: Navigator, ModelSpace

Menu: Navigator: Tools

ModelSpace: File

Action: Executes command

Usage: The **Make Available Offline** command finds any data referenced by the selected object and copies that data into the selected object's database, ensuring that all data in the selected object can be accessed even if the original database is unavailable.

- This procedure is relevant only for point clouds whose data reside on another database (e.g., a ModelSpace created in database "A" that has been reference-copied to database "B").
- If you attempt to open a ModelSpace when the original database (or the source server) is inaccessible and you have not run this command, you will receive an error message if the point cloud data cannot be accessed. Use of this command gives you the option of creating a back-up database on a local drive so that work can continue without needing to access the original database.

Note: Executing this command may increase the size of the selected object's database.

Make Circular...

Window: ModelSpace

Menu: Edit Object | Patch

Action: Opens dialog

Usage: The **Make Circular** command is used to make the selected patch circular while attempting to preserve the general dimensions of the original patch.

- The program uses a circumscribing circle (all of the patch's vertices are contained within the smallest possible circle).

To make a patch circular:

- Select the patch you want to make circular.
 - From the **Edit Object** menu, select **Make Circular...**. The **Number of Sides** dialog appears.
 - In the **Number of Sides** field, enter the number of sides you want to use to create a circular patch, and then click **OK**. The patch is transformed.
- This command can be used to create various geometric shapes with equilateral sides. Entering 3 creates a triangle, entering 4 creates a square, entering 5 creates a pentagon, etc..

Make Rectangular

Window: ModelSpace

Menu: Edit Object | Patch

Action: Executes command

Usage: The **Make Rectangular** command makes the selected patch rectangular, while attempting to preserve the general dimensions of the original patch.

- All of the original patch's vertices are contained within the smallest possible rectangle.

To make a patch rectangular:

- Select the patch you want to make rectangular.
- From the **Edit Object** menu, select **Make Rectangular**. The patch is made rectangular.

Manage...

Window: Scan Control | Script

Menu: Scanner Control

Action: Opens dialog

Usage: The **Manage** command launches the **Scanner Scripts** window, which is used to view, add, and delete scanner scripts.

Measure

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Measure** displays submenu commands used to save, edit, and take various measurements.

The following commands, submenus, and dialogs are available from the **Measure** submenu:

[Edit Measurements...](#), [Save Measurements](#), [Clear Temporary Measurement](#), [Distance](#), [Datum](#), [Elevation](#), [Angle](#), [Back Angle](#), [Centerline Length](#), [Piping Takeoff](#), [Surface Area](#), [Volume](#), [Mesh Volume submenu](#), [Interfering Points](#), [Mesh to Points Deviations](#), [Contours to Mesh Deviations](#), [Find High/Low Point](#)

Merge

Window: ModelSpace

Menu: Create Object

Action: Executes command

Usage: The **Merge** command combines two or more selected objects into one object.

- If the objects were created from point clouds, the merged object is refit to the original point clouds.
- Objects of different types cannot be merged.
- This command is useful for merging point clouds, especially after many segmenting and cutting operations.
- This command can be used to merge contour lines as well as objects, provided that the contours have compatible properties (e.g., same major and minor elevations).
- Merging fills in space between the merged objects, in the process of creating the new object. Use this command to create objects such as wide floors and ceilings or long pipes. This feature is particularly useful when filling areas not scanned directly.

To merge objects:

1. Multi-select the objects that you want to merge.
2. From the **Create Object** menu, select **Merge**. The objects are merged, and the new object appears.

To merge contour lines:

1. Multi-select contour lines with similar properties that you want to merge.
Note: You cannot merge contour lines within the same point cloud.
2. Select **Merge** from the **Create Object** menu. The contour lines are merged into one entity.
Note: Use the **Edit Properties** dialog (**Edit Object | Edit Properties...**) to modify contour

line properties, as well as contour label properties.

Merge from ModelSpace...

Window: ModelSpace

Menu: File

Action: Opens dialog

Usage: The **Select ModelSpace to Merge** dialog combines two ModelSpaces.

This command may be used to incorporate changes made in another ModelSpace.

To merge ModelSpaces:

1. Open the target ModelSpace (the ModelSpace into which you want to merge).
2. From the **File** menu, select **Merge from ModelSpace....** The **Select ModelSpace to Merge** dialog appears.
3. Navigate to the object ModelSpace (the ModelSpace that want to merge into the target ModelSpace), and click **Open**. The ModelSpaces are merged.
 - Each object in the object ModelSpace now appears in the current target ModelSpace (even if some of the objects are identical to objects that already exist in the target ModelSpace).
 - Objects that were duplicated via the **Launch | Copy Selection to New ModelSpace** command are overwritten by the copy of the object in the ModelSpace from which you are merging.
 - The default data coordinate systems are used; any user coordinate systems are ignored when merging the ModelSpaces. To take user coordinate systems into account, use the **Copy/Paste ModelSpace Viewer** commands instead.

Merge Sub-Scans

Window: Navigator

Menu: Tools

Action: Executes command

Usage: The **Merge Sub-Scans** command combines all sub-scans within the selected ScanWorld. This command applies to HDS3000, ScanStation, and ScanStation 2 scans.

- To adjust the limit above which new sub-scans are created, use the **HDS3000/ScanStation/ScanStation 2: Max # Points/Sub-Scan** preference in the **Scan** tab of the **Edit Preferences** dialog.

Note: When working on a project that includes multiple large scans, it may be more efficient to merge sub-scans while subsequent scans are being captured.

Mesh

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to Mesh displays submenu commands, submenus, and dialogs used to create and manipulate mesh objects.

The following commands and submenus are available from the Mesh submenu:

[Create Mesh...](#), [Decimate Mesh](#), [Delete Selection](#), [Fill Selected Hole](#), [Add Face](#), [Move Vertex](#), [Flip Selected Edge](#), [Polylines](#), [Breaklines](#), [Optimize Mesh](#), [Verify TIN](#).

Mesh to Points Deviations...

Window: ModelSpace

Menu: Tools | Measure

Action: Opens dialog

Usage: The **Mesh to Points Deviations...** command compares the faces in the selected mesh object with the points in the selected point cloud (or originating point cloud if no point cloud is selected), and uses the data to calculate deviations in accordance with user defined parameters.

To calculate mesh to point cloud deviation:

1. Select the mesh object whose deviation from the original point cloud you wish to measure.
 - It is recommended that you view the mesh in “Per-Face Normals” mode before executing the **Mesh to Points Deviations...** command. See the *View Objects As* entry in this chapter for more information.
2. Select the point cloud for which to take the deviation measurements. If no point cloud is selected, the point cloud originally used to create the mesh is used.
3. From the **Tools** menu, point to **Measure**, then select **Mesh to Points Deviations....**
Initialization commences, which culminates with the display of the **Deviation Analysis** dialog.
4. Use the slider bar to set the threshold on which to measure deviations. As the slider is moved, the graphical display is adjusted continually to highlight only those triangles with deviations from the point cloud that fall above the threshold.

- Note:** The **Minimum** and **Maximum** fields are read only, and indicate the range for which deviations can be measured.
5. Select whether to highlight faces based on **Maximum** or **Average** deviation, and whether to use **Absolute** values (no positive or negative distances) or **Signed** distances.

- Note:** Deviation analysis results are displayed in the **Details** field.
6. When finished, click **OK**. The **Deviation Analysis** dialog is closed.

MicroStation...

Window: ModelSpace

Menu: File | Launch

Action: Opens dialog

Usage: The **MicroStation...** command launches the current ModelSpace View in MicroStationE2 using the *Cyclone Object Exchange* (COE) intermediary functionality.

- When this command is initiated, objects in the current ModelSpace View are first transferred to a COE file, which is then loaded into MicroStation. See [COE for MicroStation](#) for information on working with COE files.
- Before executing this command, it is important to verify that MicroStation and Leica Geosystems HDS's COE Data Transfer for MicroStation are properly installed on your workstation.

To launch MicroStation from a ModelSpace View:

1. From the **File** menu, point to **Launch**, and then select **MicroStation**.
- You may be prompted to locate the MicroStation executable (**USTATION.EXE**). If this occurs, navigate to the folder containing the file and select it.
2. If you want to open an existing file, navigate to the appropriate folder and select the desired file; if you are opening a new drawing, enter a new file name.
3. Click **Open**. MicroStation is launched.
- You can open a temporary file by clicking **CANCEL**; a file will be created in your system's **TEMP** directory. To save any work done in a temporary file, be sure to save your drawing from MicroStation.

Minimize

Window: ModelSpace

Menu: Tools | Exclusion Volume

Action: Executes command

Usage: The **Minimize** command orients and shrinks the sides of the selected exclusion volume to the minimum dimensions required to enclose its contents.

To minimize the volume of an exclusion volume:

1. Select the exclusion volume.
2. From the **Tools** menu, point to **Exclusion Volume**, and then select **Minimize**. The exclusion volume is oriented and collapsed to the minimum volume required to contain its original contents.

Minimize (Limit Box Manager)

Window: Limit Box Manager

Menu: View

Action: Executes command

Usage: The **Minimize** command resizes the limit box to the minimum volume required to contain the currently selected objects (if any) or to contain all objects (if there is no selection). The orientation of the limit box is adjusted as required to produce the minimum possible volume.

Minimize Limit Box

Window: ModelSpace

Menu: View

Action: Executes command

Usage: The **Minimize Limit Box** command resizes the limit box to the minimum volume required to contain the currently selected objects (if any) or to contain all objects (if there is no selection). The orientation of the limit box is adjusted as required to produce the minimum possible volume.

- To obtain the minimum possible limit box size without changing limit box orientation, use the [Collapse Limit Box](#) command

Miter Connectors...

Window: ModelSpace

Menu: Tools | Piping

Action: Displays dialog

Usage: The **Miter Connectors...** command displays the **Number of Miter Segments** dialog, which is used to determine the number of segments to be used to connect two selected pipes with a mitered elbow connector.

To create a mitered elbow connection:

1. Pick a point on the first pipe near the end you want connected.
2. Multi-select the second pipe near the end that you want connected.
3. From the **Tools** menu, point to **Piping**, and then select **Miter Connectors....** The **Number of Miter Segments** dialog appears.
4. Enter the number of sweep segments you want to represent the elbow and click **OK**. The mitered elbow appears.

ModelSpace

Window: **Navigator**

Menu: **Create**

Action: Executes command

Usage: The **ModelSpace** command creates a new ModelSpace in the selected location.

- A ModelSpace can be created in a database, project directory, or ScanWorld.
- The contents of the ModelSpace are determined by the settings in the **Initialization** tab within the [Edit Preferences](#) dialog. See the [Preferences](#) command.

To create a new ModelSpace:

1. In the **Navigator** window, select the database, ScanWorld, or Project into which you want to add a new ModelSpace.
2. From the **Create** menu, select **ModelSpace**. The ModelSpace is created in the selected container.

ModelSpace Info

Window: **ModelSpace**

Menu: **Tools | Info**

Action: Opens dialog

Usage: The **ModelSpace Info** dialog is used to view available information on the current ModelSpace.

- The type of information varies with the contents of the ModelSpace.
- When the **Automatically Update** box is checked, the ModelSpace information changes when the ModelSpace changes. If the box is unchecked, the contents of the **ModelSpace Info** dialog do not change.

ModelSpace View

Window: **Navigator**

Menu: **Create**

Action: Executes command

Usage: The **ModelSpace View** command creates a new ModelSpace View for the selected ModelSpace. This command is active only when an existing ModelSpace is selected in the **Navigator** window.

- Views created using this command are stored in the ModelSpace folder from which they were created.

- Multiple views can be created from the same ModelSpace using this command.
- ModelSpace View names can be changed; right click on the ModelSpace View in the Navigator window, select **Rename**, then type in the new name.

ModelSpaceViews

A ModelSpace View is a collection of settings applied to a ModelSpace, including the initial viewing position and settings for graphical viewing parameters that override the object's default parameters. Multiple ModelSpace Views can be saved and restored for a given ModelSpace to provide ready access to particular aspects of a ModelSpace, or to provide a custom modeling environment for a modeler in a large, multi-person project.

All changes made to objects within the ModelSpace are immediately reflected in all of the ModelSpace's Views. Multiple users may share a single view; one user's changes to the shared View are visible to the other users of that View.

- To save ModelSpace Views, use the **Save View As** dialog.

Note: The View must be saved at least once before changes to the View are made persistent and before the View is available to other users.

- To restore saved ModelSpace Views, use the **Switch to Views** dialog or open them from the **Navigator** window.

To open a saved ModelSpace View:

1. From the **View** menu, select **ModelSpace Views....** The **Switch to View** dialog appears.
2. From the list of saved ModelSpace Views, select the ModelSpace View you want to restore.
3. Click **OK**. The selected ModelSpace View replaces the current view.

Window: ModelSpace

Menu: Edit Object

Action: Opens dialog

Usage: The **Move/Rotate** dialog is used to rotate selected objects or move selected objects a specified distance along a specified axis.

Mouse Modifiers

Window: ControlSpace, ModelSpace, Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)

Menu: Edit

Action: Displays submenu

Usage: Pointing to **Mouse Modifiers** displays submenu commands used to toggle the effects of certain keyboard and mouse buttons.

- The following modifiers are available from the **Mouse Modifiers** submenu:

Shift Key, Ctrl Key, Alt Key, Right Button, Middle Button

- While toggled ON, the corresponding key or mouse button is considered to be pushed when the primary (left) mouse button is clicked.
- Mouse Modifiers have no effect on keyboard hotkeys.

Move/Rotate...**Window:** ModelSpace**Menu:** Edit Object**Action:** Opens dialog**Usage:** The **Move/Rotate** dialog is used to rotate selected objects or move selected objects a specified distance along a specified axis.**To rotate selected objects:**

1. Select the object(s) you want to move.
2. From the **Edit Object** menu, select **Move/Rotate....** The **Move/Rotate** dialog appears.
3. Select the **Rotate** tab. A graphical indicator appears in the ModelSpace viewer to illustrate the rotation angle and axis at the current settings.
 - If multiple objects are selected, each object receives a unique identifier (such as *Cylinder 1*, *Cylinder 2*, etc.) to aid in differentiating objects and points on those objects.
 - Significant points (i.e., the end points of a cylinder) as well as current and subsequent pick points are numbered and displayed on the selected objects. These numbers can be used as references to move objects.
 - If multiple objects are selected, you may specify a subset of the selected objects to be affected by the move from the **Apply to** list.
4. Click the arrow icon to the right of the **Axis of Rotation** row. The **Axis of Rotation** subdialog appears.
5. Specify the axis around which you want to rotate using the available axes, points or picks, and then click **OK**. The **Axis of Rotation** subdialog closes.
6. Click the arrow icon to the right of the **Center of Rotation** row. The **Center of Rotation** subdialog appears.
7. Specify the center point around which you want to rotate using the available points or picks, and then click **OK**. The **Center of Rotation** subdialog closes.
8. From the **Rotate** tab, click the arrow icon to the right of the **Angle of Rotation** row. The **Angle of Rotation** subdialog appears.
9. Specify the angle you want to rotate using the available axes, points or custom values, and then click **OK**. The **Angle of Rotation** subdialog closes.
10. Click **Rotate**. The object is rotated according to the specified axis, center, and angle.
 - To rotate the object from its new position according to the current settings, click **Rotate** again.
11. When you are finished rotating the object, click **Close** to exit the dialog.

To move selected objects:

1. Select the object(s) you want to move.
2. From the **Edit Object** menu, select **Move/Rotate**. The **Move/Rotate** dialog appears, and a graphical indicator appears in the ModelSpace viewer to illustrate the move axis and offset of the current settings.
 - If multiple objects are selected, each object receives a unique identifier (such as *Cylinder 1*, *Cylinder 2*, etc.) to aid in differentiating objects and points on those objects.
 - Significant points (i.e., the end points of a cylinder) as well as current and subsequent pick points are numbered and displayed on the selected objects. These numbers can be used as references to move objects.
 - If multiple objects are selected, you may specify a subset of the selected objects to be affected by the move from the **Apply to** list.
3. Click the arrow icon to the right of the **Direction of Move** row. The **Direction of Move** subdialog appears.
4. Specify the direction for the move using axes, points or picks, and then click **OK**. The **Direction of Move** subdialog closes.
5. Click the arrow icon to the right of the **Distance of Move** row. The **Distance** subdialog appears.
6. Specify the direction for the move using points, picks, or custom distance, and then click **OK**. The **Distance** subdialog closes.
7. Click **Move**. The object is moved the specified distance along the specified axis.
8. When you are finished moving the object, click **Close** to exit the dialog.

Move Vertex

Window: ModelSpace

Menu: Tools | Mesh

Action: Executes command

Usage: The **Move Vertex** command translates the selected vertex from its initial position to a new position within a mesh.

- Mesh editing can be performed only on a *mesh object*, as opposed to a point cloud that is viewed as a mesh. To create a mesh object from a point cloud, see the [Create Mesh](#) command.

To move a vertex:

1. To select the vertex to be moved, pick a point on the mesh near the vertex you wish to move.
2. Multi-select the point to which you want to move the vertex.
 - If the destination point is another vertex, select a point on the mesh near the target vertex.

To move the vertex to a point on another object, pick a point on that object.

3. From the **Tools** menu, point to **Mesh**, and then select **Move Vertex**. The vertex is translated to either the target vertex or the picked point on another object.

Move Viewpoint

Window: ModelSpace

Menu: Viewpoint

Action: Displays submenu

Usage: Pointing to **Move Viewpoint** displays submenu commands used to move your viewpoint 90 degrees to the left, right, up, or down while maintaining your point of focus.

- To rotate your point of focus, see the [Turn Viewpoint](#) command.

The following commands are available from the **Move Viewpoint** submenu:

Left, Right, Up, Down

Multi-Pick Mode

Window: ModelSpace

Menu: Edit | Modes

Action: Executes command

Usage: The **Multi-Pick Mode** command enters **Multi-Pick** mode and selects the **Multi-Pick** mode cursor.

- **Multi-Pick** mode is used to select multiple objects without holding **SHIFT**.

Multi-Pick/View Toggle

Window: ModelSpace

Menu: Edit | Modes

Action: Executes command

Usage: The **Multi-Pick/View Toggle** command switches between **Multi-Pick** mode and **View** mode.

- The main purpose of this command is to allow for a single hotkey to toggle between the two modes. To assign hotkeys, see the *Customize Hotkeys* entry in this chapter.

New (Limit Box Manager)

Window: Limit Box Manager

Menu: Limit Box

Action: Executes command

Usage: The **New** command creates a new limit box and automatically sizes it to the objects selected in the ModelSpace. The new limit box is aligned to the current UCS axes.

- If no objects are selected, the new limit box is auto-sized to all objects in the ModelSpace.
- If more than one pick point is present, the limit box is initialized to the minimum bounding box of the pick points. This is useful for quickly isolating cloud data with a few picks.

New Scanner Position

Window: Scan Control

Menu: Project

Action: Executes command

Usage: The **New Scanner Position** command resets the Scan Control window settings and prepares a new ScanWorld for the next scan.

To reset for a new scanner position:

1. After repositioning the scanner (or if you simply want to scan into a new ScanWorld), select **New Scanner Position** from the **Project** menu.
- The **New Scanner Position** command is also available as the **New Position** button in the control panel on the right-hand side of the **Scan Control** window.
2. The following occur automatically:
 - A new ScanWorld is prepared within the same project to contain the subsequent scans and images acquired from the scanner.
 - The current image in the **Scan Control** window is cleared.
 - If an attached ModelSpace viewer (opened from the **Scan Control** window to view incoming scan data) is open, it is detached and a new one is opened; the previous ModelSpace viewer remains open.

Object Coordinate System

Window: ModelSpace

Menu: Edit Object | Handles | Constrain Motion to

Action: Executes command

Usage: The **Object Coordinate System** command constrains the movement of objects relative to the specified axis of the coordinate system defined by the object itself.

- For more information on using user coordinate systems, see the [Save/Edit Coordinate Systems](#) command.
- Some handles may not be able to move according to the constraints (e.g., rotation and other custom handles).
- For more information on using handles, see the [Handles](#) entry in the *Modeling* chapter.

Object Info...

Window: ModelSpace

Menu: Tools | Info

Action: Opens dialog

Usage: The **Object Info** dialog is used to view available information on the selected object.

- The type of information displayed depends on the type of object selected.

To view object information:

1. Select the object(s) about which you want information.
2. From the **Tools** menu, point to **Info** and then select **Object Info**. The **Object Info** dialog appears.
 - When more than one object is selected, information is displayed for each object in its own **Object Info** dialog.
 - When the **Automatically Update** box is checked, the object information changes when the object changes. If the box is unchecked, the contents of the **Object Info** dialog for that object do not change.
3. Click **Close**.

Object Info... (Navigator Window)

Window: Navigator

Menu: View

Action: Opens dialog

Usage: The **Object Info** command displays relevant information for the selected object (ModelSpace, ScanWorld, Registration, Database, etc.).

- The information displayed varies, depending on the type of object selected.

Object Preferences...

Window: Navigator, ModelSpace

Menu: Edit

Action: Opens dialog

Usage: The **Object Preferences** dialog is used to set catalog/parts table, fit tolerances, fit constraints, picking and object creation/initial value preferences.

- For an overview of fitting and inserting objects using tolerances, constraints, and tables, see the [Modeling](#) chapter.)

Object Preferences Dialog Options

General Settings

OBJECT TYPE

Select the type of object for which you want to adjust default properties. Each type of object has a unique set of editable properties, which appear when the object is selected in the **Object Type** field.

LEVEL

Select **User Default**, **Session**, or **View**. The **User Default** preferences are restored the next time *Cyclone* is run. **Session** preferences are in effect until *Cyclone* is closed, and are applied whenever a new ModelSpace View is created. **View** preferences affect the current ModelSpace View only.

DEFAULTS

Reset the selected object type's preferences in the dialog to the hard-wired settings.

Auto-Picking

- Select the **Automatically Pick** checkbox to enable auto-picking, and then make a selection from the drop-down list, which displays points that can be automatically picked for the selected object type.

Catalog/Parts Table Settings

For more information on parts tables, see the [Using Parts Tables](#) section in the *Modeling* chapter.

- Select the Catalog/Parts Table checkbox to enable table control. The table control lists the known tables for the selected object type.

TABLE

Select the appropriate table for the selected object type.

ITEM

Select the desired part from the selected table. This is the default part used in insertion; it has no bearing on fitting to a cloud.

MATCH TABLE WITHIN

Enter a value to determine the range of possible table matches that are evaluated and marked when an item is fit.

Fit Tolerances Settings

- Select the **Fit Tolerances** checkbox to access tolerance settings for the selected object type and enable fit tolerances for use in fitting objects.

- If a fit that you are trying to make is beyond one or more of these limits, a warning message appears.

MEAN ERROR

Assign a value for the maximum allowable mean of the signed point-to-surface distances for all points. The sign of the distance depends on which side of the surface the point lies.

STD DEVIATION

Assign a value for the maximum allowable standard deviation of the signed point-to-surface distances.

MEAN ABSOLUTE ERROR

Assign a value for the maximum allowable mean of the unsigned point-to-surface distances for all points.

ABSOLUTE STD DEVIATION

Assign a value for the maximum allowable standard deviation of the unsigned point-to-surface distances.

MAXIMUM ABSOLUTE ERROR

Assign a value for the maximum allowable unsigned distance between a point and the surface.

Fit Constraints Settings

- Select the **Fit Constraints** checkbox to access constraint settings for the selected object type and enable fit constraints for use in fitting objects.
- Each constraint may be **Unspecified**, **Initial**, or **Fixed**. The value of an **Unspecified** constraint is found by the fitting algorithm. An **Initial** value can help the fitter as a hint of what the value might be. The fitter uses a **Fixed** constraint as the final value of that parameter.
- The editable fitting constraints vary for each object type.

Creation/Initial Values Settings

- Select the **Creation/Initial Values** checkbox to access and enable creation and initial values settings for the selected object type.
- All object types support the optional setting of various default parameters for use when an object of that type is inserted. Some values may be overridden by the operation that creates the object.
- Most object types support a default **Insulation Thickness**, to represent insulation on piping, spray-on fire retardant on a steel section, etc., where the dimensions of the object are augmented by the thickness of the insulation or other such covering. The insulation thickness is accounted for

when inserting, fitting, and editing an object. When displaying the object's attributes, as text or graphically, the dimensions are shown with the additional insulation thickness.

To edit object preferences:

1. From the **Edit** menu, select **Object Preferences**. The **Object Preferences** dialog appears.
2. Select the object type you want to edit from the **Object Type** field.
 - Preferences that are not supported by the selected object type are disabled.
3. Select the level at which your changes take effect.
 - To select a level, select **User Default**, **Session**, or **View** from the **Level** drop-down list (see *Level* following). The **View** level is not available when the dialog is opened from the **Navigator** window.
4. Click the check box next to each preference you wish to enable when fitting, creating, or picking objects, and then click **OK**.
 - To accept current changes and continue editing preferences for other object types, click **Apply**.

One-Sided

Window: **ControlSpace, ModelSpace, Registration, Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)**

Menu: **Point Cloud Rendering**

Action: Toggles menu item ON/OFF

Usage: The **One-Sided** command toggles the hiding of cloud points estimated to be on surfaces facing away from the viewer.

- Only point clouds with valid surface normal vector estimates respond to the **One-Sided** render mode.
- The **Point Cloud Rendering | Front** and **Point Cloud Rendering | Back** commands affect which side is hidden. **Front** hides points on surfaces facing the original scanner. **Back** hides points on surfaces facing away from the original scanner.

Open

Window: **Navigator**

Menu: **File**

Action: Executes command

Usage: The **Open** command launches the selected ModelSpace View, Registration, ControlSpace, scan, or image in a new window.

- Individual scans can be viewed in a read-only ModelSpace environment using the **Open** command. In order to edit ModelSpace data, you must create and open a new ModelSpace View.

Open ModelSpace Viewer

Window: Scan Control

Menu: Scanner

Action: Executes command

Usage: The **Open ModelSpace Viewer** command opens a new ModelSpace viewer for the incoming scan data.

- This ModelSpace is created in the ScanWorld chosen as your destination with the [Select a ScanWorld...](#) command.
- The incoming point cloud being acquired is visible in this ModelSpace viewer, but is not selectable until the scan operation is completed or canceled.

Open New Viewer

Window: ModelSpace

Menu: File | Launch

Action: Executes command

Usage: The **Open New Viewer** command launches a new viewer on the current ModelSpace View.

- Use this command to view the same ModelSpace View from a different viewpoint.

Open Temporary ModelSpace View

Window: Navigator

Menu: File

Action: Executes command

Usage: The Open Temporary ModelSpace View command creates and opens a ModelSpace View for the selected ModelSpace. Unless it is saved, the ModelSpace View is deleted when closed.

Optimize

Window: Navigator

Menu: Tools

Action: Executes command

Usage: The **Optimize** command performs general consistency checking, updating, and reorganizing of the selected object. Different objects in the **Navigator** optimize differently.

- Optimizing can improve loading, browsing, and editing speed.

Note: Depending on the size and complexity of the object being optimized, this operation can be time consuming.

Note: When a database from an older version of *Cyclone* is optimized, a new ControlSpace is created for each ScanWorld. These ControlSpaces are populated with scanned HDS Targets and all objects used in registration constraints.

Optimize Cloud Alignment

Window: Registration

Menu: Cloud Constraint

Action: Executes command

Usage: The **Optimize Cloud Alignment** command is used to adjust cloud constraint alignments for use in global registration.

- For more information on the cloud registration process, see the [Cloud Registration](#) entry in the *Registration* chapter.

To optimize cloud alignments:

1. From the **Constraint List** tab, select the cloud constraints to be optimized.
2. From the **Cloud Constraint** menu, select **Optimize Cloud Alignment**. A progress dialog appears and displays an estimate of percentage completed and a continuously updated error histogram.
 - To set the optimization parameters, see the **Cloud Reg Subsampling Percentage** and **Cloud Reg Max Iterations** parameters in the *Preferences* entry in this chapter.
 - To display the continuously updated error histogram (a histogram of the error between each point and its closest overlapping surface), click **Details**.
 - To interrupt the optimization, click **Stop**. A confirmation dialog appears. Select **Stop Immediately** to stop and discard any results for the current optimization. The state of the cloud constraint is marked as *not aligned*; the cloud constraint must still be aligned before use in global registration. Select **Stop Cleanly** to stop the optimization and save the latest alignment with the cloud constraint. Select **Don't Stop** to continue the alignment optimization.
3. When the optimization is complete, the **Optimize Cloud Alignment Results** dialog appears and displays results and error statistics for each cloud constraint alignment.
 - Cloud constraints that need further adjustments before use in global registration are highlighted.
 - If any alignment searches are halted by the **Cloud Reg Max Iterations** preference (see the *Preferences* entry in this chapter), a warning dialog appears.
 - If any alignment is underconstrained, (e.g., two scans of a plane in which the alignment could be translated along the plane without changing the quality of alignment), a warning dialog appears. This case may still be usable for global registration, but the alignment should be inspected and additional constraints should be added if possible.

Optimize Mesh

Window: ModelSpace

Menu: Tools | Mesh

Action: Executes command

Usage: The **Optimize Mesh** command may speed up working with meshes that have been subject to substantial editing (e.g., deleting and adding faces, moving vertices, filling holes, etc.) in the ModelSpace View.

- Mesh editing can be performed only on a *mesh object*, as opposed to a point cloud that is viewed as a mesh. To create a mesh object from a point cloud, see the [Create Mesh](#) command.

To optimize a mesh object:

1. Select the mesh object to be optimized.
2. From the **Tools** menu, point to **Mesh**, and then select **Optimize Mesh**.

Orthographic/Perspective

Window: ModelSpace

Menu: Viewpoint

Action: Toggles menu item

Usage: The **Orthographic/Perspective** command toggles the ModelSpace View from Perspective to Orthographic.

- In orthographic projection, parallel lines remain parallel, though perceived angles may change. It may be difficult to determine relative depth, but this view is often preferred by architects and engineers in top-down or side views. The real world, however, is viewed in perspective.
- In perspective projection, parallel lines may converge and relative depth may be easier to visualize. The real world is viewed in perspective.

Pan (Lock)

Window: ModelSpace

Menu: Viewpoint | View Lock

Action: Toggles option ON/OFF

Usage: The **Pan** command toggles the ability to pan in **View** mode, which allows you to slide the viewpoint vertically or horizontally within the viewer.

- When **ON**, you cannot pan the viewpoint.

Pan (Reference Plane)

Window: ModelSpace

Menu: Tools | Reference Plane

Action: Toggles option ON/OFF

Usage: The **Pan** command toggles the pan function in **Drawing** mode, which allows you to interactively position the active Reference Plane's origin within the plane without changing its orientation.

To pan the Reference Plane:

1. From the **Tools** menu, point to **Reference Plane**, and then select **Pan** (or click the **Pan** icon). You are now in **Drawing** mode. The cursor changes to the **Drawing Tool** cursor and the **Pan** function is ON.
2. In the ModelSpace, drag the **Drawing Tool** cursor to set the origin of the active Reference Plane.
3. Release the mouse button. The Reference Plane is moved.

Panoramic

Window: Scan Control

Menu: Scanner Control | Set Fence

Action: Executes command

Usage: Pointing to **Panoramic** displays submenu commands used to select a preset panoramic fence size.

- This command does not apply to the HDS2500.

The following preset panoramic fences are available from the **Panoramic** submenu:

360 x 40, 360 x 60, 360 x 80, 360 x 90, [Target All](#)

Panoramic Rectangle Fence Mode

Window: Scan Control

Menu: Edit | Modes

Action: Executes command

Usage: The **Panoramic Rectangle Fence Mode** command enters **Panoramic Rectangle Fence** mode and selects the **Panoramic Rectangle Fence** mode cursor.

- This command does not apply to the HDS2500.

To draw a panoramic fence:

1. From the **Edit** menu, point to **Modes**, and then select **Panoramic Rectangle Fence Mode**.
The **Panoramic Rectangle Fence** cursor appears.
2. Drag the cursor around the area you want to fence.
 - Drag up/down to define the vertical bounds of the fence.
 - Drag clockwise to define a fence initialized at 0 degrees.
 - Drag counter-clockwise to define a fence initialized at 360 degrees.

Panoramic Scan Mode

Window: Scan Control (HDS4500, HDS6000)

Menu: Scanner Control

Action: Executes command

Usage: Pointing to **Panoramic Scan Mode** displays submenu commands used to select the scanning mode used for 360-degree scans. **Face 1 Scan** makes the scanner scan from the front, performing a full 360-degree turn. **Face 1 and 2 Scan** makes the scanner scan from both sides simultaneously, performing a 180-degree turn.

The following panoramic scan modes are available from the **Panoramic Scan Mode** submenu:

Face 1 Scan, Face 1 and 2 Scan

Parallel

Window: ModelSpace

Menu: Edit Object | Align

Action: Executes command

Usage: The **Parallel** command aligns the one or more objects so that their axes are parallel to a reference object.

To make objects parallel:

1. Multi-select the objects you want to make parallel.
 - The last object selected is the reference object (the one to which the other aligns).
2. From the **Edit Object** menu, point to **Align**, and then select **Parallel**. The objects are rotated to make their axes parallel to the axis of the reference object.
 - This command is useful when modeling walls of a room in order to make sure opposite pairs are parallel.
 - The shortest rotation possible is used to align the objects, so objects may end up pointing in opposite directions, though parallel.

Paste

Window: ModelSpace, Navigator

Menu: Edit

Action: Executes command

Usage: The **Paste** command pastes the contents of the *Cyclone* clipboard into the selected location.

- Objects pasted into a **ModelSpace** maintain their location and orientation relative to the current coordinate system in the destination ModelSpace.

Note: The **Navigator's** clipboard behaves differently from the **ModelSpace** clipboard; when **Copy** is selected, the selected object is *marked* for copying; however, it is not actually copied until you initiate the **Paste** command. Additionally, you have a choice to create a **Full Copy** or a **Reference**. The reference copy does not include point clouds or large numbers of image files.

Paste Link

Window: Navigator

Menu: Edit

Action: Executes command

Usage: The **Paste Link** command pastes a link to an item on the *Cyclone* clipboard in the selected location.

- Links are intended to make navigation easier. They allow you to create a shortcut to any Project, ScanWorld, ModelSpace, or Image, as well as allowing you to reference any object in multiple places.

To create a link:

1. Select the Project, ScanWorld, ModelSpace, or Image to which you want to link.
 2. From the **Edit** menu, select **Copy**.
 3. Select a location for the link.
- Links can be pasted into ScanWorlds, Scans, ModelSpaces, and Projects.
4. From the **Edit** menu, select **Paste Link**. The link appears in the selected location.
- Links are identical to their source object, and renaming a link also renames the source object.
 - Links can only be pasted into the same database as the source object
- Note:** If you have copied multiple selections and then use the **Paste Link** command, only the last object selected is linked.

Paste Reference

Window: **Navigator**

Menu: Edit

Action: Executes command

Usage: The Paste Reference command is used to paste an object (e.g., a point cloud) with references to the point cloud's data in the source database, saving time and space by avoiding duplication of data.

- When the destination database needs to access the point cloud's data, the data is retrieved from the source database (or another database that has that information; the database that has the data must be accessible on the client's local server).

Patch

Window: **ModelSpace**

Menu: **Create Object | From Curves**

Action: Executes command

Usage: The **Patch** command creates a patch from a series of selected lines or curves.

To create a patch from intersecting curves:

1. Select a sequence of lines or curves that form the outline of the patch you want to create.
 - The lines or curves must be selected in a consistently clockwise or counter-clockwise order.
2. From the **Create Object** menu, point to **From Curves**, and then select **Patch**. The patch is created.

Patch (Submenu)

Window: **ModelSpace**

Menu: **Edit Object**

Action: Displays submenu

Usage: Pointing to **Patch** displays submenu commands used to edit a selected patch.

The following commands, dialogs, and submenus are available from the **Patch** submenu:

[Make Rectangular](#), [Make Circular...](#), [Subtract](#), [Fill Selected Hole](#)

Pause

Window: Scan Control (ScanStation, ScanStation 2, HDS3000, HDS6000)
Menu: Scanner Control
Action: Executes command
Usage: The **Pause** command pauses the current scanner operation (scan or image acquisition). Push the **Resume** button to continue the operation, or the **Stop** button to discontinue the operation.

Perpendicular

Window: ModelSpace
Menu: Edit Object | Align
Action: Executes command
Usage: The **Perpendicular** command aligns one or more selected objects so that their axes are perpendicular to a reference object.

To make objects perpendicular:

1. Multi-select the objects you want to make perpendicular.
 - The last object selected is the reference object (the one to which the others align).
2. From the **Edit Object** menu, point to **Align**, and then select **Perpendicular**. The objects are rotated to make their axes perpendicular to the axis of the reference object.

Perspective/Orthographic

Window: ModelSpace
Menu: Viewpoint
Action: Toggles menu item
Usage: The **Perspective/Orthographic** command toggles the ModelSpace View from Orthographic to Perspective.

- In perspective projection, parallel lines may converge and relative depth may be easier to visualize. The real world is viewed in perspective.
- In orthographic projection, parallel lines remain parallel, though perceived angles may change. It may be difficult to determine relative depth, but this view is often preferred by architects and engineers in top-down or side views. The real world, however, is viewed in perspective.

Pick Mode

Window: ModelSpace, Registration, Scan Control

Menu: ModelSpace: Edit | Modes

Registration: Edit | Modes

ScanControl: Viewers

Action: Executes command

Usage: The **Pick Mode** command enters **Pick** mode and selects the **Pick** mode cursor.

- **Pick** mode is used to select and interact with objects.

Pick/Multi-Pick/View

Window: ModelSpace

Menu: Edit | Modes

Action: Executes command

Usage: The **Pick/Multi-Pick/View** command cycles between **Pick** mode, **Multi-Pick** mode and **View** mode.

- The main purpose of this command is to allow for a single hotkey to cycle through the modes.
To assign hotkeys, see the *Customize Hotkeys* entry in this chapter.

Pick/View Toggle

Window: ModelSpace

Menu: Edit | Modes

Action: Executes command

Usage: The **Pick/View Toggle** command switches between **Pick** mode and **View** mode.

- The main purpose of this command is to allow for a single hotkey to toggle between the two modes. To assign hotkeys, see the [Customize Hotkeys](#) command.

Pipe Modeling...

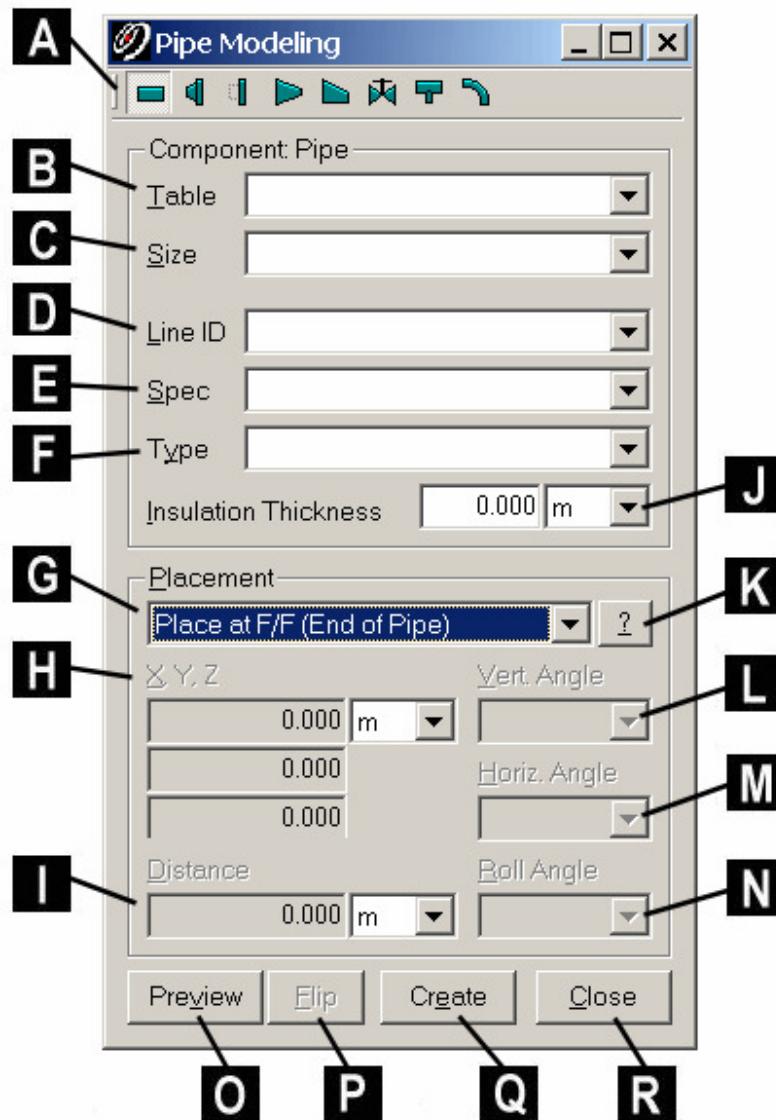
Window: ModelSpace

Menu: Tools | Piping

Action: Opens dialog

Usage: The **Pipe Modeling** dialog is used to create pipes and piping-related objects with piping-specific information and preferences.

Pipe Modeling Dialog



- A.** The type of piping component to be created.
- B.** The parts table.
- C.** The specific part from the table, used to define the dimensions of the component.
- D.** The current Line Identifier (used to identify a particular pipe run).
- E.** The current Specification (used to place restrictions during piping design; in *Cyclone*, this is simply a text field).
- F.** The current Symbol Key for each object type.

- G.** The method used to insert the component. Push the **?** button for instructions on how to use the method.
- H.** The position of the component, typically one of the component's end-points. Disabled for some **Placement** methods.
- I.** Define the position of the component as an offset from the pick point when the **Placement** method is **Place at Delta Distance**.
- J.** The insulation thickness applied to components created by this dialog.
- K.** Push to see instructions on how to use the **Placement** method.
- L.** The vertical angle of the component, in degrees. **0** is along the "up" axis and **90** is lying in the horizontal plane. Disabled for some **Placement** methods.
- M.** The horizontal angle of the component, in degrees. **0** is along the +Y axis (Northing) and **90** is along the +X axis (Easting). Disabled for some **Placement** methods.
- N.** The roll angle of the component about its centerline, in degrees. Disabled for some **Placement** methods and components.
- O.** Push to display a graphical preview of the component.
- P.** Push to flip the component to use the other end-point for placement.
- Q.** Push to create the component. The created component is automatically picked so that the next component placed will be at the component's end-point not used for placement. If possible, the created component is connected (see the *Connect Piping* entry in this chapter) with the previously-selected component.
- R.** Closes the dialog.

To model pipes:

1. From the **Tools** menu, point to **Piping**, and then select **Pipe Modeling....** The **Pipe Modeling** dialog appears.
2. Select a type of component to place from the toolbar.
3. Select a parts table.
4. Select an entry from the parts table.
5. Enter or select a **Line ID** from the pulldown menu.
6. Enter or select a **Spec**.
7. Select a **Type** (SKEY) from the pulldown menu.
 - An SKEY (symbol key) identifies a generic object (e.g., a standard *Cyclone* flange) as a specific piping component (e.g., "Slip on Flange"). The 2-4 characters at the end of the SKEY comprise the encoding of that SKEY.

- Each type of piping component has its own SKEY, and each type of piping component can be assigned only those SKEYs available to that type.
 - **Note:** Spec, LineID, and Type (SKEY) are stored as annotations on the objects that have them. They are displayed in the **Object Info** dialog box and can be managed via the standard Annotations functionality (although changing the values via the **Edit Annotations** dialog is not recommended).
8. Enter the **Insulation Thickness**.
 9. Select the **Placement** method. Push the ? button for instructions on how to use the method.
 10. Enter and edit the components's placement parameters.
 11. **Preview** the component's placement graphically.
 12. **Flip** the component to place the component using the other end-point, if needed.
 13. **Create** the component.
 14. Select the type of the next component to be placed and repeat.
 15. When finished, click **Close**.

Placement Methods:

- **Place at F/F (End of Pipe)** – Appends a component to another component (fitting-to-fitting).
- **Place at Delta Distance** – Places a component at a distance from another component.
- **Place at Position** – Places a component at a specified position.
- **Connect Fittings with Pipe** – Places a pipe between two in-line fittings, such as two flanges.
- **Insert at Pick Point** – Inserts a fitting into a pipe at a pick point. The pipe's geometry is adjusted accordingly (possibly splitting into two segments).
- **Insert at F/F (Closest Fitting)** – Inserts a fitting at the end of a pipe (fitting-to-fitting).
- **Connect Pipes with Fitting** – Connects two intersecting pipes with a fitting (elbow or tee).

Piping

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Piping** displays submenu commands used to establish and edit piping connections and annotations.

The following commands and dialogs are available from the **Piping** submenu:

[Pipe Modeling...](#), [Piping Mode...](#), [Elbow Connectors](#), [Edit Elbow Bend Ratio...](#), [Reducer Connectors](#), Eccentric Reducer Connectors, [Miter Connectors...](#), [Convert Elbow to Miter...](#), [Create Branch](#), [Connect Piping](#), [Disconnect \(Shared\)](#), [Disconnect \(All\)](#), [Select Connected](#)

Piping Mode...

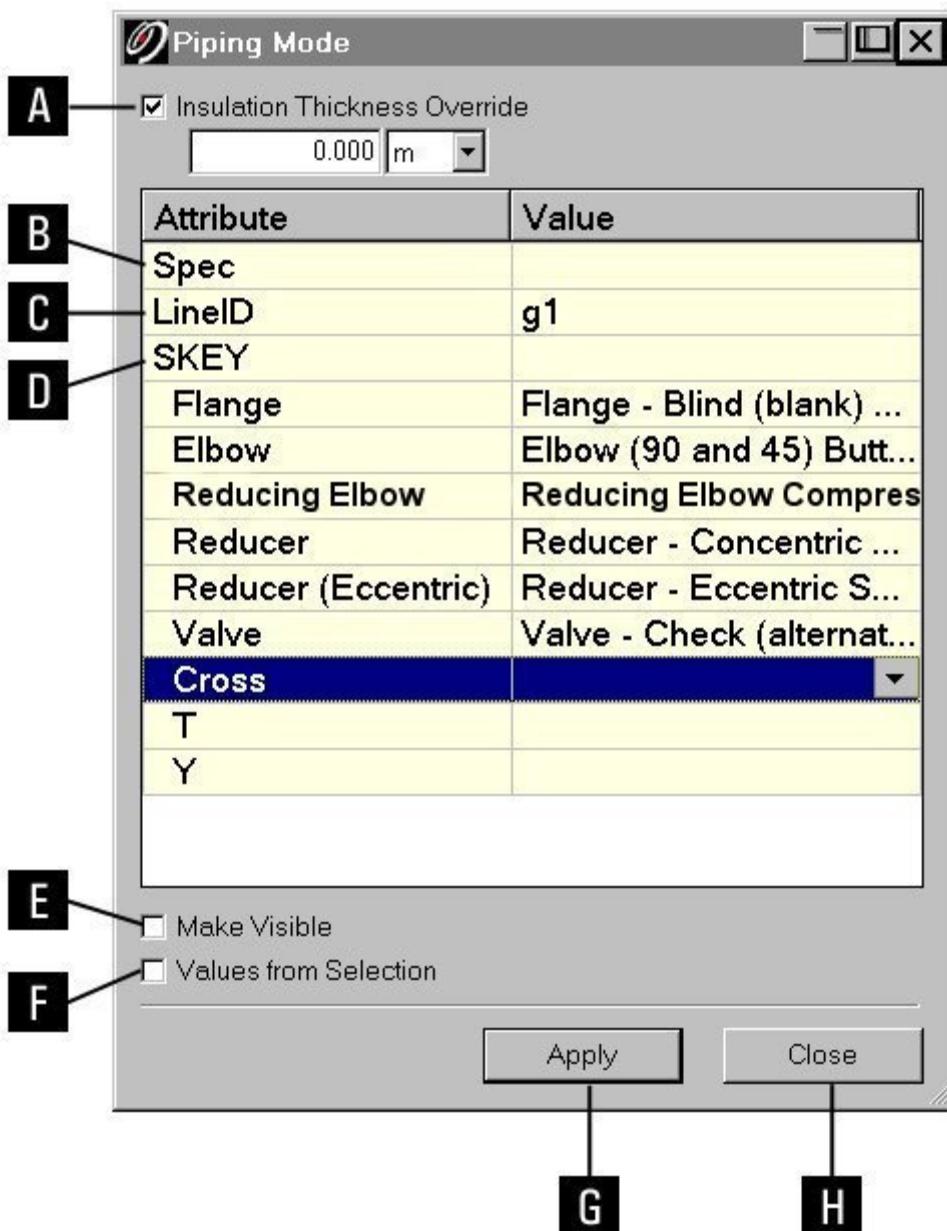
Window: ModelSpace

Menu: Tools | Piping

Action: Opens dialog

Usage: The **Piping Mode** dialog is used to assign piping-specific information and preferences to modeled objects.

Piping Mode Dialog



- A.** Activate to specify the active insulation thickness applied to objects created via fitting and insertion while this dialog is open. The value specified overrides any insulation thickness values set in **Object Preferences** (see the *Object Preferences* entry in this chapter).
- B.** The current Specification (used to place restrictions during piping design; in *Cyclone*, this is simply a text field)
- C.** The current Line Identifier (used to identify a particular pipe run).
- D.** The current Symbol Key for each object type.
- E.** Activate to display the Spec, LineID, and SKEY associated with newly created objects.
- F.** Activate to copy the Spec, LineID, and SKEY from the last selected object into this dialog. The SKEY copied is appropriate to the type of object selected.
- G. Sets** the Spec, LineID, and SKEY for the selected objects
- H.** Closes the dialog and exits Piping mode.

To apply piping annotations:

1. From the **Tools** menu, point to **Piping**, and then select **Piping Mode...**. The **Piping Mode** dialog appears.
 - While the **Piping Mode** dialog is open, the current **Spec**, **LineID**, and appropriate **SKEY** is assigned to piping components created via the **Fit to Cloud**, **Region Grow**, **Insert Object**, and **Piping Connectors** commands. For example, when an elbow is created using the **Piping Connectors | Elbow Connectors** command, it is assigned the current **Spec** and **LineID**, as well as the elbow **SKEY**.
 - Also, while the **Piping Mode** dialog is open, connectivity is automatically established when inserting a piping component via a pick point on another piping component, subject to the connection rules (see the *Connect Piping* entry in this chapter).
 - As long as the **Piping Mode** dialog is open, the current piping annotations are applied to new objects as they are inserted or created.
2. Select the desired piping annotations.
 - To use an object's existing piping annotations to set the fields in the **Piping Mode** dialog, select **Values from Selection** and select the object with the needed values. (Piping annotations that are not assigned to the selected object are not changed.)

Note: Spec, LineID, and SKEYs are stored as annotations on the objects that have them. They are displayed in the **Object Info** dialog box and can be managed via the standard Annotations functionality (although changing the values via the **Edit Annotations** dialog is not recommended).

3. Select a **Spec** from the pulldown menu.
 - To choose which Specs are displayed in the pull-down menu, right-click the **Spec** row and select **Choose Lookup List for Spec** from the pop-up menu. (If there are no lookup lists

available, see the *Customize Lookup Lists* or *Custom Annotations* entry in this chapter to generate custom lists.) Select the lookup list with the relevant Specs. The **Spec** field cannot be edited directly, since only certain Specs are typically available for a given project.

4. Select a **LineID** from the pulldown menu.
 - Choosing a lookup list for LineID is similar to choosing a lookup list for Spec.
 - The **LineID** field may be edited directly; and the new value is added to the list.
5. Select an **SKEY** from the pulldown menu.
 - An SKEY (symbol key) identifies a generic object (e.g., a standard *Cyclone* flange) as a specific piping component (e.g., "Slip on Flange"). The 2-4 characters at the end of the SKEY comprise the encoding of that SKEY.
 - Each type of piping component has its own SKEY, and each type of piping component can be assigned only those SKEYs available to that type.
 - To select which SKEYs are present in the pulldown (e.g., to limit the SKEYs to those components that will be encountered in the current project instead of all possible components), right-click on the row of the object type and select **Customize**. Next, indicate which SKEYs are to be available in the pulldown by checking them in the **Customize SKEYs** dialog.
6. Select the desired piping components and click **Apply**. The selected piping components are (re)assigned the current Spec and LineID, as well as the SKEY appropriate to the type of piping component selected.
7. When finished, click **Close**.
 - The **Piping Mode** dialog provides the automatic functionality only while the dialog is open. For example, when using cylinders and elbows to represent railing or conduit, the user should close the **Piping Mode** dialog to avoid assigning data irrelevant to such objects.

Piping Takeoff

Window: ModelSpace

Menu: Tools | Measure

Action: Executes command

Usage: The **Piping Takeoff** command calculates and displays the takeoff for the selected piping run.

- The takeoff is similar to the centerline length, except that the length of an elbow or bend is not the arc length, but the sum of the distances from each of its endpoints to the intersection points of its endpoint tangents.

To measure piping takeoff:

1. Multi-select the sequence of piping (including the connections) you want to measure.
2. From the **Tools** menu, point to **Measure**, and then select **Piping Takeoff**. The measurement is displayed in the Output Box.

Point Cloud Density

Window: ModelSpace, Registration

Menu: ModelSpace: View

Registration: Viewers

Action: Displays submenu

Usage: Pointing to **Point Cloud Density** displays submenu commands used to adjust the automatic thinning of cloud points when rendering. Increasing the level of reduction can improve performance and visualization. Other computations are unaffected by this setting.

Point Cloud Density Submenu Commands

NO REDUCTION

Point clouds are drawn with full level of detail.

LOW REDUCTION

Point clouds are drawn without some points.

MEDIUM REDUCTION

Point clouds are drawn somewhat sparsely.

HIGH REDUCTION

Point clouds are drawn sparsely.

HIGHEST REDUCTION

Point clouds are drawn very sparsely.

Point Cloud Rendering

Window: ControlSpace, ModelSpace

Menu: View

Action: Displays submenu

Usage: Pointing to **Point Cloud Rendering** displays submenu commands used to toggle different ways of drawing point clouds.

- The following modifiers are available from the **Point Cloud Rendering** submenu:

Shaded, One-Sided, Front, Back, Silhouette, Increase Point Width, Decrease Point Width

Point Cloud Rendering (Registration)

Window: Registration
Menu: Viewers
Action: Displays submenu
Usage: Pointing to **Point Cloud Rendering** displays submenu commands used to toggle different ways of drawing point clouds.

- The following modifiers are available from the **Point Cloud Rendering** submenu:

Shaded, One-Sided, Front, Back, Silhouette, Increase Point Width, Decrease Point Width

Point Cloud Rendering (Scan Control)

Window: Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)
Menu: Project
Action: Displays submenu
Usage: Pointing to **Point Cloud Rendering** displays submenu commands used to toggle different ways of drawing point clouds.

- The following modifiers are available from the **Point Cloud Rendering** submenu:

Shaded, One-Sided, Front, Back, Silhouette, Increase Point Width, Decrease Point Width

Point Cloud Sub-Selection

Window: ModelSpace
Menu: Selection
Action: Displays submenu
Usage: Pointing to **Point Cloud Sub-Selection** displays submenu commands used to make and refine point cloud selections to be used for segmentation, fitting, and other processes.

The following commands are available from the **Point Cloud Sub-Selection** submenu:

[Add Inside Fence](#), [Add Outside Fence](#), [Remove Inside Fence](#), [Remove Outside Fence](#), [Invert](#), [Select All](#)

- For more information, see the [Point Cloud Segmentation](#) entry in the *Modeling* chapter.

To make a point cloud sub-selection:

- Draw a rough fence containing all the points that you may want to use.

2. Using the **Selection | Point Cloud Sub-Selection** submenu commands, create a temporary selection.
3. Continue to refine the temporary selection (add or subtract points) by manipulating the viewpoint, drawing a new fence, and using subsequent **Selection | Point Cloud Sub-Selection** submenu commands. The points within the temporary subset can now be segmented, fit, or used for other processes.

Point Distance

Window: ModelSpace

Menu: Tools | Measure | Datum

Action: Executes command

Usage: The **Point Distance** command measures the distance between a picked point and the datum.

Point-Datum Distance Measurement Output

- Depending on the current "up" direction, one of the X, Y, and Z distances is not shown

HORIZONTAL DISTANCE

The distance in the horizontal plane between the picked point and the datum.

X DISTANCE

The distance along the X axis between the picked point and the X coordinate of the datum.

Y DISTANCE

The distance along the Y axis between the picked point and the Y coordinate of the datum.

Z DISTANCE

The distance along the Z axis between the picked point and the Z coordinate of the datum.

To take a measurement from the picked point to the datum:

1. Pick the point from which you want to measure.
2. From the **Tools** menu, point to **Measure**, then point to **Datum**, and select **Point Distance**. The measurement is calculated and displayed in the **Output** dialog.
 - Datum measurements cannot be saved. However, as long as the **Output** dialog remains enabled, multiple subsequent measurements can be made and displayed.
 - If multiple points are being measured, the results are displayed in the order in which they were picked.

Pointing

Window: Scan Control (ScanStation 2)

Menu: Window

Action: Expands Tab

Usage: The **Pointing** command expands the **Pointing** tab in the Scanner Control Panel.

- Pointing lets you aim the scanner's laser at a specific point or in a specific direction.
- The Pointing procedure supports several methods. Depending on the method, some controls cannot be changed.

Pointing Tab Options

METHOD

Displays the current method used to aim the scanner's laser.

ORTHOGONAL FROM STATION

Select a known coordinate in the **Target ID** field. This known coordinate is the starting point at which to aim the laser. This method requires Field Setup to first be applied to establish the scanner's coordinate system.

MANUAL – POLAR

Enter the **Azimuth** and **Distance** to the point at which to aim the laser. The elevation of the point remains the same when **Azimuth** and **Distance** are edited.

MANUAL – ORTHOGONAL

Enter the coordinates at which to aim the laser.

PICK POINT

In the Scan Control viewer, pick the point at which to aim the laser.

TARGET ID

When **Method** is **Orthogonal from Station**, selects the target to display in the panel. The laser will be aimed at the target's coordinates.

HT

The height of the target above the coordinate.

AZIMUTH

The horizontal angle towards the point at which to aim the laser.

STEP

The change in angle when an arrow key (see below) is pushed.

DISTANCE

The distance to the point at which to aim the laser.

X Y Z / N E EL

The coordinate of the point at which to aim the laser. The Scan Control viewer displays a cross-hair at this coordinate.

ARROWS (UP, DOWN, LEFT, RIGHT)

Push to change the elevation (vertical change) or azimuth of the point at which to aim the laser.

POINT / STOP

Push **Point** for the scanner to turn and point the laser at the configured coordinate. When in this mode, the button text changes to **Stop**. Use the arrow buttons to re-aim the laser. Push the QuickScan button to disengage or re-engage the scanner's base motor. Push **Stop** to stop aiming the laser.

To point the laser:

1. Connect to the scanner (ScanStation 2) and ensure that it is level.
2. If desired, perform a Field Setup to establish the scanner's coordinate system (this is required when using the **Orthogonal from Station** method). The azimuth and coordinates of the target point are relative to this coordinate system.
3. If using the **Orthogonal from Station** method, import or add the known or assumed coordinate of the point, using **Add/Replace Coordinate**, **Import Coordinate List**, or **Import Coordinate List (Leica System 1200)**.
4. From the **Window** menu, select **Pointing**. The **Pointing** panel appears in the control panel.
5. Select the pointing **Method**.
6. If using the **Orthogonal from Station** method, enter the **Target ID**. The **Target ID** must match a known coordinate.
7. Enter the **HT** of the target point (if any) relative to the coordinate at which to aim the laser.
8. Enter the coordinate or angles of the point at which to aim the laser.
 - If using the **Orthogonal from Station** method, the coordinates are already known.
 - If using the **Manual – Polar** method, enter the **Azimuth** and **Distance** to the point.
 - If using the **Manual – Orthogonal** method, enter the 3D coordinate of the point.
 - If using the **Pick Point** method, pick the point in the Scan Control viewer.
9. Push the **Point** button to activate the laser. The scanner turns to aim the laser at the configured point.
10. Use the arrow buttons to adjust the azimuth and elevation. The scanner turns to aim the laser at the new position.
11. Push the QuickScan button once to disengage the base motor and manually turn the scanner. Push the QuickScan button a second time to re-engage the base motor.
12. Push the **Stop** button to stop pointing.

Polygonal Fence Mode

Window: ModelSpace, ScanControl (HDS2500)

Menu: Edit | Modes

Action: Executes command

Usage: The **Polygonal Fence Mode** command enters **Polygonal Fence** mode and selects the **Polygonal Fence** mode cursor.

- For more information on working with fences, see the [Using a Fence](#) entry in the *Modeling* chapter.

To draw a polygonal fence:

1. From the **Edit** menu, point to **Modes**, and then select **Polygonal Fence Mode**. The **Polygonal Fence** cursor appears.
2. Click to place the first vertex.
 - Drag to adjust the vertex position.
 - To continuously add vertices as you move the mouse, press and hold **CTRL**, and drag the cursor.
3. Continue adding vertices as needed.
 - A vertex can also be added by clicking anywhere along the line segment between existing vertices.
 - An existing vertex can be deleted by clicking the vertex without moving the mouse.
 - The newly drawn fence can adjusted by dragging its vertices with the **Polygonal Fence** cursor.
 - To clear the newly drawn fence, select **Clear** from the **Fence** submenu.

Polyline (From Curves)

Window: ModelSpace

Menu: Create Object | From Curves

Action: Executes command

Usage: The **Polyline** command creates a polyline from a series of selected lines or curves.

To create a polyline from intersecting curves:

1. Select a sequence of lines or curves that are to form the polyline you want to create.
2. From the **Create Object** menu, point to **From Curves**, and then select **Polyline**. The polyline is created.

Polyline (From Pick Points)

Window: ModelSpace

Menu: Create Object | From Pick Points

Action: Executes command

Usage: The **Polyline** command creates line segments that connect a series of picked points.

To create a polyline:

1. Multi-select a series of points.
2. From the **Create Object** menu, point to **From Pick Points**, and then select **Polyline**. The polyline is created.
 - A polyline is drawn from point to point in the same order in which the points were picked.
 - To edit a polyline, select it and then use the vertex handles to change its shape.
 - To draw a polyline without the use of pick points, see the *Draw Polyline* entry in this chapter.

Polylines

Window: ModelSpace

Menu: Tools | Mesh

Action: Displays submenu

Usage: Pointing to **Polylines** displays submenu commands used to manipulate polylines with regards to mesh objects or vice versa.

The following commands are available from the **Polylines** submenu:

[Project Polyline to TIN](#), [Extend TIN to Polyline](#).

Preferences...

Window: ModelSpace, Navigator, Registration, Scan Control, Image

Menu: Edit

Action: Opens dialog

Usage: The **Edit Preferences** dialog is used to set a variety of *Cyclone* settings.
Level Options

SESSION

These preferences are non-persistent and last for the duration of the current session only. The next time *Cyclone* is started, these preferences are reset to default settings.

DEFAULT

These preferences are persistent and apply to the current and all future sessions. They may be temporarily overridden by making preference changes at the Session level.

Units Tab Settings

Units settings control the units used throughout *Cyclone*.

LINEAR UNITS

Select the type of unit used for linear measurements.

IMPERIAL TYPES

Select **English** or **U.S. Survey**. **English** uses the factor of *25.4 millimeters/inch*, while **U.S. Survey** uses *39.37 inches/meter*.

STATION NOTATION

Toggle station notation **ON/OFF**. When this preference is **ON** and there is an active alignment, alignment coordinates are displayed for pick points.

ALIGNMENT OFFSET NOTATION

Select a style for horizontal offset notation when using station notation.

ANGULAR UNITS

Select the type of unit used for angular measurements.

COORDINATE LABELS

Select **X, Y, Z** or **N, E, EI** (Northing, Easting, Elevation) coordinate labels for all dialogs, menus, and other occurrences of coordinate labels.

TEMPERATURE UNITS

Select the unit used to display temperature measurements.

PRESSURE UNITS

Select the type of unit used for pressure measurements.

Resolution Tab Settings

Resolution settings allow you to adjust the resolution with which values are displayed throughout *Cyclone*, by setting the number of decimal digits to be displayed. For example, with a **Linear Units** of *meters*, setting the **Decimal Digits: Distance** to 2 means that all distance measurements are displayed to a resolution with enough decimal places to represent one centimeter. The **Decimal Digits** setting is also available for **Area**, **Volume**, **Angle**, **Temperature**, **Pressure**, **Time**, and **Percentage**.

Most resolution settings are self-explanatory. Those that are not are described below.

DECIMAL DIGITS: UNITLESS

Set the number of decimal places for the display of unitless measurements such as direction vectors.

STATION PLUS DIGITS

Sets the format used for station notation.

Measurements Tab Settings

Measurement settings allow you to adjust the display of measurements made in the **ModelSpace** window.

BODY COLOR

Use the color picker to select the color to be used for the display of measurement lines.

LABEL COLOR

Use the color picker to select the color to be used for the display of measurements.

FONT NAME

Type the name of the font you want to use to display measurements. The font must be currently installed on your workstation.

FONT SIZE

Enter a value for the font size used to display measurements.

CALLOUT LINE LENGTH

Enter a value for the length of callout lines.

CALLOUT LINE ANGLE

Enter a value for the angle of callout lines.

TIN VOLUME: FILL COLOR

Use the color picker to specify the color of the TIN Volume Fill lines in the ModelSpace View.

TIN VOLUME: CUT COLOR

Use the color picker to specify the color of the TIN Volume Cut lines in the ModelSpace View.

Registration Tab Settings

AUTO-ADD CONSTRAINT TOLERANCE

Enter a value between 0.0001 meter and 1.0 meter for the tolerance used for the **Auto-Add Constraint** function. This is the maximum distance used when considering which objects are geometrically close to each other.

MAX ITERATIONS

Enter a value between 1 and 1000 for the maximum number of iterations performed by registration.

CONVERGENCE CRITERIA

Enter a value to define the smallest improvement in registration error after which the registration optimization ends. A smaller number may make registration take longer.

CLOUD REG: DEFAULT MAX SEARCH DISTANCE

Enter a value for the maximum distance to be searched matching pick points during cloud registration. Setting this value above the default (four inches) may decrease the speed and reliability of the optimal alignment.

CLOUD REG: DEFAULT SUBSAMPLING PERCENTAGE

Enter a percentage to specify the percentage of each cloud used with the **Optimize Cloud Alignment**. This setting amounts to a speed versus accuracy trade-off. The default is 3%, which is effective and efficient for typical scan cloud sizes (i.e., point clouds with one million points). In general, a value of 100% may give results up to twice as accurate as a value of 3%, but may take approximately ten times as long.

CLOUD REG: DEFAULT MAX ITERATIONS

Enter a value for the number of iterations above which a warning appears when the **Optimize Cloud Alignment** function has not converged on a solution.

CLOUD REG: DEFAULT INITIAL GUESS METHOD

Select a method to use as the starting point for cloud registration calculations.

Fitting Tab Settings

MULTIPLE POINT CLOUD STYLE

Fit Multiple Objects fits a separate object to each point cloud when more than one point cloud is selected. **Fit Single Object** fits a single object to selected multiple point clouds.

NUMBER OF POINTS WARNING THRESHOLD

Enter a value for the number of points above which a warning appears before a fitting operation begins.

Modeling Tab Settings

REGION GROWING: TOLERANCE

Enter a value between 0.001 meter and 0.1 meter which defines the initial tolerance for region growing.

FIT EDGE: MIN STEP SIZE

Enter a value for the minimum interval distance to fit templates to the surface of the point cloud. A value of zero lets *Cyclone* automatically set this parameter.

FIT EDGE: MAX STEP SIZE

Enter a value for the maximum interval distance to fit templates to the surface of the point cloud. A value of zero lets *Cyclone* automatically set this parameter.

FITTING: ADJUST PATCH BOUNDARY

Select **Yes** to display the **Adjust Patch Boundary** dialog automatically when fitting a concave patch.

INTERFERENCE: TOLERANCE

Enter a value between 0.001 meter and 10.0 meters to define the initial tolerance used when determining if points are interfering with other geometry in a ModelSpace when using the **Interference Points** command in the **Measure** submenu.

TEXTURE MAP: TOLERANCE: MAXIMUM ERROR (PIXELS)

Enter a value between 0.1 pixel and 100.0 pixels to set the maximum pixel error that any single texture map constraint can have for a valid texture map computation.

TEXTURE MAP: TOLERANCE: OVERALL ERROR (PIXELS)

Enter a value between 0.1 pixel and 100.0 pixels to set the highest average pixel error that the texture map's constraints can have for a valid texture map computation.

PIPING: PIPE RADIUS

Leave Radii Unchanged leaves each pipe radius at its current value, and adjusts pipe centerlines so that sequential pipes are coplanar.

Force Uniform Radius forces all pipes to have the same radius. If points are used when refitting the pipes, each pipe's radius is set to the size that best fits all point clouds used in the fitting. Otherwise, each pipe's radius is set to the average of the initial pipe radii.

Refit Individual Radii optimizes each pipe's radius if points are used when refitting the pipe. Otherwise, the pipe's radius is left at its current value.

PIPING: RE-FITTING

Ignore Points disregards points used in the original fitting of the pipe during connection by elbow, and forces sequential pipes to be coplanar by minimal movement of their end points, so that pipes are not refit.

Refit to Points refits pipes using points that were used in the original fitting.

PIPING: MAX. OFFSET (LINEAR OF BRANCH CONNECTION)

Enter a value for the greatest distance allowed between the opposing center points for a linear connection, and the greatest distance allowed between the center point of the branch to the centerline of the run when connecting piping.

PIPING: MAX. ANGLE (LINEAR CONNECTION)

Enter a value for the greatest angle allowed between the tangents of the opposing faces when connecting piping. A value of 0 means that the tangent vectors must be exactly opposite.

PIPING: MIN. ANGLE (BRANCH CONNECTION)

Enter a value for the smallest angle allowed for a branching connection (see *Connect Piping*). A value of 90 means that the branching angle must be exactly perpendicular.

ModelSpace Tab Settings

POINT CLOUD: LENGTH OF GRAPHICAL NORMALS

Set the scale for the display of point cloud normals.

To toggle the display of point cloud normals, see the *View Object As* entry in this chapter.

OBJECT SIZE CULL BIAS

Set the size of objects that are too small or too far away to be loaded and drawn. The value that you enter is the number of square pixels, below which an object is not loaded and rendered.

LAYER OF NEW OBJECT

Original Object's Layer places new objects on the same layer as the object from which they were created.

Current Layer places new objects that are created using an existing object on the current layer regardless of which layer the object is on.

PICKING

Ignore Auto-Pick overrides and disables auto-picking, regardless of the state of the **Auto-Pick Points** toggle in the **Selection** menu. **Use Auto-Pick Instead** allows auto-picking when the **Auto-Pick Points** toggle is ON, and auto-picking preferences have been set for the type of object you wish to pick. See the *Auto-Pick Points* and *Object Preferences* entries in this chapter for more details.

VIEWER: START WITH OUTPUT BOX ENABLED

Select **Yes** to have the Output Box enabled when opening a new ModelSpace window. See the *Enable Output Box* entry in this chapter for more information on Output Boxes.

VIEWER: SAVE LAST VIEWPOINT

Select **Yes** to save the current viewpoint within the ModelSpace View when closing the ModelSpace window. Select **No** if you want to retain the initial viewpoint.

VIEWER: POINT SELECTION TRANSPARENCY

Enter a value between 0 (not transparent) and 1 (completely transparent) to affect how unselected point clouds are displayed when other point clouds are selected.

VIEWER: BACKGROUND COLOR

Use the color picker to specify the background color of the 3D graphics viewers (including ModelSpace viewer, Registration Constraint viewers, etc).

VIEWER: DRAW TEXT ON TOP

Select **Yes** to always draw all graphical text on top of any 3D graphics within the ModelSpace View.

VIEWER: MAX MULTI-IMAGE MEMORY

Enter a value for the maximum amount of memory that can be used to load multi-images.

VIEWPOINT MANIPULATION SENSITIVITY (DEFAULT)

Enter a value between .001 and 10.0 to determine the speed of viewpoint manipulation. A sensitivity of 1.0 is the default; a higher value makes the viewpoint move faster and a lower value makes the viewpoint move slower given the same mouse motion.

VIEWPOINT MANIPULATION SENSITIVITY (ALTERNATE)

Enter a value between .001 and 10.0 to determine the speed of viewpoint manipulation with **ALT** pressed and held. A sensitivity of 0.1 makes the viewpoint moves ten times more slowly than the default of 1.0. A higher value makes the viewpoint move faster and a lower value makes the viewpoint move slower given the same mouse motion.

NAVIGATOR: DOUBLE-CLICK ACTION

Select the desired result when double-clicking a ModelSpace object in the **Navigator** window.

HEAD LIGHT TYPE

Directional light is a light source infinitely far away, such that all the rays of light are parallel.

Point source light (default) is a light source where all the rays of light emanate from a single 3D point (the camera's position).

TIN OVERLAP THRESHOLD

Specify the minimum distance between any two points on the horizontal plane. If two or more points are within this threshold of each other, the point with the lowest elevation is kept to make the TIN and the others are discarded.

Scan Tab Settings

SCAN BACKUP DIRECTORY

Enter the path to the location of the backup scanning (*.scan and .sc2) files. These files are recorded while a scan is taking place and are automatically removed when the scan successfully completes when using the HDS2500. The data from a *.scan and *.sc2 file may be recovered by importing it into the **Navigator**.

HDS4500/HDS6000 SCAN DIRECTORY

Enter the path to the location of the [HDS4500](#) and HDS6000 scanning (*.zfs) files. These files are recorded while a scan is taking place and are used for importing of HDS4500 and HDS6000 scans into the *Cyclone* database.

HDS3000/SCANSTATION/SCANSTATION 2: MAX # POINTS/SUB-SCAN

Enter a value for the maximum number of points allowed in any single sub-scan. Each time this limit is reached during scanning, a new sub-scan is created. This preference also applies to ScanStation and ScanStation 2 scans.

SCANSTATION 2: MAX # POINTS AUTO-IMPORTED

Enter a value for the maximum number of points for importing ScanStation 2 scans. Use the **Re-Import Scans** command to import the unimported points.

HDS4500/HDS6000: MAX # POINTS AUTO-IMPORTED

Enter a value for the maximum number of points for importing [HDS4500](#) and HDS6000 scans. For the HDS4500, when the value is **0**, a placeholder Scan is created but the scan is not imported to save time in the field. Use the **Re-Import Scans** command to import the unimported points. For the HDS6000, when the value is **0**, a placeholder Scan is created but

the scan is not downloaded from the scanner to save time in the field. Use the **Download HDS6000 Scans** command to download the missing file.

HDS4500/HDS6000: KEEP AUTO-IMPORTED ZFS FILE

Select whether or not the HDS4500 or HDS6000 ZFS file will be kept after each scan. If set to **Yes**, only the ZFS files for scans that are completely imported (that is, those with fewer points than the **HDS4500/HDS6000: Max # Points Auto-Imported** preference) will be deleted. Otherwise, all ZFS files produced by scanning will be kept on disk.

HDS6000: LEVEL TOLERANCE

Enter a value for the maximum angle that the scanner can be from perfectly level, but still be considered to be “level” by Scan Control workflows (e.g., Field **Setup**, **Traverse**) that require the scanner to be level.

HDS6000: TILT: MAX DRIFT

Enter a value for the maximum tilt change that the scanner can undergo, from the start of the first scan in the ScanWorld, before the scanner must be re-leveled. This is to ensure that scans in the same ScanWorld are in a consistent coordinate system.

SIMULATED SCANNER SPHERICAL NOISE

Enter a value for the standard deviation of noise applied to each simulated sample.

SIMULATED SCANNER MAXIMUM RANGE

Enter a value for the furthest sample that the simulated scanner will take.

SPHERE TARGET DIAMETER

Enter a value for the diameter expected when acquiring sphere targets in the **Scan Control** window.

SPHERE TARGET SCAN DIMENSIONS

Select **Standard**, **Medium**, or **Low** resolution target scan density for fine scans used when acquiring sphere targets in the **Scan Control** window.

RESECTION MAX ERROR

Enter a value for the maximum error allowed for a valid resection. This affects only the content of the **Resection Report**.

STAKEOUT: HIDE PREVIOUS SCAN

Select whether or not to hide the previous coarse scan acquired through the **Stakeout** panel in the **Scan Control** window. Hiding the previous coarse scan can help improve visualization when acquiring several successive coarse scans during the stakeout workflow.

Point Cloud Tab Settings

WORK DIRECTORY

Enter the path to the location for temporary *Cyclone* files.

INTERACTIVE: TARGET FRAME RATE

Specify the desired frame rate to be maintained while manipulating the viewpoint.

LOAD: MAX POINTS (MILLIONS)

Set the maximum number of points that can be loaded at one time.

DISPLAY: MAX POINTS (MILLIONS)

Set the maximum number of points that can be drawn in a single view at one time.

DISPLAY: SPACING LIMIT

Enter an average distance between any two points displayed.

COMPUTATION: MAX POINTS (MILLIONS)

Enter a value for the maximum number of points that can be used in any computation.

COMPUTATION: SPACING LIMITS

Enter a value for the minimum average spacing between points used in various computations.

COMPUTATION: CREATE TIN: MAX POINTS (MILLIONS)

Enter a value for the maximum number of points used to create a TIN.

SEGMENTATION: PREVIEW MAX POINT COUNT (THOUSANDS)

Enter a value for the maximum number of points that will be used to indicate the points that will be segmented by various operations, including [Segment Cloud: Cut by Intensity](#), [Segment Cloud: Cut Near Ref Plane](#), and [Measure: Interfering Points](#).

POINT CACHE: MAXIMUM CACHE SIZE (MB)

Enter a value for the maximum size of physical memory used to cache points.

IMPORT: COMPUTE MISSING NORMAL VECTORS

Select YES to compute missing normal vectors when importing point clouds. PTX data contains normals, so this setting has no effect. PTS data does not contain normals, so imported PTS data with this preference turned OFF cannot be used for steel fitting, cloud registration, and region growing smooth surfaces.

Initialization Tab Settings**CONTROLSPACE: COLOR MAP MODE**

Select **Colors from Scanner or Intensity Map** to set the color map used in subsequently-launched ControlSpaces. Select **Custom** to keep the current color map when launching an existing ControlSpace. (This may be overridden by any global color mapping.)

CONTROLSPACE: COLOR MAP SCHEME

Select **Multi-Hue/Rainbow** or **Grayscale** to set the color scheme used in subsequently-launched ControlSpaces. (This may be overridden by any global color mapping.)

NEW MODELSpace VIEW: COLOR MAP MODE

Select **Colors from Scanner** or **Intensity Map** to set the color map used in subsequently-created ModelSpace Views. (This may be overridden by any global color mapping.)

NEW MODELSpace VIEW: COLOR MAP SCHEME

Select **Multi-Hue/Rainbow** or **Grayscale** to set the color scheme used in subsequently-created ModelSpace Views. (This may be overridden by any global color mapping.)

NEW MODELSpace VIEW: GLOBAL COLOR MAP

Select **Enabled** to initially use global color map settings when creating a new ModelSpace View.

NEW MODELSpace VIEW: GLOBAL COLOR MAP MODE

Select **Colors from Scanner**, **Color Map**, **Elevation Map**, or **Single Color** to be applied when the global color map is enabled in ModelSpace Views created thereafter.

NEW MODELSpace VIEW: GLOBAL COLOR MAP SCHEME

Select **Multi-Hue/Rainbow**, **Grayscale**, a topography-based palette, or a single scaled color with which to display the intensity or elevation map (for the corresponding Mode) when the global color map is enabled in ModelSpace Views created thereafter. If the **Global Color Map Mode** is **Colors from Scanner**, this scheme is used when the cloud/mesh lacks per-point color.

NEW MODELSpace VIEW: GLOBAL INTENSITY MAP RANGE

Enter an intensity range for the color map used when the global color map is enabled in ModelSpace Views created thereafter.

NEW MODELSpace VIEW: GLOBAL ELEVATION MAP START

Enter the start elevation for the elevation map used when the global color map is enabled in ModelSpace Views created thereafter.

NEW MODELSpace VIEW: GLOBAL ELEVATION MAP DELTA

Enter the difference in elevation represented color in the elevation map used when the global color map is enabled in ModelSpace Views created thereafter.

NEW MODELSpace VIEW: GLOBAL COLOR MAP: NUMBER OF COLORS

Enter the total number of colors used when the global color map is enabled in ModelSpace Views created thereafter.

NEW MODELSpace VIEW: GLOBAL COLOR MAP: REPEAT COLORS OUTSIDE RANGE

Select **Yes** to use a cyclic color scheme when the global color map is enabled in ModelSpace Views created thereafter.

NEW MODELSpace VIEW: GLOBAL COLOR MAP GAMMA CORRECTION

Enter a value for the gamma correction used when the global color map is enabled in ModelSpace Views created thereafter.

NEW MODELSpace VIEW: POINT THICKNESS

Specify the visual width and height (in pixels) of each point in an unselected point cloud. Thicker points may be easier to see.

NEW MODELSpace VIEW: POINT THICKNESS (HIGHLIGHTED)

Specify the visual width and height (in pixels) of each point in a selected point cloud. Thicker points may be easier to see and pick.

NEW MODELSpace: EXCLUDE POINT CLOUDS

Select **Yes** to not populate a new ModelSpace with the existing point clouds available to the ScanWorld when creating a ModelSpace from that ScanWorld.

NEW MODELSpace: EXCLUDE TARGET OBJECTS

Select **Yes** to not populate a new ModelSpace with the existing HDS Targets/targets available to the ScanWorld when creating a ModelSpace from that ScanWorld.

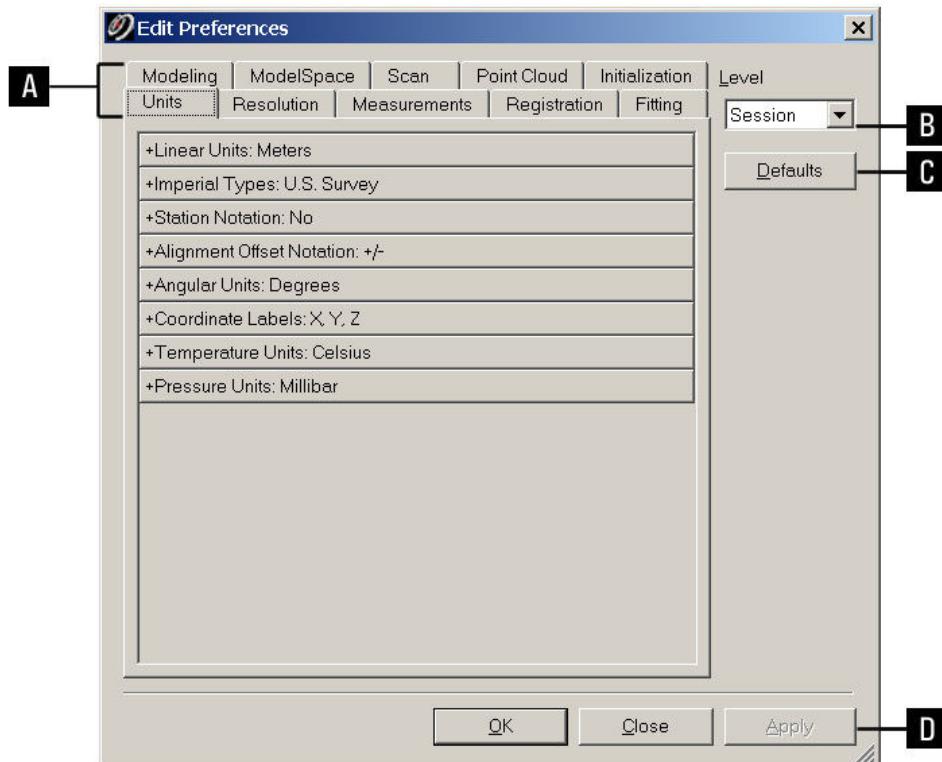
NEW MODELSpace: EXCLUDE TARGET SCANS

Select **Yes** to exclude the existing HDS Target/sphere target point clouds available to the ScanWorld when creating a ModelSpace from that ScanWorld.

NEW MODELSpace: MAXIMUM # POINTS BEFORE EXCLUDING

Enter a value for the total number of points (in a ScanWorld) above which all scans are excluded when creating a ModelSpace. Enter a value of *0* (default) to never exclude based on the number of points. **Create ModelSpace: Exclude Point Clouds** must be set to **Yes** for this option to take effect.

Edit Preferences Dialog



- A. Click a tab to access tab settings. See above for descriptions of tab contents.
- B. Select a level for changes.
- C. Reset the selected preference tab to the default settings.
- D. Apply changes and continue working in the Edit Preferences dialog.

To edit preferences:

1. From the **Edit** menu, select **Preferences....** The **Edit Preferences** dialog appears.
2. Select the level at which your changes take effect.
 - To select a level, select **Default** or **Session** from the **Level** drop-down list (see *Level Options* following).
3. Select the tab that contains the settings that you want to change.
4. Change the appropriate settings, and then click **OK**.

Print...

Window: ModelSpace, Image Viewer, Scan Control

Menu: File

Action: Opens dialog

Usage: The **Print...** command launches the standard Windows **Print** dialog to print the contents of the current window.

Probe

Window: Scan Control

Menu: Scanner Control

Action: Executes command

Usage: The **Probe** command samples the range at the targeting cross-hair.

Probe (Tab)

HZ

Displays horizontal angle of the probe direction.

V

Displays height of probe point (relative to the scanner's horizontal plane).

- Hz and V can be set by entering values in the fields or by clicking ctrl+left in target mode.

RANGE

Displays distance from scanner to nearest object aligned to the probe point.

PROBE BUTTON

Click to measure Range at the Hz and V angles.

- To set the range manually, enter a value into the **Range** field.

Project

Window: Navigator

Menu: Create

Action: Executes command

Usage: The **Project** command creates a new project folder in the selected location.

- Project folders can be created in a database, within another project folder, or in a ScanWorld.

To create a new project:

1. In the **Navigator** window, select the folder into which you want to add a new project.
2. From the **Create** menu, select **Project**. The project is created.

Project Polyline to TIN

Window: ModelSpace

Menu: Tools | Mesh | Polylines

Tools | Mesh | Breaklines

Action: Executes command

Usage: The **Project Polyline to TIN** command wraps a polyline onto a TIN object, creating edges in the mesh based on the projection of the polyline vertically onto the mesh (although the polyline itself is unchanged); the edges are then "stretched" vertically to conform to the polyline's shape.

- If you select the **Project Polyline to TIN** command from the **Breaklines** submenu, the polyline is converted to a breakline on the mesh. If you select this command from the **Polylines** submenu, the polyline will only be projected onto the mesh, and the resulting line will be merely a geometric object.
- Polylines are projected along the "up" direction of the current coordinate system
- The mesh onto which you are projecting a polyline must be qualified to be a TIN. See the [Verify TIN](#) command.

To project a polyline onto a TIN:

1. Multi-select a polyline and the TIN mesh onto which the polyline will be projected.
2. From the **Tools** menu, point to **Mesh**, point to **Polylines**, and then select **Project Polyline to TIN**. The polyline is projected onto the mesh.

To project a polyline onto a TIN and create a breakline within the mesh:

1. Multi-select a polyline and the TIN mesh onto which the polyline will be projected as a breakline.
2. From the **Tools** menu, point to **Mesh**, point to **Breaklines**, and then select **Project Polyline to TIN**. The polyline is duplicated, then projected onto the TIN. Its vertices are incorporated into the mesh as a breakline.

Project Setup

Window: Scan Control

Menu: Window

Action: Expands Tab

Usage: The **Project Setup** command expands the **Project Setup** tab in the Scanner Control Panel.

- You must select a project before beginning a scan or getting an image.
- The **Select a Project** dialog is displayed automatically when you open the **Scan Control** window. Once you have selected a project, *Cyclone* automatically assigns ScanWorlds as you create new scans, and you only need to use **Project Setup** if you wish to manually select or create different projects and ScanWorlds.

Project Setup tab Options

PROJECT

Displays the current project. Click the ... button to select a different project or to create a new project.

SCANWORLD

Displays the current ScanWorld. Click the ... button to select a different ScanWorld or to create a new ScanWorld.

NEXT SCAN

Displays the name of the next scan in the current project and ScanWorld.

NEW POSITION

Click to prepare a new ScanWorld for the next scan.

OPEN VIEWER

Click to open a new ModelSpace viewer for the incoming scan data.

Note: This ModelSpace is created in the ScanWorld chosen as your destination using the **Select ScanWorld...** command.

Note: The point cloud being acquired is visible in this ModelSpace viewer, but is not selectable until the scan operation is completed or canceled.

Publish Site Map...

Window: ModelSpace

Menu: File

Action: Opens dialog

Usage: The **Publish Site Map...** command creates an HTML page with a snapshot of the current viewpoint, with hot-spots at the scanner positions. The command also generates datasets for use by Leica TruView.

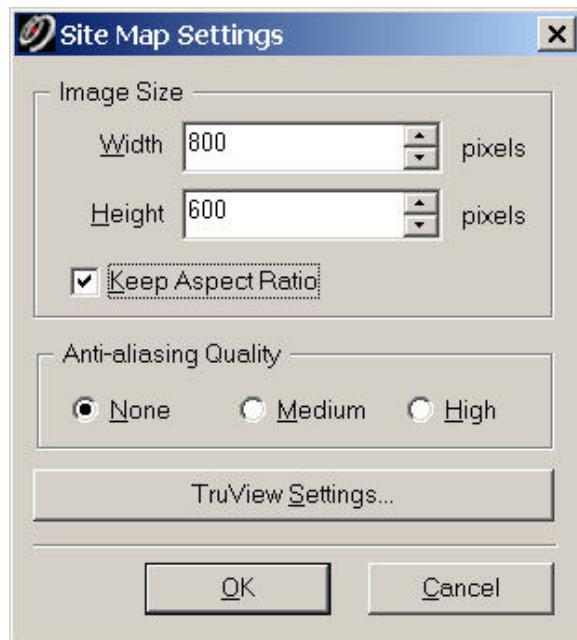
- Also see the *Using Cyclone Publisher* document.

To publish a site map:

1. Set the viewpoint in the ModelSpace viewer. This viewpoint will be used for the site map image.

- Any scanner locations contained in this viewpoint will be used to generate TruView datasets.
 - If the viewpoint does not contain any scanner locations, then no TruView datasets will be generated, although the site map will still be generated.
2. Adjust the visualization and graphics options of the ModelSpace View. The site map image will use these settings.
 3. In the **File** menu, select **Publish Site Map....** The **Browse for Folder** dialog appears.
 4. In the **Browse for Folder** dialog, select the folder to contain the site map files (and TruView datasets). The **Select ScanWorlds** dialog appears next.
 5. In the **Select ScanWorlds** dialog, select the ScanWorlds for which TruView datasets will be generated. Selecting a ScanWorld will also generate TruView datasets for its subordinate ScanWorlds.
 - One TruView dataset consists of only the scans specific to the original ScanWorld. Registered ScanWorlds have no scans of their own.
 - The TruView dataset uses the original ScanWorld's default point clouds. Use the **Set ScanWorld Default Clouds** or **Set as Default Scan** commands in the original ScanWorld to modify its default point clouds.
 - The coordinate system and "up" direction used in the site map are applied to each TruView.
 6. In the **Site Map Settings** dialog, select the options to use for the site map.
 7. Click the **TruView Settings...** button to customize how the TruView datasets are generated.
 8. Click the **OK** button. The site map is created. Any TruView datasets that were generated are contained in sub-folders.

Site Map Settings Dialog



A. Width, Height: The dimensions of the site map image.

B. Keep Aspect Ratio: When checked, changing either the **Width** or the **Height** adjusts the other dimension proportionately.

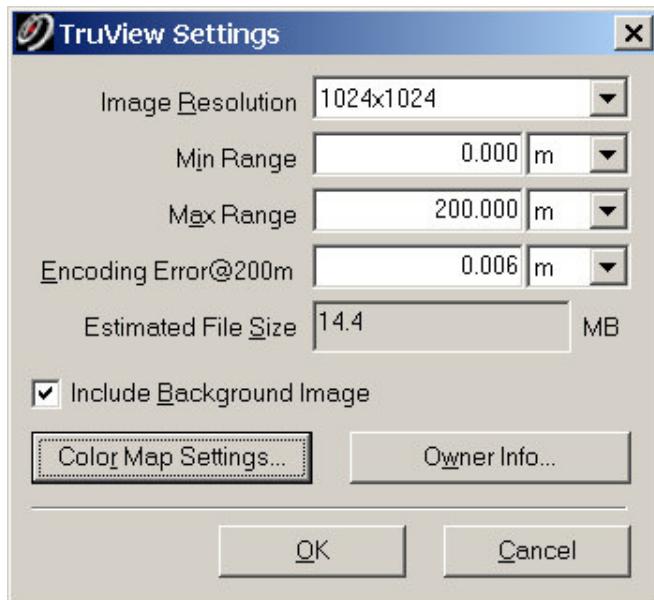
C. Anti-aliasing Quality: Select the level of anti-aliasing in the site map image. A higher level helps smoothen jagged edges in the image.

D. TruView Settings...: Click to invoke the **TruView Settings** dialog.

E. OK: Click to accept the settings and publish the site map and TruView datasets.

F. Cancel: Click to cancel this operation.

TruView Settings Dialog



A. Image Resolution: The maximum resolution used in each of the six images generated for a TruView dataset. The six images form a complete cube. A higher resolution may improve image quality when zoomed in using TruView, but it may also result in larger files.

B. Min Range: Points closer to the scanner than this value are not included in the TruView dataset.

C. Max Range: Points farther from the scanner than this value are not included in the TruView dataset. Use this to avoid including points that are beyond the recommended range of the scanner, or to limit the visualization to a local region.

D. Encoding Error@200m: The desired precision of the 3D point at each TruView pixel. A higher precision may result in better measurements using TruView, but it may also result in larger files.

E. Estimated File Size: The estimated size of the files in a typical TruView dataset generated with these settings. The estimate updates when the settings are changed. Refer to this to gauge

the impact on disk space and/or speed of downloading the TruView datasets from a network server.

F. Include Background Image: When checked, the Multi-Image from a HDS3000, ScanStation, or ScanStation 2 ScanWorld is used as a background image wherever points from the scan are not available. This can help improve the visualization in TruView.

G. Color Map Settings...: Click to invoke the **Color Map Settings** dialog.

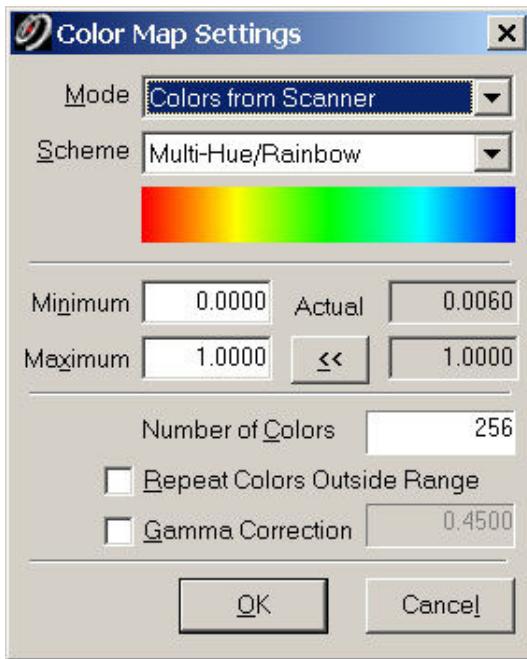
H. Owner Info...: Click to invoke the **TruView Owner Information** dialog.

I. OK: Click to accept the TruView settings and return to the **Site Map Settings** dialog.

J. Cancel: Click to return to the **Site Map Settings** dialog without accepting any changes to the TruView settings.

Color Map Settings Dialog

The **Color Map Settings** dialog is used to specify how point clouds will be displayed when generating a TruView dataset. See [Edit Color Map](#) in the *Commands* chapter for details. Some options are not available when generating a TruView dataset.

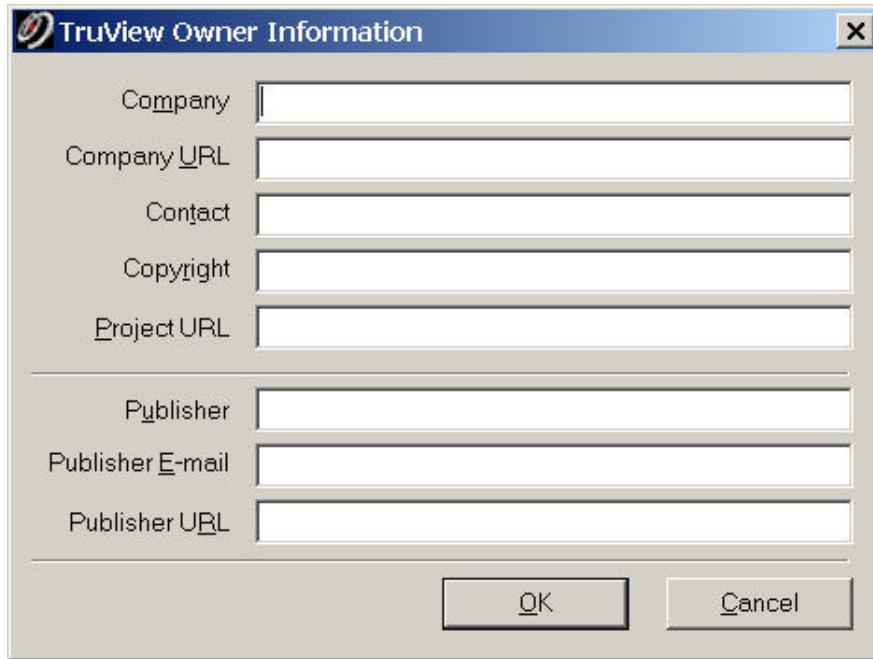


A. OK: Click to accept the TruView color map settings and return to the **TruView Settings** dialog.

B. Cancel: Click to return to the **TruView Settings** dialog without accepting any changes to the TruView color map settings.

TruView Owner Information Dialog

The **TruView Owner Information** dialog is used to supply information about the owner of the TruView datasets to be generated. Information about the publisher of the TruView datasets is also entered in this dialog. The TruView datasets include this information.



- A. Company:** Enter the name of the company that owns the TruView dataset.
- B. Company URL:** Enter the URL (for example, "<http://hds.leica-geosystems.com>") of the company that owns the TruView dataset.
- C. Contact:** Enter the main contact information for the company that owns the TruView dataset.
- D. Copyright:** Enter the copyright notice for the TruView dataset.
- E. Project URL:** Enter the URL of the project to which the TruView dataset belongs.
- F. Publisher:** Enter the name of the company that is generating the TruView dataset.
- G. Publisher E-mail:** Enter the e-mail address of the main contact of the company that is generating the TruView dataset.
- H. Publisher URL:** Enter the URL of the company that is generating the TruView dataset.
- I. OK:** Click to accept the TruView owner and publisher information and return to the **TruView Settings** dialog.

J. Cancel: Click to return to the **TruView Settings** dialog without accepting any changes to the TruView owner and publisher information.

Put at Angle...

Window: ModelSpace

Menu: Edit Object | Align

Action: Opens dialog

Usage: The **Put at Angle...** dialog is used to align one or more selected objects at a specific angle to a reference object.

To align two objects at an angle:

1. Multi-select the objects you want to align.
 - The last object selected is the reference object (the one to which the others align).
2. From the **Edit Object** menu, point to **Align**, and then select **Put at Angle....** The **Put at Angle** dialog appears.
3. Enter the angle you want, and then click **OK**. The selected objects are rotated so that their axes form the specified angle with the axis of the reference object.

Put at Distance...

Window: ModelSpace

Menu: Edit Object | Align

Action: Opens dialog

Usage: The **Put at Distance** dialog is used to set one object at a specific distance from a reference object.

To set two objects at a specified distance:

1. Multi-select the two objects you want to align.
 - The last object selected is the reference object (the one from which the other moves).
2. From the **Edit Object** menu, point to **Align**, and then select **Put at Distance....** The **Put at Distance** dialog appears, displaying the current distance between the objects in the **Distance** field.
3. Enter a value in the **Distance** field, and then click **OK**. The first object selected is moved to the specified distance from the reference object.
 - The direction of translation depends on the types of the objects being used.

QuickScan (command)

Window: Scan Control

Menu: Scanner Control

Action: Executes command

Usage: The **QuickScan** command is used to quickly initiate a scan when your field of view settings do not need to be precise and your other scan control settings are already satisfactory.

See also, the [Station Data tab](#) entry in the *Commands* chapter.

To QuickScan:

1. In the **Scan Control Window Field-of-View** tab, click the **QuickScan** button. The **Aim Scanner** dialog appears.
 - The scanner motor disengages to allow you to rotate the scanner on its mount.
2. Rotate the scanner on its mount and aim it toward the left extent of the region you want to scan.
 - To accept the current left extent, click the **Skip** button to proceed to step 4.
 - To cancel QuickScan without changing any scanner settings, click the **Cancel** button.
3. Press the multi-purpose button on the scanner. The **Aim Scanner** dialog prompts you to aim the scanner toward the right extent of the region you want to scan.
4. Rotate the scanner on its mount and aim it toward the right extent of the region you want to scan.
 - To accept the scanner's current position as the right extent, click the **Skip** button to set the right extent.
 - Any dependent values (e.g., number of points) also update.
5. Press the multi-purpose button on the scanner.
 - The target region's right horizontal angle in the Scan tab updates.
 - Any dependent values (e.g., number of points) also update.
 - To cancel the QuickScan without changing any scanner settings, click the **Cancel** button. If any angles were changed, the values revert back to what they were before you clicked the **QuickScan** button.

Note: If you have previously set the scanner orientation, all horizontal angles are relative to the current orientation. See also, [Station Data](#).

Raise Active Cutplane

Window: ModelSpace

Menu: Tools | Cutplane

Action: Executes command

Usage: The **Raise Active Cutplane** command raises the Cutplane by the amount specified in the **Offset** command (see the *Add/Edit Cutplanes* entry in this chapter).

Raise/Lower

Window: ModelSpace

Menu: Tools | Reference Plane

Action: Toggles function ON/OFF

Usage: The **Raise/Lower** command toggles the raise/lower function in **Drawing** mode, which allows you to interactively raise or lower the active Reference Plane without changing its orientation.

To raise/lower the Reference Plane:

1. From the **Tools** menu, point to **Reference Plane**, and then select **Raise/Lower** (or click the **Raise/Lower** icon). You are now in **Drawing** mode. The cursor changes to the **Drawing Tool** cursor, and the raise/lower function is ON.
2. In the ModelSpace, drag the **Drawing Tool** cursor up/down to raise/lower the active Reference Plane.
3. Release the mouse button. The Reference Plane is raised/lowered.

Randomize Colors of Visible (Selected) Objects

Window: ModelSpace

Menu: Edit Object | Appearance

Action: Executes command

Usage: The **Randomize Colors of Visible Objects** command assigns random colors to all visible or selected objects.

- If any objects are selected, this command changes from **Visible** to **Selected** objects and affects only those selected objects.
- This command is useful when there is a large number of similarly-colored objects that you need to quickly distinguish from each other.

Rectangle Fence Mode

Window: ModelSpace

Menu: Edit | Modes

Action: Executes command

Usage: The **Rectangle Fence Mode** command enters **Rectangle Fence** mode and selects the **Rectangle Fence** mode cursor.

- For more information on working with fences, see the [Using a Fence](#) entry in the *Modeling* chapter.

To draw a rectangular fence:

1. From the **Edit** menu, point to **Modes**, and then select **Rectangle Fence Mode**. The **Rectangular Fence** cursor appears.
2. Drag the cursor around the area that you want to fence.
 - The newly drawn fence can adjusted by dragging its sides or vertices with the **Rectangular Fence** cursor.
 - To clear the newly drawn fence, select **Clear** from the **Fence** submenu.

Rectangular

Window: Scan Control

Menu: Scanner Control | Set Fence

Action: Executes command

Usage: Pointing to **Rectangular** displays submenu commands used to select a preset rectangular fence size.

The following preset panoramic fences are available from the **Rectangular** submenu:

40 x 40, 60 x 60, 80 x 80

Redlining

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Redlining** displays submenu commands used to create new redlines and manage saved redlines.

- Redlines are used to create 2D lines and add comments to a ModelSpace.
- Redlining behaves identically to 2D drawing. However, drawing is done on the active Reference Plane, whereas Redlining is done on an imaginary plane that is perpendicular to and centered in the current ModelSpace viewpoint.
- Redlines are part of the ModelSpace View. You can only view or edit redlines that were created in the current View.

The following commands and dialogs are available from the **Redlining** submenu:

[Edit Redlines](#), [Create Redlines](#)

Redo

Window: ModelSpace, Navigator, Registration, Scan Control, Image Viewer
Menu: Edit
Action: Executes command
Usage: The **Redo** command reverses the effects of the most recent **Undo** command.

- A series of **Undo** commands can be reversed using a series of **Redo** commands unless a new action is performed, at which time all changes that have not been redone are discarded.
- Any action that can be undone with the **Undo** command (see the *Undo* entry in this chapter), can be redone with the **Redo** operation.

Reduce Point Cloud...

Window: ModelSpace
Menu: Edit Object
Action: Opens dialog
Usage: The **Reduce Point Cloud** dialog is used to reduce the number of points in a cloud.

Reduce Point Cloud Dialog Fields

HORIZONTAL SAMPLING

Enter the ratio of original-to-subsample horizontal points.

VERTICAL SAMPLING

Enter the ratio of original-to-subsample vertical points.

POINT COUNT

Enter a specific value for the number of points you want in the resulting point cloud.

PERCENTAGE OF ORIGINAL

Enter a percentage by which to reduce the number of points.

ORIGINAL POINT COUNT

Displays the number of points in the original cloud.

To reduce the number of points in a cloud:

1. Select the point cloud that you want to reduce.
2. From the **Edit Object** menu, select **Reduce Point Cloud...**. The **Reduce Point Cloud** dialog appears.
3. Select the sample rate by horizontal and vertical ratio, point count, or percentage of original, and then click **OK**. The point cloud is replaced by a subsampled version of the cloud.

Reducer Connectors

Window: ModelSpace

Menu: Tools | Piping

Action: Executes command

Usage: The **Reducer Connectors** command connects two roughly parallel selected pipes.

To connect parallel pipes:

1. Pick a point on the first pipe near the end you want connected.
2. Multi-select the second pipe near the end that you want connected.
3. From the **Tools** menu, point to **Piping**, and then select **Reducer Connectors**. The pipes are connected with a reducer.

Reference Plane

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Reference Plane** displays submenu commands used to display, hide, and manipulate the active Reference Plane.

- The Reference Plane is an infinite two-dimensional plane used for creating shapes that are otherwise unavailable in *Cyclone*, or for creating shapes where object fitting or insertion are impractical. It is also used as a reference for several operations such as measuring a mesh volume or orienting the Cutplane.

The following commands and dialogs are available from the **Reference Plane** submenu:

[Add/Edit Reference Planes...](#) [Show Active Plane](#), [Align View to Active Plane](#), [Edit Active Plane...](#), [Set to Viewpoint](#), [Set to UCS](#), [Set to X-Y Plane](#), [Set to X-Z Plane](#), [Set to Y-Z Plane](#), [Set on Object](#), [Set Plane Origin at Pick Point](#), [Pan](#), [Rotate](#), [Tilt](#), [Raise/Lower](#)

- For information on 2D drawing procedures, see the [Drawing](#) entry in the *Modeling* chapter.

Region Grow

Window: ModelSpace

Menu: Create Object

Action: Displays submenu

Usage: The **Region Grow** submenu contains commands that are used to fit a patch, cylinder, or sphere to a shape within a point cloud without having to first segment the points. It also contains the **Smooth Surfaces** command, which is used to segment a selected smooth surface in a point cloud, and the **Pipe Run** command, which is used

to grow a series of pipes connected by elbows.

- When region growing a patch, cylinder, or sphere, the points on the surface representing the selected object are replaced with the fitted geometry.
- For the **Smooth Surfaces** command, the points representing the smooth surface are merely segmented from the point cloud; no object is inserted.
- To set default region growing parameters (except for Smooth Surfaces), see the [Region Growing](#) section of the *Preferences* entry in this chapter.
- Region growing can make use of parts tables to fit standard objects. To set and enable parts tables, see the *Object Preferences*, *Use Parts Table*, and *Show Table Matches* entries in this chapter.

The following commands are available from the **Region Grow** submenu:

Patch..., **Cylinder...**, **Sphere...**, **Smooth Surface...**, **Pipe Run...**, and **Edit Parameters...**

To fit a patch using region growing:

1. Select a point in the center of the region of the cloud that make up the interior of the Patch you want to segment and fit.
- A patch can often be fit with a single pick point. However, for best results, pick three or four points (no more than two collinear) as far apart from each other as possible near (but not on) the edges of the object you want to fit.
- If region growing is not performing adequately, pick three or more points (no more than two collinear).
- When region growing, multiple clouds may be selected as long as there is at least one pick point.
2. From the **Create Object** menu, point to **Region Grow** and select **Patch**. The **Region Grow Patch** dialog appears.
3. Use the sliders to adjust parameters.
 - **Region Thickness** defines a surface thickness outside of which points are excluded from the fit calculation.
 - **Maximum Gap to Span** defines a maximum distance between any two adjacent points included in the fit calculation.
 - To restart the fit with the current settings, click **Restart**.
4. Click **OK**.
5. If the **Fitting: Adjust Patch Boundary** setting in the **Edit Preferences** dialog is set to **Yes**, the **Adjust Patch Boundary** dialog appears. Use the slider to adjust the patch's polygonal boundary. The closer the slider moves towards **Tight Boundary**, the more rigidly the boundary of the new patch adheres to the points on the edge of the point cloud. In some cases this adjustment may necessitate the creation of more than one patch.

To fit a cylinder or sphere using region growing:

1. Select a point in the center of the region of the cloud that make up the interior of the object you want to segment and fit.
- For best results, pick three or four points (no more than two collinear) as far apart from each other as possible near (but not on) the edges of the object you want to fit.
- When region growing, multiple clouds may be selected as long as there is at least one pick point.

2. If region growing is not performing adequately, pick additional points as follows:
 - **Cylinder:** Pick six or more points (no more than three collinear).
 - **Sphere:** Pick four or more points (no more than three coplanar).
3. From the **Create Object** menu, point to **Region Grow**, and select the type of object you want to fit. The **Region Growing Sphere/Cylinder** dialog appears.
4. Use the sliders to adjust parameters for the object being fit and then click **OK**.
 - **Region Thickness** defines a surface thickness outside of which points are excluded from the fit calculation.
 - **Maximum Gap to Span** defines a maximum distance between any two adjacent points included in the fit calculation.
 - **Details** displays the fitting error.

To region grow a smooth surface:

1. Select a point on the surface of the flat region of the point cloud that you want to segment.
 - Select a point near the center of the region.
 - When region growing, multiple clouds may be selected as long as there is at least one pick point.
2. From the **Create Object** menu, point to **Region Grow**, and then select **Smooth Surface....** The **Region Grow Smooth Surface** dialog appears.
3. Use the sliders to adjust the smooth surface parameters.
 - **Region Thickness** defines the smooth surface thickness outside of which points are not considered a part of the smooth surface.
 - **Maximum Gap to Span** defines the maximum distance that may exist between portions of the same smooth surface.
 - **Details** displays the number of points in the selected point cloud, along with the number of points conforming to the parameters set in this dialog.
4. Click **OK**. The point cloud is segmented.
 - When enabled, the Limit Box can improve performance by reducing the amount of points processed; only those visible points are used.

To region grow a pipe run:

1. Pick a point on the first pipe in the run.
2. Shift+Pick the other pipes in the run in any order, with one pick per pipe.
 - The pipes must have the same outside diameter.
3. From the **Create Object** menu, point to **Region Grow**, and then select **Pipe Run....** The **Region Grow Pipe Run** dialog appears.
4. Use the sliders to adjust the pipe run parameters.
 - **Region Thickness** defines the surface thickness outside of which points are excluded from the fit calculation.
 - **Maximum Gap to Span** defines a maximum distance between any two adjacent points included in the fit calculation.
 - **Angle Tolerance** is disabled.
 - **Region Size** defines a distance beyond the pick points at the ends of the pipe run. Points within this distance are used in fitting those end pipes. This can be used to control the extents of end pipes.

- **Details** displays the fitting status.
- 5. The **Pipe** menu in the dialog provides additional options.
- **Discard Last Pipe** removes the pipe grown last. This can be used when you pick a suboptimal point or the region grower performs a poor fit. The pipe picked last is shown in red.
- **Snap Bend Ratios**, when selected, uses the closest bend ratio in the set { 0.5xOD, 0.75xOD, 1.0xOD, 1.5xOD, 2xOD, 3xOD, 4xOD, 5xOD, ... } when fitting elbows.
- **Segment Points**, when selected, segments the points used in the region grow from the rest of the cloud when the **OK** button is clicked. Not segmenting the points may result in better performance.
- **Reverse Run** changes the order of the pipes in the pipe run. This can be used in conjunction with **Discard Last Pipe** to remove the pipe at either end of the pipe run.
- 6. Make modifications or corrections to the region growing. Some modifications require a **Restart** instead of a **Continue**.
- If the region grow algorithm fails to fit a pipe in the sequence, the **Details** reports which pipe was not found and suggests the next steps.
- To include additional pipes in the pipe run, Shift+Pick additional points on the pipes adjacent to either end of the pipe run already grown, then click **Continue**. The additional picks must be in sequence. That is, pick the next pipe connected to one end, then the next pipe in that direction, and so on.
- 7. Click **OK**. The pipe run is fit and added to the ModelSpace.

Register

Window: Registration

Menu: Registration

Action: Executes command

Usage: The **Register** command is the step in the registration process during which the optimal alignment transformations are computed for each ScanWorld in the Registration, such that the constraints are satisfied as closely as possible.

- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.

Registration

Window: Navigator

Menu: Create

Action: Executes command

Usage: The **Registration** command creates a new Registration in the selected location.

- A Registration can be created in a database or project.

- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.

To create a new Registration:

1. In the **Navigator** window, select the database or project for your new registration.
2. From the **Create** menu, select **Registration**. The Registration is created.

Registration (Tools)

Window: ModelSpace

Menu: Tools

Action: Displays submenu

Usage: Pointing to **Registration** displays submenu commands used to add, edit, and delete Registration labels, as well as to add objects manually to the ControlSpace for the current ScanWorld.

The following commands, dialogs, and submenus are available from the **Registration** submenu:

[Add/Edit Registration Label...](#), [Remove Registration Label](#), [Set Target Height](#), [Copy to ControlSpace](#), [Elevation Check](#)

Registration Diagnostics

Window: Registration

Menu: Registration

Action: Opens dialog

Usage: The **Registration Diagnostics** dialog shows registration status and statistics for the current registration.

Re-Import Scans

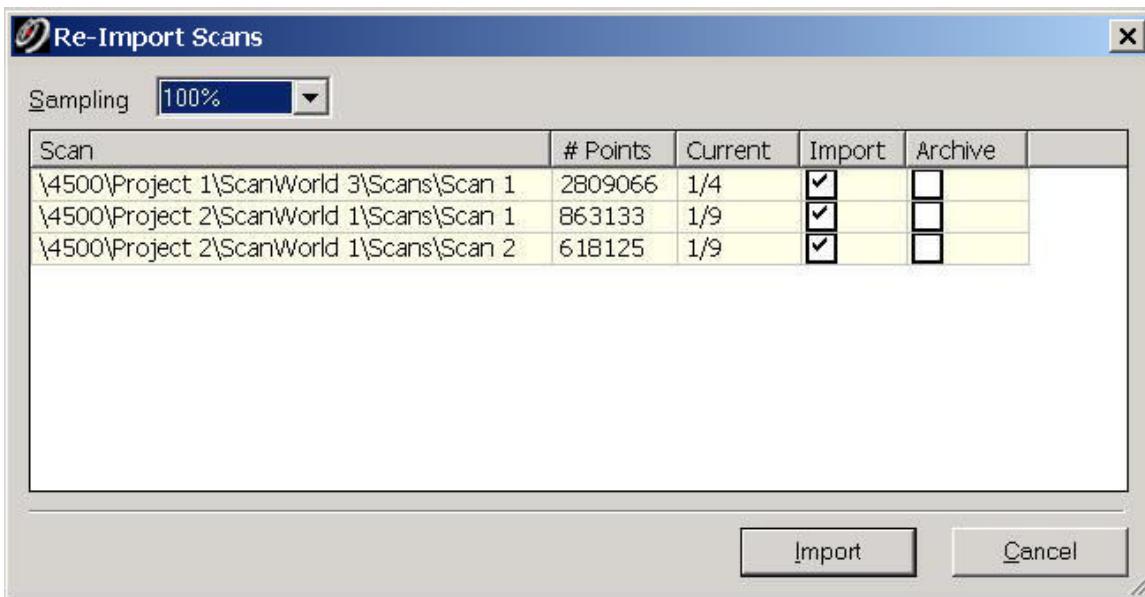
Window: Navigator

Menu: Tools

Action: Opens dialog

Usage: The **Re-Import Scans** dialog is used to import reduced or full HDS4500 and HDS6000 scans as point clouds in the database, replacing the corresponding cloud currently in the database. Only ScanStation 2 scans that were not automatically imported (as per the **ScanStation 2: Max # points Auto-Imported** preference) and HDS4500 and HDS6000 scans that were not automatically imported (as per the **HDS4500/HDS6000: Max # Points Auto-Imported** preference) are considered.

Re-Import Scans Dialog



Sampling Select the percent reduction in number of points to be imported relative to the total available number.

Scan Displays the path to the scan in the Navigator.

Points Displays the approximate number of points that will be imported.

Current Displays the current sampling of the scan.

Import Select this check box to import the points.

Archive Select this check box to move the corresponding SC2 or ZFS scan file to the **Archive** subfolder.

Import Imports (or archives) the selected scans.

Cancel Cancels the import.

To re-import ScanStation 2, HDS4500, and HDS6000 scans:

1. In the **Navigator** window, select the container under which ScanStation 2, HDS4500, and/or HDS6000 Scans eligible for import are located.
 - The container can be a database, Project, ScanWorld, Registration, or Scan.
 - You can select multiple containers.
2. In the **Navigator** window on the **Tools** menu, select **Re-Import Scans....**
 - The **Re-Import Scans** dialog displays.
3. To select a Scan for import, toggle ON the corresponding **Import** flag.
4. Select the **Sampling** level to be used when importing the Scans selected for import.
5. To remove a specific Scan from future consideration, toggle ON the **Archive** flag.
6. Click **Import**. The indicated **Import** and/or **Archive** actions are performed.

Remove Both Caps

Window: ModelSpace

Menu: Edit Object | End Caps

Action: Executes command

Usage: The **Remove Both Caps** command deletes both end caps from the selected object.

To remove both end caps:

1. Select the object from which you want to remove the caps.
2. From the **Edit Object** menu, point to **End Caps**, and then select **Remove Both Caps**. The caps are removed.

Remove Cap Closest to Pick

Window: ModelSpace

Menu: Edit Object | End Caps

Action: Executes command

Usage: The **Remove Cap Closest to Pick** command deletes the end cap nearest to the picked point on the selected object.

To remove the end cap closest to the pick:

1. Pick a point on the object near the cap that you want to remove.
2. From the **Edit Object** menu, point to **End Caps**, and then select **Remove Cap Closest to Pick**. The cap closest to the pick point is removed.

Remove from Group

Window: ModelSpace

Menu: Edit | Group

Action: Executes command

Usage: The **Remove from Group** command removes the picked object from an existing group.

To remove an object from a group:

1. Pick the group member you want to remove from the group.
 2. From the **Edit** menu, point to **Group**, and then select the **Remove from Group** command.
The picked object is removed from the group and placed back into the ModelSpace.
- If there are only two items in a group and one is removed, the single remaining item is also removed, and the group is destroyed (a group must have at least two items). Both objects are placed back into the ModelSpace.

Remove Inside Fence

Window: ModelSpace
Menu: Selection | Point Cloud Sub-Selection
Action: Executes command
Usage: The **Remove Inside Fence** command removes any points inside a fence from the sub-selection set.

- For more information, see the [Advanced Segmentation](#) entry in the *Modeling* chapter.

Remove Outside Fence

Window: ModelSpace
Menu: Selection | Point Cloud Sub-Selection
Action: Executes command
Usage: The **Remove Outside Fence** command removes any points outside a fence from the sub-selection set..

- For more information, see the [Advanced Segmentation](#) entry in the *Modeling* chapter.

Remove Registration Label

Window: ModelSpace
Menu: Tools | Registration
Action: Executes command
Usage: The **Remove Registration Label** command deletes the registration label from the selected object(s).
Note: Removing a registration label from an object does not remove that object from the ControlSpace.

To remove a registration label:

1. Select the object(s) from which you want to remove a registration label.
2. From the **Tools** menu, point to **Registration**, then select **Remove Registration Label**. The registration label is deleted.

Rename

Window: Navigator
Menu: Edit
Action: Executes command
Usage: The **Rename** command allows you to edit the name of the selected object.

To rename an object in the Navigator window:

1. Select the object you want to rename.
2. From the **Edit** menu, select **Rename**. The object's name is selected and ready for editing.
3. Type a new name, and then press **ENTER**.

Reset Active Alignment

Window: ModelSpace

Menu: Tools | Alignment and Section

Action: Executes command

Usage: The **Reset Active Alignment** command clears the active alignment.

- Station-notation values are only supported when there is an active alignment.
- Existing measurements are not affected.

Reset Alignment

Window: Registration

Menu: Cloud Constraint

Action: Executes command

Usage: The **Reset Alignment** command resets the selected cloud constraints to the *not aligned* state and resets its current alignment transform to that specified by the initial cloud alignment.

- For more information on the cloud registration process, see the [Cloud Registration](#) entry in the *Registration* chapter.

Reset ScanWorld Coordinate System

Window: ModelSpace

Menu: View | Coordinate System

Action: Executes command

Usage: The **Reset ScanWorld Coordinate System** command restores the original coordinate system and default "up" direction of the current ModelSpace's parent ScanWorld.

Reset Target

Window: Scan Control

Menu: Scanner Control

Action: Executes command

Usage: The **Reset Target** command deletes any existing targeting fence and replaces it

with a centered rectangular 60 x 60 fence.

Resolution (HDS2500)

Window: Scan Control

Menu: Scanner Control

Action: Executes command

Usage: Pointing to **Resolution** displays submenu commands used to set a standard interval between data points to be captured in the next scan operation.

The following intervals are available from the **Resolution** submenu:

1x1 mm, 1x1 cm, 1x1 m, 1x1 in, 1x1 ft, 1x1 yd

- For more information on the scanning process, see the [Scanning](#) entry in the *Getting Started* chapter.

Resolution (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)

Window: Scan Control

Menu: Window

Action: Expands Tab

Usage: The **Resolution** command expands the **Resolution** tab in the Scanner Control Panel.

Resolution Tab Options

RANGE

Enter a representative distance from the scanner location to the scene to be scanned.

PROBE BUTTON

Click to measure the Range in the direction of the targeting cross-hair.

SAMPLE SPACING

Displays spacing between sample points for the current Range and selected number of points.

PRESETS

Select a preset sample spacing from the dropdown. If sample spacing does not match the presets, then it displays "Custom". Sample spacing can change by either direct editing or changes in the **Number of Points** field.

NUMBER OF POINTS

Displays total number of points in the Horizontal and Vertical directions, based on the selected **Range** and **Sample Spacing**.

Note: The dependencies are: 1. Changing **Range** changes **Number of Points** based on **Sample Spacing**. 2. Changing **Sample Spacing** changes **Number of Points** based on **Range**. 3. Changing **Number of Points** changes **Sample Spacing** based on **Range**.

Resolution (HDS4500)

Window: Scan Control

Menu: Scanner Control

Action: Executes command

Usage: Pointing to **Resolution** displays submenu commands used to set a standard interval between data points to be captured in the next scan operation..

The following intervals are available from the **Resolution** submenu:

Highest – Sample at every 0.018 x 0.018 degree (20,000 samples in a 360 degree circle)

High – Sample at every 0.036 x 0.036 degree (10,000 samples in a 360 degree circle)

Medium – Sample at every 0.072 x 0.072 degree (5,000 samples in a 360 degree circle)

Low (Preview) – Sample at every 0.288 x 0.288 degree (1,250 samples in a 360 degree circle)

Resolution (HDS6000)

Window: Scan Control

Menu: Scanner Control

Action: Executes command

Usage: Pointing to **Resolution** displays submenu commands used to set a standard interval between data points to be captured in the next scan operation..

The following intervals are available from the **Resolution** submenu:

Ultra-High – Sample at every 0.009 x 0.009 degree (40,000 samples in a 360 degree circle)

Highest – Sample at every 0.018 x 0.018 degree (20,000 samples in a 360 degree circle)

High – Sample at every 0.036 x 0.036 degree (10,000 samples in a 360 degree circle)

Medium – Sample at every 0.072 x 0.072 degree (5,000 samples in a 360 degree circle)

Low (Preview) – Sample at every 0.288 x 0.288 degree (1,250 samples in a 360 degree circle)

Restore Default Cloud from Scan

Window: ModelSpace

Menu: Tools | Scanner

Action: Executes command

Usage: The **Restore Default Cloud from Scan** command is used to restore the original point cloud to a ModelSpace, including any points hidden using the **Set as Default Cloud** command.

To restore the original point cloud:

1. Click any point within the point cloud.

Note: This command is not available if there are no points to select.

2. From the **Tools** menu, select **Scanner | Restore Default Cloud from Scan**. The original point cloud is restored.

Restore Original Scans

Window: Navigator

Menu: Tools

Action: Executes command

Usage: The **Restore Original Scans** command replaces the default point clouds in the selected ScanWorld with the original point clouds from the scans under the ScanWorld.

- This command is useful if you have unified your scan cloud data, made the unified cloud the default for the ScanWorld, and then want to revert to the original scan data.

Note: Within a ScanWorld, you are able to have a ModelSpace that is unified alongside a ModelSpace that is not unified. Changing the ScanWorld default clouds does not affect existing ModelSpaces. It only affects newly created ModelSpaces and new ControlSpaces if **Add Cloud Constraint** is used. The **Add Cloud Constraint** command will place the default ScanWorld clouds in the ControlSpace if no clouds already exist in the ControlSpace.

Resume

Window: Scan Control (ScanStation, ScanStation 2, HDS3000, HDS6000)

Menu: Scanner Control

Action: Executes command

Usage: The **Resume** command continues the currently-paused scanner operation (scan or image acquisition).

Right Isometric

Window: ModelSpace

Menu: Viewpoint | Standard Views

Action: Executes command

Usage: The **Right Isometric** command moves the current viewpoint to the standard isometric viewing position vector $(1, -1, 1)$ at a distance where the entire scene is visible.

- The **Isometric** command moves the current viewpoint to the standard isometric viewing position vector $(-1, -1, 1)$.

Rotate (Reference Plane)

Window: ModelSpace

Menu: Tools | Reference Plane

Action: Toggles function ON/OFF

Usage: The **Rotate** command toggles the rotate function in **Drawing** mode, which allows you to interactively rotate the active Reference Plane about its origin.

To rotate the Reference Plane:

- From the **Tools** menu, point to **Reference Plane**, and then select **Rotate** (or click the **Rotate** icon). You are now in **Drawing** mode. The cursor changes to the **Drawing Tool** cursor and the **Rotate** function is ON.
- In the ModelSpace, drag the **Drawing Tool** cursor to rotate the active Reference Plane. The Reference Plane is rotated about its origin such that the cross axis (the positive X axis) passes beneath the cursor.
- Release the mouse button. The active Reference Plane is rotated.

Rotate (View Lock)

Window: ModelSpace

Menu: Viewpoint | View Lock

Action: Toggles option ON/OFF

Usage: The **Rotate** command toggles the ability to rotate the viewpoint in **View** mode.

- When **ON**, you cannot rotate the viewpoint.

Save/Edit Coordinate Systems...

Window: ModelSpace

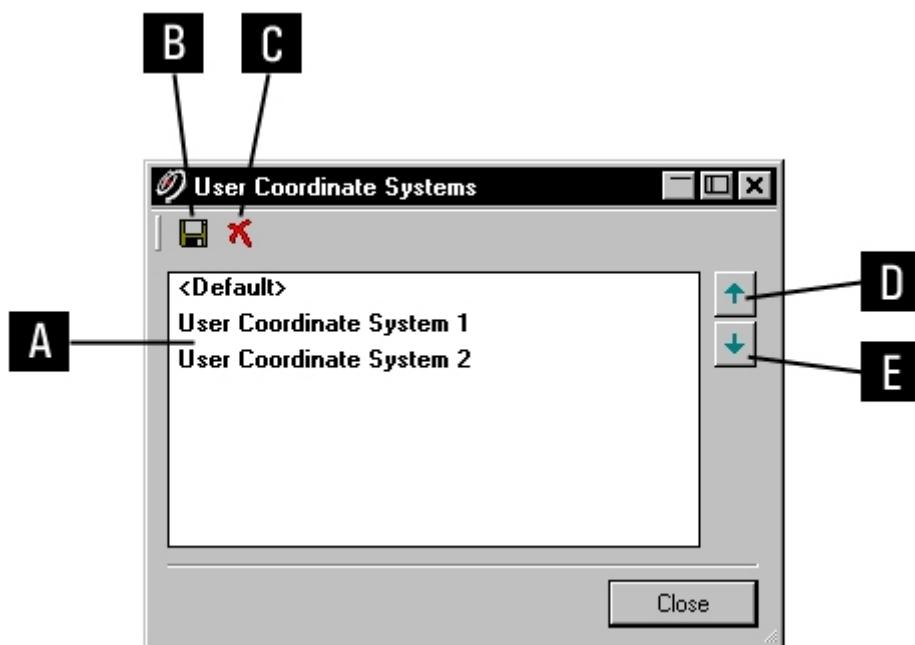
Menu: View | Coordinate System

Action: Opens dialog

Usage: The **User Coordinate Systems** dialog is used to save, restore, and delete user-defined coordinate systems.

- Using multiple user-defined coordinate systems can be very helpful in the navigation and analysis of large ModelSpaces. You may find it useful to define temporary coordinate systems based on objects that provide necessary axes and origins of interest for use in the insertion or editing of objects with, for instance, the **Rotate** command.

User Coordinate Systems Dialog



- List of saved coordinate systems. Double-click to set the coordinate system. The current coordinate system is shown in bold-face.
- Save the current coordinate system.
- Delete the selected coordinate system.
- Move the selected coordinate system up through the list saved coordinate systems.
- Move the selected coordinate system down through the list saved coordinate systems.

To save the current coordinate system:

1. From the **View** menu, point to **Coordinate System**, and then select **Save/Edit Coordinate Systems....** The **User Coordinate Systems** dialog appears.
2. Click the **Save** icon. The **Coordinate System Name** dialog appears.
3. Enter a name for the current coordinate system, and then click **OK**. The current coordinate system is saved.

To restore a saved coordinate system as the current coordinate system:

1. From the **View** menu, point to **Coordinate System**, and then select **Save/Edit Coordinate Systems....** The **User Coordinate Systems** dialog appears.
2. From the list of saved coordinate systems, double-click the coordinate system you want to restore. The selected coordinate system is restored.

To delete a saved coordinate system:

1. From the **View** menu, point to **Coordinate System**, and then select **Save/Edit Coordinate Systems....** The **User Coordinate Systems** dialog appears.
2. Select the coordinate system you want to delete, and then click the **Delete** icon. The selected coordinate system is deleted.

Sample Grid

Window: **ModelSpace**

Menu: **Tools | Mesh**

Action: Opens dialog

Usage: The **Sample Grid** command displays a dialog that indicates how the surface of the selected mesh(es) will be sampled over a regular grid.

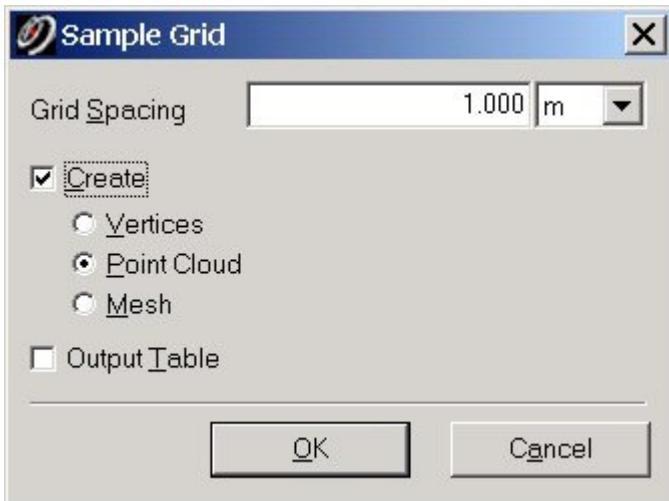
- The sampling is performed along the axis perpendicular to the active reference plane. The reference plane should be perpendicular to one of the major axes of the current UCS. The mesh(es) must be TIN relative to the reference plane.
- If a polyline/polygon is also selected, only the area within the polyline/polygon (projected onto the active reference plane) is sampled.

To Sample Grid

1. Select the mesh(es) to be sampled.
- To Sample Grid within a specific boundary, also multi-select a polyline or polygon object.
2. From the **Tools** menu in the **ModelSpace** window, point to **Mesh**, then select **Sample Grid....** The **Sample Grid** dialog appears.
3. Change the **Grid Spacing** of the active reference plane. The samples will be taken on the reference plane grid with this spacing.
4. Select the **Create** check box to create an object in the ModelSpace to represent the samples.
Select the form of the sampling output to be generated.
- Select **Vertices** to generate individual Vertex objects at each surface sample.

- Select **Point Cloud** to create a Point Cloud whose points correspond to the surface samples.
- Select **Mesh** to create a Mesh whose vertices correspond to the surface samples.
- 5. Select the **Output Table** check box to display the **Sample Grid Results** dialog that contains a table listing the individual sample points.
- 6. Click **OK** to sample the surfaces and generate the requested output.
- Where multiple meshes overlap, the sample with the highest elevation relative to the measurement direction is used.

Sample Grid Dialog



Grid Spacing: Change the **Grid Spacing** of the active reference plane. The samples will be taken on **Spacing**:the reference plane grid with this spacing.

Create: Select the **Create** check box to create an object in the ModelSpace to represent the samples. Select the form of the sampling output to be generated.

Vertices: Select **Vertices** to generate individual Vertex objects at each surface sample.

Point Cloud: Select **Point Cloud** to create a Point Cloud whose points correspond to the surface

Cloud: samples.

Mesh: Select **Mesh** to create a Mesh whose vertices correspond to the surface samples.

Output Table: Select the **Output Table** check box to display the **Sample Grid Results** dialog that

Table: contains a table listing the individual sample points.

OK: Click **OK** to sample the surfaces and generate the requested output.

Save/Edit Viewpoints...

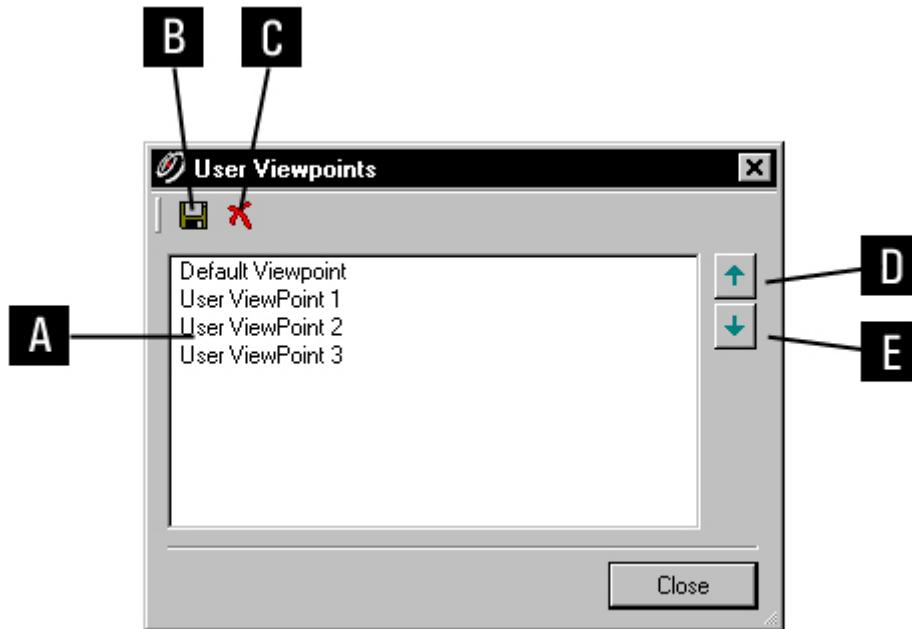
Window: **ModelSpace**

Menu: **Viewpoint**

Action: Opens dialog

Usage: The **User Viewpoints** dialog is used to save, restore, and delete user-defined viewing positions.

User Viewpoints Dialog



- A. List of saved viewpoints. Double-click to set the viewpoint.
- B. Save the current viewpoint.
- C. Delete the selected viewpoint.
- D. Move the selected viewpoint up through the list saved coordinate systems.
- E. Move the selected viewpoint down through the list saved coordinate systems.

To save the current viewpoint:

1. From the **Viewpoints** menu, select **Save/Edit Viewpoints....** The **User Viewpoints** dialog appears.
2. Click the **Save** icon. The **User Viewpoint Name** dialog appears.
3. Enter a name for the current viewpoint and click **OK**. The viewpoint is saved.

To set the current viewpoint:

1. From the **Viewpoint** menu, select **Save/Edit Viewpoints....** The **User Viewpoints** dialog appears.
2. Double-click the viewpoint you want to restore. The selected viewpoint is restored.

To delete a saved viewpoint:

1. From the **Viewpoint** menu, select **Save/Edit Viewpoints....** The **User Viewpoints** dialog

appears.

2. Select the viewpoint you want to delete, and then click the **Delete** icon. The selected viewpoint is deleted.

Save Measurements

Window: ModelSpace

Menu: Tools | Measure

Action: Toggles menu item ON/OFF

Usage: The **Save Measurements** command toggles the saving of measurements as you take them.

- When **ON**, the current measurement and all new measurements are saved and can be viewed and edited from the **Measurements** dialog. (See the *Edit Measurements* entry in this chapter).
- When **OFF**, each new measurement replaces the previous unsaved measurement.

Save View

Window: ModelSpace

Menu: View

Action: Executes command

Usage: The **Save View** command functions identically to the **Save View As...** command, except that the view is not renamed.

Save View As...

Window: ModelSpace

Menu: View

Action: Opens dialog

Usage: The **Save ModelSpace View** dialog is used to create a new ModelSpace View, which is a collection of overriding settings for graphical viewing parameters, including initial viewing position.

- Multiple views of a ModelSpace can be saved and restored as a convenient means of storing various presentations of a ModelSpace. You can save and then restore as many ModelSpace views as needed.
- For a comprehensive list of what is saved with a ModelSpace View, see the [ModelSpace Views](#) entry in the *Modeling* chapter.

To save the current ModelSpace View:

1. When you have a ModelSpace View that you want to save, select **Save View As...** from the **View** menu. The **Save ModelSpace View** dialog appears.
 - To specify a name for the view, enter the name you want in the **ModelSpace View Name** field.
2. Click **OK**. The ModelSpace View is saved and remains the active ModelSpace View. Any further changes to this View are persistent.
 - Saved ModelSpace views are stored with their ModelSpace in the **Navigator** window.
 - To open a ModelSpace View in a new ModelSpace window, double-click the view you want from the **Navigator** window.
 - To open a different ModelSpace View in the current ModelSpace window, use the **ModelSpace Views...** command.

Scan

Window: Scan Control

Menu: Scanner Control

Action: Executes command

Usage: The **Scan** command begins a scan of the targeted area using the current settings.

- Scanning progress is displayed in the **Progress** bar in the bottom of the window. By default, sample points are displayed in the connected ModelSpace viewer (see the *Open ModelSpace Viewer* entry in this chapter) as they are acquired (see the [Show Incoming Points](#) command).
- Scans are saved in the **Scans** folder under the destination ScanWorld.
- For an overview of the scanning process, see the [Scanning](#) entry in the *Getting Started* chapter.

Scan Filter

Window: Scan Control

Menu: Window

Action: Expands Tab

Usage: The **Scan Filter** command expands the **Scan Filter** tab in the Scanner Control Panel.

- Points sampled by the scanner that fall outside the specified range and/or intensity parameters are discarded without being recorded to the database.

Scan Filter Tab Options

RANGE

To specify minimum or maximum range, click the checkboxes. Ranges can be entered directly using the fields, or by selecting points in the ModelSpace viewer.

INTENSITY

To specify minimum or maximum intensity, click the checkboxes. Ranges can be entered directly using the fields, or by selecting points in the ModelSpace viewer. Any point with intensity less than the minimum intensity is rejected; any point with intensity greater than maximum intensity is rejected. Intensities are from 0 to 1. More energy returned to the scanner for a given sample results in a higher intensity.

Scanner

Window: **ModelSpace**

Menu: **Tools**

Action: Displays submenu

Usage: Pointing to **Scanner** displays submenu commands, submenus, and dialogs to used to manage scan data, set and restore point clouds, and create and use a simulate scanner.

The following commands, submenus, and dialogs are available from the **Scanner** submenu:

[ScanWorld Explorer](#), [Set As Default Cloud](#), [Restore Default Cloud from Scan](#), [Set ScanWorld Default Clouds](#), [Show Incoming Points](#), [Create Simulated Scanner](#), [Update Simulated Scanner Position](#)

Scanner Diagnostics

Window: **Navigator**

Menu: **View**

Action: Opens dialog

Usage: The **Scanner Diagnostics...** command displays scanner diagnostics related to the selected object.

- When *Cyclone* connects to an HDS3000, ScanStation, or ScanStation 2, *Cyclone* queries the scanner for the scanner's initialization diagnostics. *Cyclone* stores the diagnostics with the scanner object, marked with the current date and time.

▪▪▪When *Cyclone* starts a scan or acquires an image, the initialization diagnostics obtained at the time of connection are stored with the destination ScanWorld. Also, the current scanner parameters are retrieved from the scanner and stored with the resulting Scan/Image.

Scanner Laser Mode

Window: Scan Control (HDS6000)
Menu: Scanner Control
Action: Displays submenu
Usage: Pointing to **Scanner Laser Mode** displays submenu commands (**Normal Power** and **Low Power**) used to configure the power of the laser. **Normal Power** may detect darker surfaces farther away, and the minimum range is 30 cm. **Low Power** is useful when objects are closer to the scanner than 30 cm, because its minimum range is 0 cm. However, **Low Power** may not detect darker surfaces. The maximum range is the same in each mode. The sampling noise may be slightly different for each mode.

Scanner Range Mode

Window: Scan Control (HDS4500)
Menu: Scanner Control
Action: Displays submenu
Usage: Pointing to **Scanner Range Mode** displays submenu commands (**Normal** and **Close-In**) used to configure the power of the laser, resulting in different minimum and maximum ranges. **Normal** achieves the maximum range of the scanner, but there is a minimum range within which samples are ignored. **Close-In** is useful when objects are closer to the scanner than the **Normal** mode's minimum range, because it decreases the laser power and has a shorter minimum range. However, **Close-In** has half the maximum range of **Normal**. The sampling noise may be slightly different for each mode.

Scanner Speed

Window: Scan Control (HDS4500, HDS6000)
Menu: Scanner Control
Action: Displays submenu
Usage: Pointing to **Scanner Speed** displays submenu commands (**Default** and **Low**) used to configure the rotational speed of the scanner. **Low** speed turns the scanner more slowly, but the scan makes less audible noise and the scan may be more stable. Some scanner resolutions support only one speed, in which case this setting is ignored.

Scanners...

Window: Navigator

Menu: Configure

Action: Opens dialog

Usage: The **Configure Scanners** dialog is used to add and remove scanners.

To add a scanner:

1. From the **Configure** menu, select **Scanners....** The **Configure Scanners** dialog appears.
2. Click **Add**. The **Add Scanner** dialog appears.
3. From the **Scanner Model** list, select the model of the scanner that you are adding.
4. In the **Scanner Name** field, type a logical name for the scanner you are adding.
5. In the **IP Address** field, enter the IP Address for the scanner you are adding.
 - The IP Address is printed on the scanner.
6. Click **OK**. The scanner is added.
- To close the **Configure Scanners** dialog and return to the **Navigator** window, click **Close**.

Note: The Cyclone Scan Control interface with the [HDS4500](#) is via firewire connection and no IP Address is entered.

To remove a scanner:

1. From the **Configure** menu, select **Scanners....** The **Configure Scanners** dialog appears.
2. From the list of scanners, select the scanner that you want to remove, and then click **Remove**.

The scanner is removed.
 - To close the **Configure Scanners** dialog and return to the **Navigator** window, click **Close**.

To toggle scanner visibility:

- Scanners with visibility toggled OFF are not displayed in the **Navigator** window, but are not removed.
- 1. From the **Configure** menu, select **Scanners....** The **Configure Scanners** dialog appears.
- To toggle visibility, select/deselect the box below the **Visibility** icon next to the appropriate scanner.
- To close the **Configure Scanners** dialog and return to the **Navigator** window, click **Close**.

ScanWorld

Window: Navigator

Menu: Create

Action: Executes command

Usage: The **ScanWorld** command creates a new ScanWorld in the selected location.

- A new ScanWorld can be created in a database, or project.

To create a new ScanWorld:

1. In the **Navigator** window, select the database or project into which you want to add a new ScanWorld.
2. From the **Create** menu, select **ScanWorld**. The ScanWorld is created.

ScanWorld Explorer...

Window: ModelSpace

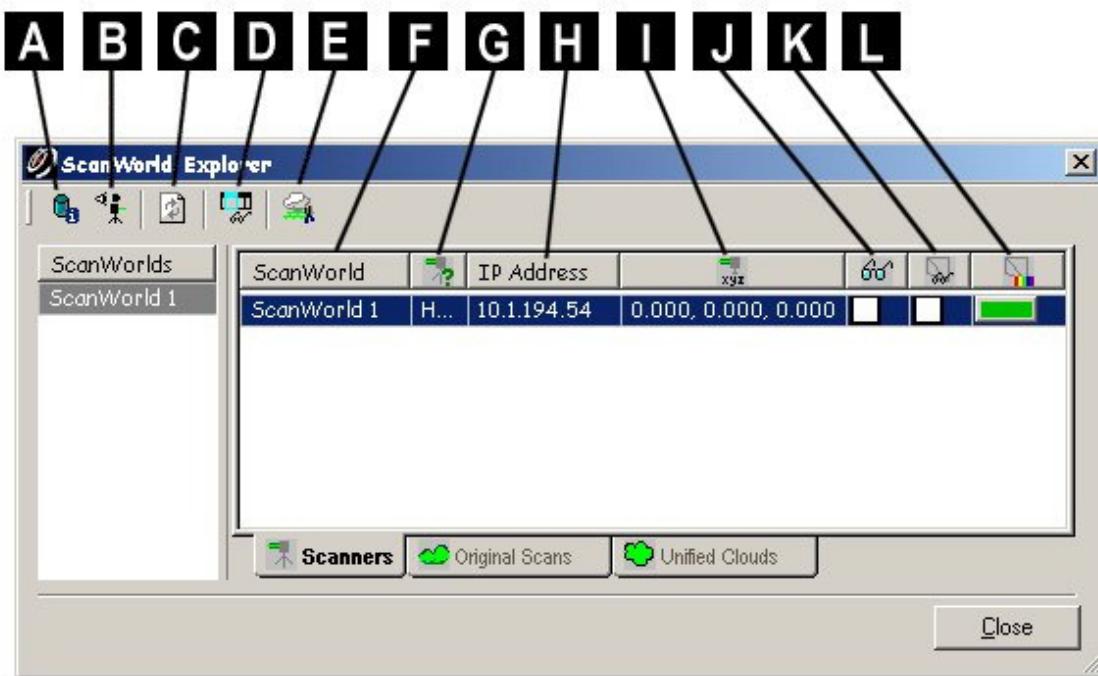
Menu: Tools | Scanner

Action: Opens dialog

Usage: The **ScanWorld Explorer** dialog is used to show and hide all scanners and scans registered in the current ScanWorld.

- The **ScanWorld Explorer** contains three tabs: **Scanners**, **Original Scans**, and **Unified Clouds**. The different tabs contain a number of columns containing information about the scanners used, scans taken, and unification of point clouds.
- To customize the **ScanWorld Explorer** dialog, right-click the column header. Customization options appear.

ScanWorld Explorer Dialog - Scanners Tab

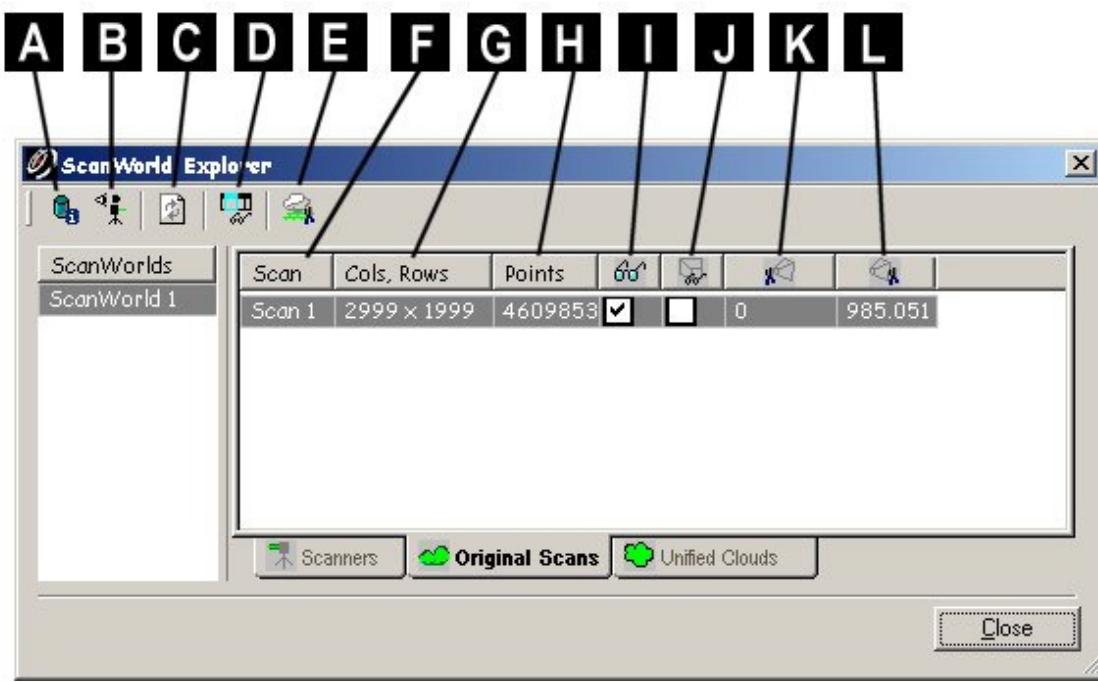


A. Opens separate dialog where information about the selected scanner is displayed

B. Aligns the viewpoint in the ModelSpace View window so it matches the viewpoint of the selected scanner

- C. Refresh All
- D. Customize columns
- E. Segments the selected scan based on each point's distance to the scanner
- F. The name of the ScanWorld containing the scan
- G. Make / Model / ID column
- H. Displays IP address for the selected scanner
- I. Scanner position in ModelSpace
- J. Display a scanner object in ModelSpace representing the selected scanner
- K. Display boundary of the maximum Field of View of the selected scanner.
- L. Set Field of View color for the selected scanner

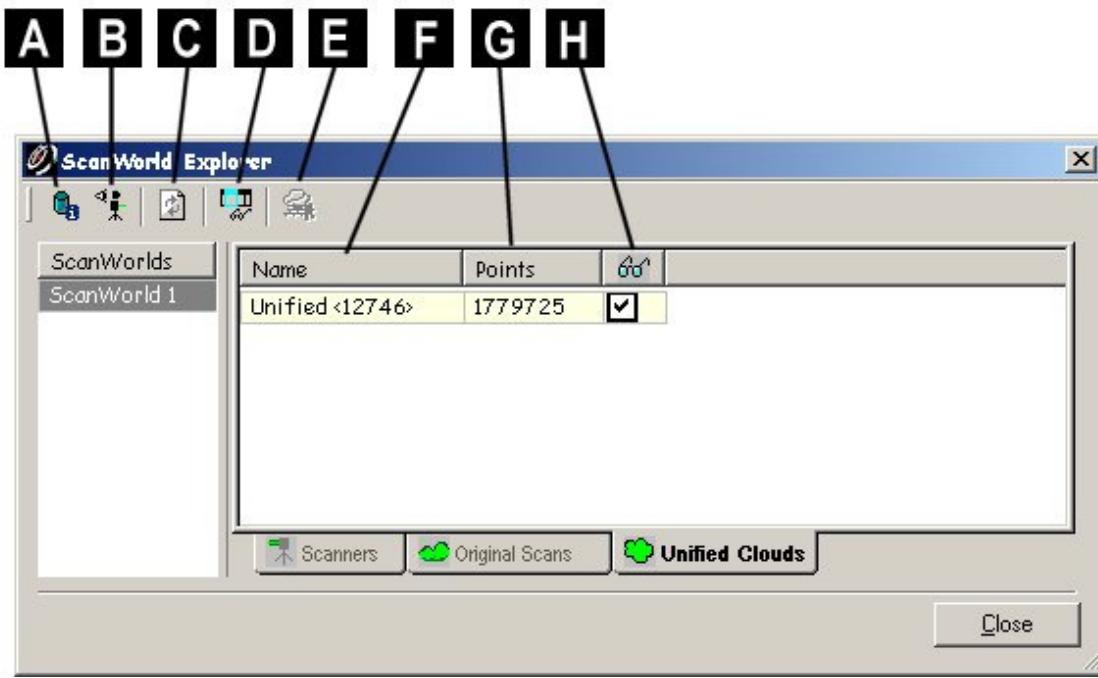
ScanWorld Explorer Dialog Original Scans



- A. Opens separate dialog where information about the selected scanner is displayed
- B. Aligns the viewpoint in the ModelSpace View window so it matches the viewpoint of the selected scanner

- C. Refresh All
- D. Customize columns
- E. Segments the selected scan based on each point's distance to the scanner
- F. Displays the name of the scan
- G. Displays the actual dimensions of the selected scan
- H. Displays the number of points contained within the selected scan
- I. Toggles display of selected scan's point cloud in ModelSpace
- J. Toggles display of the Field of View boundary for the selected scan
- K. Displays Field of View's Near Extent for the scan
- L. Displays Field of View's Far Extent for the scan

ScanWorld Explorer Dialog Unified Clouds



- A. Opens separate dialog where information about the selected scanner is displayed
- B. Aligns the viewpoint in the ModelSpace View window so it matches the viewpoint of the selected scanner

- C. Refresh all
- D. Customize columns
- E. Segments the selected scan based on each point's distance to the scanner
- F. Displays the name of the unified cloud.
- G. Displays the number of points contained within the selected unified cloud
- H. Toggles the display of the selected unified cloud

To segment a scan at a user-specified distance from its scanner:

1. Invoke the ScanWorld Explorer.
2. Select the **Scanners or Original Scans** tab.
3. Select one or more ScanWorld in the left pane.
 - If no ScanWorlds are selected in the left pane, all ScanWorlds are considered to be selected.
4. Select one or more items in the **Scanners or Original Scans** tab.
 - If no items are selected in the right pane, all Scans for the ScanWorlds selected in the left pane are considered to be selected.
5. Click the **Segment Scans by Distance** toolbar icon. The **Segment Scans by Distance** dialog appears.
6. Enter the **Distance** at which the Scans will be segmented. Each cloud will be segmented based on its points' distances to the scanner that scanned the cloud.
 - Unified clouds, fine/coarse target scans, and merged clouds are not affected.
7. Select the action that will be applied to the segmented points.
 - Select **Keep Points** to keep all points.
 - Select **Delete Points Beyond Distance** to delete all points further than the user-specified distance.
 - Select **Delete Points Within Distance** to delete all points closer than the user-specified distance.
8. Click **OK**. The clouds corresponding to the selected clouds are segmented. Depending on the options, some of the segmented portions may be deleted.

ScanWorld Info

Window: ModelSpace

Menu: Tools | Info

Action: Opens dialog

Usage: The **ScanWorld Info** dialog is used to view available information about the ScanWorld(s) from which the current ModelSpace is derived.

- The data contained in this dialog is a snapshot of the state of the ScanWorld at the time the dialog is displayed; to update the information, close the dialog and execute the command again.

Sections Manager

Window: ModelSpace

Menu: Tools | Alignment and Section

Action: Opens dialog

Usage: The **Sections Manager** dialog is used to generate, edit, and manage cross-sections along alignments.

Seek Mode

Window: ModelSpace

Menu: Edit | Modes

Action: Executes command

Usage: The **Seek Mode** command initiates **Seek** mode and displays the **Seek** mode cursor, which is used to reset the focal point.

- The viewpoint rotates around and zooms in/out from a focal point. The focal point is a point that is a certain distance in front of the center of the viewpoint.

To reset the focal point:

- From the **Edit** menu, point to **Modes**, and then select **Seek Mode**. The cursor changes to the **Seek Mode** cross-hair.
- Pick the point that you want as your new focal point. The viewpoint is adjusted and the focal point is set.

Segment Cloud

Window: ModelSpace

Menu: Create Object

Action: Displays submenu

Usage: Pointing to **Segment Cloud** displays submenu commands for subdividing a cloud of points into smaller subsets.

The following commands and dialogs are available from the **Segment Cloud** submenu:

Cut by Fence, Cut by Intensity..., Cut Near Ref Plane..., Trim Edges, Cut Sub-Selection

- For more information on the segmentation process, see the [Segmentation](#) entry in the *Modeling* chapter.

Segment by Polyline

Window: ModelSpace

Menu: Tools | Mesh

Action: Executes command

Usage: The **Segment by Polyline** command splits faces in the selected mesh wherever the selected line or polyline crosses it.

- If a line is selected, the line is extended infinitely in each direction for purposes of this command, effectively dividing the mesh into two pieces.
- If a polyline is selected, it is automatically closed into a polygon for purposes of this command.
- The crossing is determined using the projection of the selected line or polyline to the active Reference Plane.
- The mesh must be a TIN relative to the active Reference Plane.

To Segment Faces on a mesh:

1. Select the mesh(es) to be segmented.
2. Multi-select the line, polyline, or polygon object that defines the boundary to use.
3. From the **Tools** menu, point to **Mesh**, then select **Segment by Polyline**. The mesh is segmented using the projection of the selected line, polyline, or polygon to the active Reference Plane.

Segment Polyline

Window: ModelSpace

Menu: Create Object

Action: Executes command

Usage: The **Segment Polyline** command segments the selected polyline(s) into subsets.

To segment a polyline:

1. Draw a fence around the portions of the polyline that you want to segment. (See the *Using a Fence* section in the *Modeling* chapter.)
 2. From the **Create Object** menu, point to **Segment Polyline**. The polyline is segmented where it crosses the fence.
- The resultant separate polylines can be selected independently.

Select a Project...

Window: Scan Control

Menu: Project

Action: Opens dialog

Usage: The **Select a Project** dialog is used to select a destination project folder for scan and image data acquired from the scanner.

- You must select a project folder before beginning a scan or getting an image.
- The **Select a Project** dialog can also be accessed by clicking the button to the right of the **Project** box in the **Scan Control** panel.

To select a project for your scan data:

1. From the **Project** menu, select **Select a Project....** The **Select a Project** dialog appears.
- You can create a new project for your scan data by clicking **Create**; you can then select the created project
2. Navigate to an existing project, and then click **OK**. Your destination project is selected.

Select a ScanWorld...

Window: Scan Control

Menu: Project

Action: Opens dialog

Usage: The **Select a ScanWorld...** command opens the **Select a ScanWorld** dialog, which is used to select a destination ScanWorld for scans and images acquired from the scanner.

- If you do not select a ScanWorld before beginning a scan or getting an image, a ScanWorld is automatically created in the parent project.
- The **Select a ScanWorld** dialog can also be accessed by clicking the button to the right of the **ScanWorld** box in the **Scan Control** panel.

To select a ScanWorld for your scans and images:

1. From the **Project** menu, select **Select ScanWorld....** The **Select a ScanWorld** dialog appears.
- You can create a new ScanWorld for your scan data by clicking **Create**; you can then select the created ScanWorld.
2. Navigate to an existing ScanWorld, and then click **OK**. Your destination ScanWorld is selected.
- Only ScanWorlds within the parent project can be selected.

Select All

Window: ModelSpace, Image Viewer

Menu: Selection

Action: Executes command

Usage: The **Select All** command selects every selectable object currently loaded in the ModelSpace.

- In the **Image Viewer** window, this command selects all annotation pick points.
- To deselect all objects, press **ESC**.

Note: An object will not be selected if the object has not yet been loaded, nor if the Object Type has been set “not selectable” in the View Properties dialog. See the *Set Object Visibility* entry in this chapter for more information.

Select All (Point Cloud Sub-Selection)

Window: ModelSpace

Menu: Selection | Point Cloud Sub-Selection

Action: Executes command

Usage: The **Select All** command selects all selectable point clouds in the ModelSpace.

Select Connected

Window: ModelSpace

Menu: Tools | Piping

Action: Executes command

Usage: The **Select Connected** command selects all of the piping components connected to the selected piping component

- This command can be useful for setting or copying the piping annotations for a selected pipe run, putting the selected pipe run on a layer, changing the material of the selected pipe run to distinguish it from the other pipe runs, etc..

Select Fenced

Window: ModelSpace

Menu: Selection

Action: Executes command

Usage: The **Select Fenced** command selects every object and point cloud within or overlapping the drawn data fence.

- For more information on working with fences, see the [*Using a Fence*](#) entry in the *Modeling* chapter.

Self-Test

Window: Scan Control (HDS4500)

Menu: Scanner

Action: Executes command

Usage: The **Self-Test** command performs an operational safety check on the scanner's laser.

Servers...

Window: Navigator

Menu: Configure

Action: Opens dialog

Usage: The **Configure Servers** dialog is used to add, remove, and toggle visibility of servers.

To add a server:

1. From the **Configure** menu, select **Servers....** The **Configure Servers** dialog appears.
2. Click **Add**. The **Add Server** dialog appears.
3. In the **Server Name** field, type the Microsoft Network name for the server to be added, and then click **Add**. The **Configure Servers** dialog reappears displaying the newly added server.
4. Click **Close**. The server is created.

To remove a server:

1. From the **Configure** menu, select **Servers....** The **Configure Servers** dialog appears.
2. From the list of servers, select the server that you want to remove, and then click **Remove**.
The server is removed.
- To close the **Configure Servers** dialog and return to the **Navigator** window, click **Close**.

To toggle server visibility:

1. From the **Configure** menu, select **Servers....** The **Configure Servers** dialog appears.
- To toggle visibility, select/deselect the box below the **Visibility** icon next to the desired server.
- To close the **Configure Servers** dialog and return to the **Navigator** window, click **Close**.

Set Active Alignment

Window: ModelSpace

Menu: Tools | Alignment and Section

Action: Executes command

Usage: The **Set Active Alignment** command enables station notation on the selected point, relative to the active alignment.

- Station-notation values are only supported for alignments that lie completely in the horizontal plane.
- For more information about alignments and the sections process, see the *Sections Manager* chapter.

Note: Station Notation in Preferences must be set to On to display station notation.

To enable use of station notation for a selected alignment:

1. Select an alignment.
- If you change the “Up” direction, all station-notations are relative to the new “up” direction, except for existing measurements. The alignment must be perpendicular to the current up direction for station notation to be available.
2. From the **Tools** menu, point to **Alignment and Section** and then select **Set Active Alignment**
- All station-notation values will be relative to the selected alignment (except in Sections Manager).
- Existing measurements are not affected.
- The active alignment may be hidden.
- If the active alignment is deleted, the active alignment is reset.

Set as Default Cloud

Window: ModelSpace

Menu: Tools | Scanner

Action: Executes command

Usage: The **Set as Default Cloud** is used to remove unwanted or unintended point samples from a selected point cloud within the current ScanWorld and any subsequent ScanWorld derived from the current ScanWorld.

- Invoking this command prevents unwanted points from appearing as part of the point cloud, even when a new ModelSpace is created, or when the “original” point cloud is shown via the [ScanWorld Explorer](#) dialog.

Note: Points are not actually deleted from the database when this command is activated; rather, they are hidden, and can be restored using the [Restore Default Cloud from Scan](#) command.

Set Atmospheric Correction...

Window: Scan Control

Menu: Scanner Control

Action: Opens dialog

Usage: The **Atmospheric Correction** dialog is used to adjust the range of each sampled point to compensate for current temperature and pressure conditions while scanning.

To make atmospheric corrections:

1. From the **Scanner Control** menu, select **Set Atmospheric Correction....** The **Atmospheric Correction** dialog appears.
2. Edit the appropriate settings and click **OK**. Atmospheric corrections are applied to adjust the range of each sampled point for subsequent scans within the session.
 - To set temperature and range settings, select the appropriate radio button and enter values in the **Ambient Temperature** and **Atmospheric Pressure** fields. The **ppm Total** (parts per million) is calculated.
 - To set **ppm Total** directly, click the ppm radio button, and then enter a value in the **ppm Total** field.
 - To reset temperature or pressure defaults, click the appropriate **Default** button.

Set Datum to Pick Point

Window: ModelSpace

Menu: Tools | Measure | Datum

Action: Executes command

Usage: The **Set Datum to Pick Point** command sets the datum point at the picked point.

To set the datum at the picked point:

1. Pick the point where you want to locate the datum point.
2. From the **Tools** menu, point to **Measure**, and then point to **Datum** and select **Set Datum to Pick Point**. The datum is relocated.

Set Distance to Next Vertex...

Window: ModelSpace

Menu: Tools | Drawing

Action: Opens dialog

Usage: The **Distance to Next Vertex** dialog is used in **Drawing** mode to set the distance between a vertex that has been plotted and the next vertex to be plotted.

- After plotting at least one drawing vertex, the distance to the next vertex plotted can be set using the **Distance to Next Vertex** dialog. The next vertex plotted will be exactly that distance

away from the previous vertex. Some shapes and steps may not support this distance (e.g., plotting the last vertex in a square).

Set from Active Ref Plane (Cutplane)

Window: ModelSpace

Menu: Tools | Cutplane

Action: Executes command

Usage: The **Set from Active Ref Plane** command aligns the active Cutplane to the active Reference Plane.

- For information on Cutplane attributes, see the [Add/Edit Cutplanes](#) command.

Set From Points...

Window: ModelSpace

Menu: View | Coordinate System

Action: Opens dialog

Usage: The **Set Coordinate System from Pick Points** dialog is used to set any or all of the following: a reference point, the azimuth (horizontal orientation), and the “up” direction (vertical orientation).

To set the reference point only:

1. In **Pick** mode, click the point to use as the reference point.
2. From the **View** menu, point to **Coordinate System**, and then select **Set from Points**. The **Set Coordinate System From Pick Points** dialog appears.
 - Setting the reference point does not affect the current orientation of the coordinate axes.
3. Enter the coordinate for the reference point, and then click **OK**. The new coordinate is applied.

To set the reference and azimuth:

1. In **Pick** mode, click the point to use as the reference point.
2. Mult-select the point you want to use for azimuth reference.
3. From the **View** menu, point to **Coordinate System**, and then select **Set from Points**. The **Set Coordinate System from Pick Points** dialog appears.
 - The pick points are displayed in the ModelSpace View along with a number indicating the order in which the points were picked. Drop-down lists are available in each field in the dialog to choose the appropriate reference pick point.
4. In the **Reference Point** fields, enter the coordinate for the reference point.
5. In the **Azimuth Point** field, enter a value for the degrees to the azimuth point from the reference point.
6. Click **OK**. The new coordinates are applied, and the horizontal axes are rotated.
 - Setting the origin and azimuth does not affect the current “up” direction or axis.

To set the reference, azimuth, and “up” direction:

1. In **Pick** mode, click to use as the reference point.
2. Click the point you want to use for azimuth reference.
3. Mult-select a pair of points known to be in vertical alignment for the “up” direction reference.
(Pick points used for either an azimuth or reference (origin) can also be used as part of an “up direction pair”).
- These point pairs can come from a Leica Geosystems HDS dual-target pole or another structural feature.
4. If there are any points with known elevation, multi-select them as well to aid in vertical accuracy.
5. From the View menu, point to Coordinate System, and then select Set from Points. The Set Coordinate System from Pick Points dialog appears.
- The pick points are displayed in the ModelSpace View along with a number indicating the order in which the points were picked. Drop-down lists are available in each field in the dialog to choose the appropriate reference pick point.
6. In the **Reference Point** fields, enter the coordinate for the reference point.
7. In the **Azimuth Point** field, enter a value for the degrees to the azimuth point from the reference point.
8. In the **Top Point** and **Bottom Point** fields, select the corresponding pick points from the drop-down lists.
9. If known elevation points have been picked, select the appropriate pick points from the drop-down list in the **Elevation Point 1** and **2** fields, and then enter a value for the elevations.
10. Click **OK**. The new coordinates are applied, and the horizontal and vertical axes are rotated.

Set from Reference Plane

Window: **ModelSpace**

Menu: **View | Coordinate System**

Action: Executes command

Usage: The **Set from Reference Plane** command aligns the coordinate system of the ModelSpace to the active reference plane.

Set from Selection

Window: **ModelSpace**

Menu: **Edit | Fence**

Action: Executes command

Usage: The **Set from Selection** command uses the selected linear object (such as an arc, circle, polyline, polygon, etc.) to define a data fence.

- For more information on working with fences, see the [Using a Fence](#) entry in the *Modeling* chapter.

To create a fence from a linear object:

1. Select the linear object you wish to use to define a data fence.
- Linear objects can be created by a variety of methods including via the 2D drawing tools. If an arc or other open curvilinear object is selected, the end points are connected for purposes of defining the data fence.
2. From the **Edit** menu, point to **Fence**, and then select **Set from Selection**. The fence is created.
- The resulting data fence can be used in all fencing operations.

Set Grid Spacing...

Window: ModelSpace

Menu: Edit Object | Snapping Grid

Action: Opens dialog

Usage: The **Set Grid Spacing** dialog is used to set the interval by which a handle can be moved or a vertex can be placed (in **Drawing** mode) when using the snapping grid.

To set grid spacing:

1. From the **Edit Object** menu, point to **Snapping Grid**, and then select **Set Grid Spacing...**. The **Set Grid Spacing** dialog appears.
2. In the **Grid Spacing** field, enter a value for the interval by which a handle can be moved when **Snap to Grid** is ON.
3. Click **OK**. The grid spacing is set.

Set Half-Space at Pick

Window: ModelSpace

Menu: Tools | Cutplane

Action: Displays submenu

Usage: Pointing to **Set Half-Space at Pick** displays submenu commands to specify the active Cutplane as a half-space, based on the last pick point.

See also: [View Half-Space](#) and [Add/Edit Cutplanes](#)

To set the Cutplane half-space at the pick point:

1. Pick the point through which the active Cutplane will pass.
2. From the **Tools** menu, point to **Cutplane**, and then point to **Set Half-Space at Pick** and select the axis you want (X, Y, or Z Axis), in the direction you want (along the positive or negative axis). The active Cutplane is set on the pick point, perpendicular to the selected axis. The direction of the axis is the side that is hidden.
- The active Cutplane and the **View Half-Space** option are automatically enabled.

Set Home ScanWorld

Window: Registration

Menu: ScanWorld

Action: Executes command

Usage: The **Set Home ScanWorld** command designates the Home ScanWorld for the current Registration.

- The Home ScanWorld defines the reference coordinate system for the ScanWorld created from a successful Registration.
- By default, the first ScanWorld added to the Registration is set as the Home ScanWorld.
- The Home ScanWorld is displayed in bold type with red axis in its icon.

To set the Home ScanWorld:

1. From the **ScanWorlds' Constraints** tab or the **ModelSpaces** tab, select the ScanWorld that you want to be the Home ScanWorld.
- There must already be ScanWorlds added to the Registration before a Home ScanWorld can be chosen.
2. From the **ScanWorld** menu, select **Set Home ScanWorld**. The Home ScanWorld is set and is displayed in bold type.

Set Limit Box by Cursor

Window: ModelSpace

Menu: View

Action: Executes command

Usage: The **Set Limit Box by Cursor** command lets the user drag the cursor to interactively define the extents of the limit box, centered around the current focal point. The limit box is aligned to the current viewpoint.

- To set the focal point at a selected point, use the **Seek** mode.
- To cancel setting the limit box with the cursor, click the right mouse button at any time.

Set Limit Box by Fencing

Window: ModelSpace, Registration

Menu: ModelSpace: View

Registration: Viewers

Action: Executes command

Usage: The **Set Limit Box by Fencing** command lets the user drag the rectangular fence to interactively define the extents of the limit box anywhere in the window. The

center of the limit box is determined automatically in 3D. The limit box is aligned to the current viewpoint.

- If the center of the fence falls over an object or cloud point, the center of the limit box is set to that coordinate, and the limit box is a cube. Otherwise, the limit box's depth is set to contain the objects within the fence.
- To cancel setting the limit box with the fence, click the right mouse button while dragging the fence.

Set Object Visibility...

Window: ModelSpace

Menu: View

Action: Opens dialog

Usage: The **Set Object Visibility...** command opens the **Selectable/Visible** tab of the **View Properties** dialog, which is used to set the visible and selectable properties for classes of objects.

- To make a class of objects visible, select the check box in the **Visible** column next to the object type.
- To make a class of objects selectable, select the check box in the **Selectable** column next to the object type.
 - Settings in the **Selectable/Visible** tab interact with **Layers** settings. For an object to be visible, both its layer and its class must be visible; otherwise, it is not drawn.

Set Offset...

Window: ModelSpace

Menu: Tools | Cutplane

Action: Opens dialog

Usage: The **Cutplane Offset** dialog is used to set the increment by which the active Cutplane is raised and lowered using the **Raise Active Cutplane** and **Lower Active Cutplane** commands in the **Cutplanes** submenu.

To set the offset for the active Cutplane:

1. From the **Tools** menu, point to **Cutplane**, and then select **Set Offset...**. The **Cutplane Offset** dialog appears.
2. In the **Offset** field, enter a value for the distance by which the **Raise Active Cutplane** and **Lower Active Cutplane** commands raise/lower the active Cutplane.
 - A positive value indicates that the offset is in the same direction as the Cutplane's normal vector.
3. Click **OK**. The offset is set.

Set on Object (Cutplane)

Window: ModelSpace

Menu: Tools | Cutplane

Action: Executes command

Usage: The **Set on Object** command aligns the active Cutplane with the axis of the picked object, through the pick point.

Set on Object (Reference Plane)

Window: ModelSpace

Menu: Tools | Reference Plane

Action: Executes command

Usage: The **Set on Object** command aligns the active Reference Plane with a picked object.

- The type of picked object and the location of the pick point determine various orientations for the active Reference Plane. The rules can be sophisticated; for example, a pick near the end of a cylinder centers the active Reference Plane at the closest endpoint and perpendicular to the centerline; otherwise, the active Reference Plane is aligned parallel to the cylinder's centerline and perpendicular to the tangent of the cylinder surface at the pick point.

Set Origin

Window: ModelSpace

Menu: View | Coordinate System

Action: Executes command

Usage: The **Set Origin** command sets the coordinate system's origin at the picked point.

To set the origin:

1. In **Pick** mode, click the point where you want to locate the origin.
 2. From the **View** menu, point to **Coordinate System**, and then select **Set Origin**. The origin is relocated to the pick point.
- Setting the origin does not affect the orientation of the coordinate axes.

Set Path

Window: ModelSpace
Menu: Tools | Animation
Action: Executes command
Usage: The **Set Path** command designates the selected spline or spline loop and optional Camera objects as the active path from which an animation can be created or restored.

- For an overview of the animation process, see the [Animation](#) chapter.

Set Plane Origin at Pick Point

Window: ModelSpace
Menu: Tools | Reference Plane
Action: Executes command
Usage: The **Set Plane Origin at Pick Point** command sets the origin of the active Reference Plane at the picked point.

To set the Reference Plane to a picked point:

1. Pick the point at which you want to set the origin of the active Reference Plane.
2. From the **Tools** menu, point to **Reference Plane**, and then select **Set Plane Origin at Pick Point**. The Reference Plane is moved to pass through the pick point without changing its orientation or axes, centered at the pick point.

Set Points Selectable

Window: Registration
Menu: Viewers
Action: Toggles menu item ON/OFF
Usage: The **Set Points Selectable** command toggles the ability to select point clouds in the ModelSpace viewers.

Set Rotational Spacing...

Window: ModelSpace
Menu: Edit Object | Snapping Grid
Action: Opens dialog
Usage: The **Set Rotational Spacing** dialog is used to set the angular interval by which an object can be rotated using its rotation handle when **Snap to Grid** is ON.

To set rotational spacing:

1. From the **Edit Object** menu, point to **Snapping Grid**, and then select **Set Rotational Spacing**.... The **Set Rotational Spacing** dialog appears.
2. In the **Rotational Spacing** field, enter a value for the angular interval by which an object can be rotated when **Snap to Grid** is ON.
3. Click **OK**. The rotational spacing is set.

Set Scan Filters...

Window: Control
Menu: Scanner Control
Action: Opens dialog
Usage: The Set Scan Filters dialog is used to set range and/or intensity parameters outside of which points sampled by the scanner are discarded without being recorded to the database.

To set scan filters:

1. From the **Scanner Control** menu, select **Set Scan Filters**.... The **Set Scan Filters** dialog appears.
2. Set range and intensity parameters.
 - To set range parameters, select the **Min** and/or **Max** check boxes and enter values for the low/high end of the range you want recorded to the database. Next, select **Rectangular** (HDS2500 only) or **Spherical** to designate distance measurements as either point-to-point distances directly to the scanner (spherical) or as a plane in front of the scanner (rectangular).
 - To set intensity parameters, select the **Min** and/or **Max** check boxes and enter values for the low/high intensities of points you want recorded to the database.
3. Click **OK**. The dialog closes and filters are set for any subsequent scans using the current **Scan Control** window.

Set ScanWorld Coordinate System

Window: ModelSpace
Menu: View | Coordinate System
Action: Executes command
Usage: The **Set ScanWorld Coordinate System** command sets the current ModelSpace View's coordinate system and "up" direction as the default for its parent ScanWorld.

- After setting the ScanWorld coordinate system, all new ModelSpaces derived from the ScanWorld are set to the new coordinate system and "up" direction. Also, if set as the Home ScanWorld in a Registration, the ScanWorld's default coordinate system and "up" direction is used as the default coordinate system of the ScanWorld created from that Registration.

Set Selectable...

Window: ModelSpace

Menu: Selection

Action: Opens dialog

Usage: The **Set Selectable...** command opens the **Selectable/Visible** tab of the **View Properties** dialog, which is used to set the visible and selectable properties for classes of objects.

- To make a class of objects selectable, select the check box in the **Selectable** column next to the object type.
- To make a class of objects visible, select the check box in the **Visible** column next to the object type.
- Settings in the **Selectable/Visible** tab interact with **Layers** settings. For an object to be selectable, both its layer and its class must be selectable.

Set ScanWorld Default Clouds

Window: ModelSpace

Menu: Tools | Scanner

Action: Executes command

Usage: The **Set ScanWorld Default Clouds** command designates the current point clouds as the default point clouds that are loaded whenever a new ModelSpace is created from the parent ScanWorld.

- This command is useful after deleting unwanted points from the point clouds, or after using the **Unify Clouds** command.
- To revert to the original scan clouds as the default, use **Restore Original Scans** command.

To set a default point cloud:

1. After creating two or more point clouds through segmentation, select a single point cloud.
2. From the **Tools** menu, point to **Scanner**, and then select **Set ScanWorld Default Clouds**.
The selected point cloud is loaded whenever a new ModelSpace is created from the parent ScanWorld.

Set Slice from Picks

Window: ModelSpace

Menu: Tools | Cutplane

Action: Displays submenu

Usage: Pointing to **Set Slice from Picks** displays submenu commands to specify the active Cutplane as a slice, based on the last two pick points.

See also: [View Slice](#) and [Add/Edit Cutplanes](#)

To set the Cutplane slice from pick points:

1. Pick the point through which one plane of the Cutplane slice will pass.
 2. Shift+Pick the second point through which the other plane of the Cutplane slice will pass.
 3. From the **Tools** menu, point to **Cutplane**, and then point to **Set Slice from Picks** and select the axis to use (X, Y, or Z Axis). The active Cutplane is set perpendicular to the selected axis, and its slice planes are positioned to pass through the two pick points.
- The active Cutplane and the **View Slice** option are automatically enabled.

Set Slice Thickness...

Window: ModelSpace

Menu: Tools | Cutplane

Action: Opens dialog

Usage: The **Set Slice Thickness** dialog is used to set the thickness of the slice used in the **Slice** command (see the *Slice* entry in this chapter).

- When the **Slice** command is toggled ON, all geometry beyond the slice is hidden.

To set slice thickness:

1. From the **Tools** menu, point to **Cutplane**, and then select **Set Slice Thickness...**. The **Set Slice Thickness** dialog appears.
2. In the **Slice Thickness** field, enter a value.
3. Click **OK**. The slice thickness is adjusted.

Set Target Height

Window: ModelSpace

Menu: Tools | Registration

Action: Executes command

Usage: The **Set Target Height** command displays the **Set Target Height** dialog. Set the height of the target above its base point (e.g., if the target had been placed a known height above a known survey point), for use in registration.

Set to { X-Y, X-Z, Y-Z} Plane

Window: ModelSpace

Menu: Tools | Reference Plane

Action: Executes command

Usage: The **Set to { X-Y, X-Z, Y-Z} Plane** command orients the active Reference Plane to the chosen coordinate system axes and centers the Reference Plane's origin at the focal point of the viewpoint.

Set to Object

Window: ModelSpace

Menu: View | Coordinate System

Action: Executes command

Usage: The **Set to Object** command aligns the coordinate system of the ModelSpace to the selected object.

Set to UCS

Window: ModelSpace

Menu: Tools | Reference Plane | Set to UCS

Action: Executes command

Usage: The **Set to UCS** command aligns the active reference plane to the current user coordinate system of the ModelSpace.

Set to Viewpoint

Window: ModelSpace

Menu: Tools | Reference Plane

Action: Executes command

Usage: The **Set to Viewpoint** command sets the active Reference Plane perpendicular to the current viewpoint, centered on the focal point.

Set Up Direction

Window: ModelSpace

Menu: View

Action: Displays submenu

Usage: Pointing to **Set Up Direction** displays submenu commands used to set the "up" direction to one of the major axes.

The following commands are available from the **Set Up Direction** submenu:

X Axis, Y Axis, Z Axis

Set Using One Axis

Window: ModelSpace

Menu: View | Coordinate System

Action: Displays submenu

Usage: The **Set Using One Axis** command displays submenu commands used to reset the orientation of the X, Y, or Z coordinate system axis to match the axis of a selected object.

The following commands are available from the **Set Using One Axis** submenu:

X Axis, Y Axis, Z Axis

To set a coordinate system axis:

1. Select the object to which you want to align the chosen axis.
- The direction in which the axis is oriented is based on the location of the pick point on the reference object. The axis is oriented towards the end of the reference object nearest the pick point.
2. From the **View** menu, point to **Coordinate System**, and then point to **Set Using One Axis** and select the axis you want (X, Y, or Z Axis). The axis in the command you select is aligned exactly to the axis of the reference object, and the other two axes are made perpendicular to the set axis.
- Multiple user-defined coordinate systems can be saved and restored. (See the *Save/Edit Coordinate System* entry in this chapter).
- To specify new orientations for all three coordinate system axes, see the *Set Using Two Axes* (X then Y, etc.) entry in this chapter.

Set Using Two Axes

Window: ModelSpace

Menu: View | Coordinate System

Action: Displays submenu

Usage: Pointing to **Set Using Two Axes** displays submenu commands that are used to reset the orientation of the specified axes to match the axes of two selected objects.

The following commands are available from the **Set Using Two Axes** submenu:

X then Y Axis, X then Z Axis, Y then X Axis, Y then Z Axis, Z then X Axis, Z then Y Axis

To set two coordinate system axes:

1. Select the object to which you want to align the first axis in the command.
- The first axis in the command you select is aligned exactly to the axis of the first object you select.
- The directions in which the axes are oriented are based on the locations of the pick points on the reference objects. Each axis is oriented towards the end of the reference object nearest the pick point.
2. Multi-select a second object to which you want to align the second axis in the command.
- The second axis in the command that you select is aligned as closely as possible to the axis of the second reference object while still being perpendicular to the first axis set by the command. The remaining axis is made perpendicular to the other two axes.
3. From the **View** menu, point to **Coordinate System**, and then point to **Set Using Two Axes** and select the pair of axes you want (**X then Y Axis**, **X then Z Axis**, etc.). The coordinate system axes are set.
- Multiple user-defined coordinate systems can be saved and restored (See the *Save/Edit Coordinate System* entry in this chapter).

Set Weight...

Window: **Registration**

Menu: **Constraint**

Action: Opens dialog

Usage: The **Set Constraint Weight** dialog is used to set the relative weight of individual constraints used in registration.

- The weight value can be set from 0.0 to 1.0, with 0.0 indicating that the constraint has no effect on the registration. A constraint set to 0.5 has half the influence of a constraint set to 1.0.

Note: The practice of decreasing the weight of all constraints with higher than average error is not recommended as it can lead to lower overall registration accuracy. Decreasing the weight of a constraint should only be done if you have reason to believe that the measurement of one or both of the objects in the constraint is less accurate than other measured objects.

To set a constraint's weight:

1. In the **ScanWorlds' Constraints** tab of the **Registration** window, select the constraint you want to adjust.
2. From the **Constraint** menu, select **Set Weight....** The **Set Constraint Weight** dialog appears.
3. Enter a value between 0.0 and 1.0 in the **Weight** field, and then click **OK**. The weight is adjusted.

Shaded

Window: ControlSpace, ModelSpace, Registration, Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)

Menu: Point Cloud Rendering

Action: Toggles menu item ON/OFF

Usage: The **Shaded** command toggles the lighting of point clouds, similar to meshes.

- Only point clouds with valid surface normal vector estimates respond to the **Shaded** render mode.
- The **Point Cloud Rendering | Front** and **Point Cloud Rendering | Back** commands affect which side of the cloud is lit. **Front** shades points on surfaces facing the original scanner. **Back** shades points on surfaces facing away from the original scanner.

Shortcut

Window: Navigator

Menu: Create

Action: Executes command

Usage: The **Shortcut** command creates a shortcut to the selected object in the **Shortcuts** top-level folder.

- A shortcut provides quick access to often-used projects, ScanWorlds, ModelSpaces, and Registrations that may be several layers down in the database structure.

To create a shortcut:

1. In the **Navigator** window, select the object for which you need a shortcut.
2. From the **Create** menu, select **Shortcut**. The shortcut is added to the **Shortcuts** folder.

Shortcut in Windows

Window: Navigator

Menu: Create

Action: Displays submenu

Usage: The **Shortcut in Windows** command displays a submenu that creates a Windows shortcut to the selected object on the Windows Desktop, Start Menu, Quick Launch Bar or as a Windows Favorite. A shortcut provides quick access to often-used projects, Scan Worlds, ModelSpaces, and Registrations that may be several levels down in the database structure. When the shortcut is selected, *Cyclone* will launch, locate the target of the shortcut and open the appropriate dialog or viewer for it..

- Windows shortcuts created with this command are different from *Cyclone* shortcuts created in the *Cyclone* Navigator. Windows shortcuts may be placed anywhere within the Windows

operating system and function outside of *Cyclone* to launch *Cyclone* and browse to their target objects. *Cyclone* shortcuts are located only within the *Cyclone* Navigator and only function within *Cyclone*.

To create a shortcut in Windows

1. In the **Navigator** window, select the object to which you want to create a Windows shortcut.
2. From the **Create** menu, select **Shortcut in Windows** | [submenu command]. The shortcut is added to the indicated location.
 - The shortcut name is based on the name of the object the shortcut points to.
 - If there is already an existing shortcut with the same name, it will be overwritten with the new shortcut. You are prompted to confirm or cancel.

Show Active Plane

Window: ModelSpace
Menu: Tools | Reference Plane
Action: Toggles menu item ON/OFF
Usage: The **Show Active Plane** command toggles the display of the active Reference Plane.

Show All (Limit Box Manager)

Window: Limit Box Manager
Menu: View
Action: Executes command
Usage: The **Show All** command toggles visibility of limit boxes between only the activated limit box and all limit boxes..

....When all limit boxes are visible, the active limit box is displayed is highlighted with thick lines and limit box selected in the list is displayed with dashed lines

Show Annotation Keys

Window: ModelSpace
Menu: Tools | Annotations
Action: Toggles menu item ON/OFF
Usage: The **Show Annotation Keys** command toggles the display of annotation keys in the window. When checked, annotations keys are shown with the annotation values; when not checked, annotation keys are not shown, although the annotation values are still displayed.

Show Annotation Points

Window: Image Viewer

Menu: Annotation

Action: Toggles menu item ON/OFF

Usage: The **Show Annotation Points** command toggles the display of annotation points.

- When checked, the annotation points are displayed on the image.

Show Annotations

Window: Image Viewer, ModelSpace

Menu: Image Viewer: Annotation

ModelSpace: Tools | Annotations

Action: Toggles menu item ON/OFF

Usage: The **Show Annotations** command toggles the display of annotations. When checked, annotations are shown in the window; when not checked, annotations are not shown.

Note: The visibility of text added to the ModelSpace using *Cyclone*'s drawing tools is also toggled using this command.

Note: Use the [Add/Edit Annotations](#) command to make individual annotations visible; an annotation must be individually visible and the **Show Annotations** command must be ON for the annotation to actually be displayed.

Show Constraint

Window: Registration

Menu: Constraint

Action: Executes command

Usage: The **Show Constraint** command displays the objects involved in the selected constraint in the **Constraint Viewer** panes.

- For complete registration procedures, see the [Registration](#) entry in the *Getting Started* chapter.

To display a constraint:

- From either the **Constraint List** or the **ScanWorlds' Constraints** pane, select the constraint you want to view.
- From the **Constraint** menu, select **Show Constraint**. The objects involved in the constraint are displayed in the **Constraint Viewer** panes of the **Registration** window.
 - The objects involved in the constraint are selected and centered in each ModelSpace's pane.
 - When a cloud constraint is selected, the ControlSpaces referenced by the cloud constraint appear in the two Constraint viewers, and the two viewers are synchronized so that the view

navigation in one viewer is matched in the other viewer. All pick points from the cloud constraint appear as well. Note that any of these pick points are editable, and the **Update Initial Cloud Alignment** command can be used to modify the alignment.

Show Constraint Viewer 1 (2)

Window: **Registration**

Menu: **Viewers**

Action: Toggles menu item ON/OFF

Usage: The **Show Constraint Viewer 1** and **Show Constraint Viewer 2** commands toggle the display of the two Constraint viewer windows.

Show Coordinate System Axes

Window: **ModelSpace**

Menu: **View | Coordinate System**

Action: Toggles menu item ON/OFF

Usage: The **Show Coordinate System Axes** command toggles the visibility of the coordinate axes in the ModelSpace viewer.

- When checked, the coordinate system axes are displayed.
- Depending on the coordinate system and viewpoint, the origin may or may not be visible. The axes may also be hidden behind objects.
- The coordinate axes X, Y, and Z are always displayed in red, green, and blue respectively.

Show Current Transform

Window: **Registration**

Menu: **ScanWorld**

Action: Opens dialog

Usage: The **Show Current Transform** dialog displays the current transform for the selected ScanWorlds as computed by the latest registration.

- The transform is a rigid six degrees of freedom transformation consisting of three rotation degrees of freedom (roll, pitch, and yaw) and three translation degrees of freedom (X, Y, Z offset).

To show the current transform

1. From the **ScanWorlds' Constraints** or the **ModelSpaces** tab, select the ScanWorlds whose transforms you want to view.
2. From the **ScanWorld** menu, select **Show Current Transform**. The **Show Current**

Transform dialog appears, displaying the current transform.

Show Datum

Window: ModelSpace

Menu: Tools | Measure | Datum

Action: Toggles menu item ON/OFF

Usage: The **Show Datum** command toggles the visibility of the current datum point.

- The datum is represented visually by a callout line and a “Datum” label at the reference point of the datum.

Show Diagnostics...

Window: Registration

Menu: Cloud Constraint

Action: Opens dialog

Usage: The **Cloud Constraints Diagnostics** dialog is used to view alignment accuracy data for the selected cloud constraint.

- For more information on the cloud registration process, see the [Cloud Registration](#) entry in the *Registration* chapter.

Cloud Constraint Diagnostics Fields

CLOUD/MESH

The name of the constraint.

TRANSLATION

The translation vector for the alignment of the second cloud onto the first cloud.

ROTATION

The rotation axis and angle for the alignment of the second cloud onto the first cloud.

OBJECTIVE FUNCTION VALUE

The value of the overlap error function being minimized during alignment.

OVERLAP POINT COUNT

The number of overlapping points between the two ScanWorlds.

RMS

The root mean square value of the absolute errors of between overlapping points.

AVG

The average value of the absolute errors between overlapping points.

MIN

The minimum value of the absolute errors between overlapping points.

MAX

The maximum value of the absolute errors.

STD DEV

The standard deviation of the absolute errors between overlapping points.

OVERLAP CENTER

An estimate of the center of mass of all the overlapping points in the coordinate system of the first ScanWorld

FILTER PARAMETERS

The user preferences used during alignment.

STOPPING CRITERIA

The user preferences used during alignment.

To view cloud constraint diagnostics:

1. From the **Constraint List** tab, select the cloud constraints you want to inspect.
2. From the **Cloud Constraint** menu, select **Show Diagnostics....** The **Cloud Constraints Diagnostics** dialog appears displaying alignment accuracy data (see the following descriptions).
 - If more than one alignment has been performed, accuracy data is displayed for each alignment, starting from the most recent.
 - To view a temporary ModelSpace containing the overlapping points, click **Show Overlap**. From the first ScanWorld, overlapping points are colored blue and non-overlapping points are colored dark blue. From the second ScanWorld, overlapping points are colored green and non-overlapping points are colored dark green.
 - To save the diagnostics as a text file, click **Save...** and use the dialog that appears to specify a name and location for the text file.
 - To print the diagnostic, click **Print....** A standard print dialog launches.
 - To view a graphical histogram of the errors of the overlapping clouds with lines indicating user preferences, click **Histogram**. See the *Error Histogram* entry in the *Registration* chapter for details.

Show Grid

Window: **Scan Control**

Menu: **Image**

Action: Toggles menu item ON/OFF

Usage: The **Show Grid** command toggles the display of the targeting grid in the **Scan Control** window.

Show Handles

Window: ModelSpace

Menu: Edit Object | Handles

Action: Toggles menu item ON/OFF

Usage: The **Show Handles** command toggles the visibility of object handles.

- Handles allow you to manipulate an object's position and shape in various ways.
- For more information on using handles, see the [Handles](#) entry in the *Modeling* chapter.

Show Hidden Clouds

Window: ModelSpace

Menu: View

Action: Executes command

Usage: The **Show Hidden Clouds** command displays cloud points that have been hidden using the **Hide Selected Clouds** or **Hide Unselected Clouds** command..

Show Images

Window: Scan Control

Menu: Image

Action: Executes command

Usage: The **Show Images** command toggles visibility of camera images in the **Scan Control** window.

- Having a valid image in the **Scan Control** window is not required for targeting the scanner or for scanning.

Show Incoming Points

Window: ModelSpace

Menu: Tools | Scanner

Action: Toggles menu item ON/OFF

Usage: The **Show Incoming Points** command toggles the display of scan data as they are captured.

- It can be helpful to toggle this command OFF if modeling is being done in the same ModelSpace viewer.

Show ModelSpace

Window: Scan Control

Menu: Scanner Control

Action: Toggles menu item ON/OFF

Usage: The **Show ModelSpace** command toggles the display of the destination ModelSpace in the active **Scan Control** window.

- When ON, any changes are displayed in the active **Scan Control** window when the window is refreshed. The ModelSpace is displayed from the scanner's point of view in that ModelSpace.
- In order to use this command, the destination ModelSpace viewer must be opened. (See the [Open ModelSpace Viewer](#) command)

Show Object's Cloud

Window: ModelSpace

Menu: View

Action: Toggles menu item ON/OFF

Usage: The **Show Object's Cloud** command toggles the display of the point cloud(s) that were used to create the selected object.

- Viewing an object's points provides access to the points for visual comparison or verification.
- When an object or its source points have changed significantly, this command is not available.

To view an object's point cloud:

1. Select the object whose point cloud(s) you want to make visible.
 2. From the View menu, select Show Object's Cloud.
- If you want to be able to select and edit a copy of the point cloud used to create an object, use the **Insert Copy of Object's Points** command instead.

Show Output Box

Window: ModelSpace

Menu: View

Action: Toggles menu item ON/OFF

Usage: When **Show Output Box** is selected, the Output Box is displayed until closed by the user.

- Each new measurement is recorded consecutively in the Output Box; the data in this dialog is easily copied and pasted into other applications.
- Selecting **Show Output Box** automatically turns the **Enable Output Box** option ON.

Note: The Output Box can be docked to the side of the **ModelSpace** window.

Show Pick Points

Window: **ModelSpace**
Menu: **Selection**
Action: Toggles menu item ON/OFF
Usage: The **Show Pick Points** command toggles the display of pick points as white points.

- The points can be used later to create vertices, line segments, polylines, arcs, and splines.
- Pick points need not be visible to be used by other commands.

Show Pick Polyline

Window: **ModelSpace**
Menu: **Selection**
Action: Toggles menu item ON/OFF
Usage: The **Show Picked Polyline** command toggles the display of a polyline along the current series of pick points.

- This command does not create a polyline. It displays the order of the pick points.
- This may be useful for keeping track of the order of picks for operations like creating a cubic spline or a polyline from pick points.

Show Point Clouds

Window: **Registration**
Menu: **Viewers**
Action: Toggles menu item ON/OFF
Usage: The **Show Point Clouds** command toggles the display of point clouds in the Constraint viewers in the Registration viewer.

Show Rotation Handles

Window: ModelSpace

Menu: Edit Object | Handles

Action: Toggles menu item ON/OFF

Usage: The **Show Rotation Handles** command toggles the display and availability of object rotation handles.

- Rotation handles allow you to rotate an object around any of its principal axes.
- For more information on using handles, see the [Handles](#) entry in the *Modeling* chapter.

Show Scanners

Window: Navigator

Menu: View

Action: Toggles menu item ON/OFF

Usage: The **Show Scanners** command toggles the display of the Scanners folder in the **Navigator**.

Show Servers

Window: Navigator

Menu: View

Action: Toggles menu item ON/OFF

Usage: The **Show Servers** command toggles the display of the Servers folder in the **Navigator**.

Show Shortcuts

Window: Navigator

Menu: View

Action: Toggles menu item ON/OFF

Usage: The **Show Shortcuts** command toggles the display of the Shortcuts folder in the **Navigator**.

Show Table Matches

Window: ModelSpace

Menu: Create Object | Fit to Cloud Options

Action: Toggles menu item ON/OFF

Usage: The **Show Table Matches** command enables/disables the use of parts tables when

fitting objects.

- In order to use parts tables when fitting, **Catalog/Parts Table** and **Table** must be selected within the **Object Preferences** dialog. See the *Object Preferences* entry in this chapter.
- For an overview of fitting objects in *Cyclone*, see the [Modeling](#) entry in the *Getting Started* chapter.

Show Traverse Report...

Window: Registration

Menu: Registration

Action: Opens dialog

Usage: The **Traverse Report** dialog is used to view the parameters and accuracy of a Registration created through Traverse.

- For more information on the traverse process, see the [Traverse \(ScanStation, ScanStation 2\)](#) entry in the *Scanning with the ScanStation, ScanStation 2, and HDS3000* chapter, or the *Traverse (HDS6000)* section in the *Scanning with the HDS6000 and HDS4500* chapter.

To view the traverse report:

1. From the **Registration** menu, select **Show Traverse Report...**. The **Traverse Report** dialog appears, displaying the parameters and quality of the traverse.

Silhouette

Window: ControlSpace, ModelSpace, Registration, Scan Control (ScanStation, ScanStation 2, HDS3000, HDS4500, HDS6000)

Menu: Point Cloud Rendering

Action: Toggles menu item ON/OFF

Usage: The **Silhouette** command affects the transparency of each cloud point based on the orientation of that point's estimated surface relative to the viewer, such that outlines of surfaces inherent in the point cloud are revealed..

- Only point clouds with valid surface normal vector estimates respond to the **One-Sided** render mode.
- Points that are on estimated surfaces facing the viewer are opaque. Points gradually become more transparent as the estimated surface is oriented to be perpendicular to the viewer.

Slice

Window: ModelSpace

Menu: Create Object

Action: Displays submenu

Usage: Pointing to **Slice** displays submenu commands to cut selected object(s) using a patch, ref plane, or all selected objects.

The following commands are available from the **Slice** submenu:

[by Last Selection](#), [by Ref Plane](#), [All in Selection](#)

Snap to Grid

Window: ModelSpace

Menu: Edit Object | Snapping Grid

Action: Toggles menu item ON/OFF

Usage: The **Snap to Grid** command toggles the snapping grid ON/OFF.

- The snapping grid is used to control object movement when using handles.
- For 2D drawing, the snapping grid is aligned to the active Reference Plane's origin and axes.
- For object manipulation, the snapping is an invisible 3D grid aligned with the coordinate system and spaced by the given spacing relative to the origin, along all three axes.
- When ON, objects/vertices jump to the next grid increment as you move them/draw them, and objects jump to the next interval when rotated using rotation handles. When OFF, objects move independently of grid settings.
- To adjust grid spacing, see the [Set Grid Spacing](#) command.
- To adjust rotational spacing, see the [Set Rotational Spacing](#) command.

Snapping Grid

Window: ModelSpace

Menu: Edit Object

Action: Displays submenu

Usage: Pointing to **Snapping Grid** displays submenu commands used to toggle the snapping grid ON/OFF and edit grid settings.

The following commands are available from the **Snapping Grid** submenu:

[Set Grid Spacing...](#), **[Set Rotational Spacing...](#)**, **[Snap to Grid](#)**

- The snapping grid is used to control object movement when using handles.
- For 2D drawing, the snapping grid is aligned to the active Reference Plane's origin and axes.
- For object manipulation, the snapping is an invisible 3D grid aligned with the active coordinate system and spaced by the given spacing in all three directions.
- When ON, objects/vertices jump to the next grid increment as you move them/draw them, and objects jump to the next interval when rotated using its rotation handle. When OFF, objects move independently of grid settings.

Snapshot...

Window: **ModelSpace, Scan Control, Image Viewer**

Menu: **File**

Action: Opens dialog

Usage: The **Save Snapshot to File** dialog is used to save an image of the current ModelSpace, Scan Control, or Image.

To save a snapshot of the current window:

1. From the **File** menu, select **Snapshot....** The **Save Snapshot to File** dialog appears.
2. Select the file format you want from the **Save as** type box.
3. Navigate to the desired destination directory, type a name for the file, and then click **Save**. Your image is saved.

Sphere Target

Window: **ModelSpace**

Menu: **Create Object | Fit to Cloud**

Action: Executes command

Usage: The **Sphere Target** command fits a 6" diameter sphere target to the selected point cloud. When the sphere target is created, it is added automatically to the ControlSpace as well.

To fit a sphere target:

1. Pick the point cloud near the center of the area to which you want to fit a sphere target.
 - It is assumed that the pick is on the surface of the target.
 - It is not necessary to segment the point cloud prior to fitting.

2. From the **Create Object** menu, point to **Fit to Cloud**, and then select **Sphere Target**. A sphere is fit to the spherical target.

Stakeout

Window: Scan Control (ScanStation, ScanStation 2, HDS6000)

Menu: Window

Action: Expands Tab

Usage: The **Stakeout** command expands the **Stakeout** tab in the Scanner Control Panel.

- The Stakeout procedure requires Field Setup to first be applied to establish the scanner's coordinate system.
- The Stakeout procedure is optimally executed with at least two personnel: one person to operate the scanner, and one person to set up and move the target until it is over the stakeout point.

Stakeout Tab Options

METHOD

Displays the current method used to determine the position of a specified point.

ORTHOGONAL FROM STATION

Select a known coordinate in the **Target ID** field. This known coordinate is the stakeout point.

MANUAL – POLAR

Enter the **Azimuth** and horizontal **Distance** to the stakeout point. The elevation of the stakeout point is relative to the HI measurement mark on the scanner.

MANUAL – ORTHOGONAL

Enter the coordinates of the stakeout point.

TARGET ID

Selects the target settings to display in the panel.

HT

The height of the target above the stakeout point.

AZIMUTH

The horizontal angle towards the stakeout point.

X Y Z / N E EL

The position of the stakeout point.

TYPE

The type of the target to be acquired.

RODMAN INSTRUCTIONS

Displays the instructions to relay to the rodman to reposition the target based on the target's current position.

D CROSS

The distance to move the target left or right (facing the scanner) from its current position.

D LENGTH

The distance to move the target towards or away from the scanner.

D HEIGHT

The distance to move the target up or down from its current position.

COARSE TARGET SCAN

Performs a coarse scan around the current pick point to pick up at least one point on the target..

ACQUIRE TARGET

Acquires the target at the current pick point. The Rodman Instructions are updated to reflect the new offsets.

To perform a stakeout:

13. Connect to the scanner (ScanStation, ScanStation 2, or HDS6000) and ensure that it is level.
14. Perform a Field Setup to establish the scanner's coordinate system. The azimuth and coordinates of the target point are relative to this coordinate system.
15. If using the **Orthogonal from Station** method, import or add the known or assumed coordinate of the stakeout point, using **Add/Replace Coordinate**, **Import Coordinate List**, or **Import Coordinate List (Leica System 1200)**.
16. From the **Window** menu, select **Stakeout**. The **Stakeout** panel appears in the control panel.
17. Select the stakeout **Method**.
18. Enter the **Target ID**. For the **Orthogonal from Station** method, the **Target ID** must match a known coordinate. The **Target ID** is also applied to the acquired target.
19. Enter the **HT** of the target relative to the stakeout point.
20. Select the **Type** of the target to be acquired to establish the offsets to the stakeout point.
21. Enter the position of the stakeout point.
 - If using the **Orthogonal from Station** method, the stakeout coordinates are already known.
 - If using the **Manual – Polar** method, enter the **Azimuth** and **Distance** to the stakeout point.
 - If using the **Manual – Orthogonal** method, enter the 3D coordinate of the stakeout point.
22. Set up the target in the general area of the stakeout point.
23. Perform a preliminary scan using the primary **Scan** command.
24. Pick a point in the preliminary scan that is close to the target. Click **Coarse Target Scan** to perform a coarse scan to acquire at least one point on the target. Repeat as needed.

25. Pick a point in the coarse scan that is on the target. Click **Acquire Target** to perform a fine scan on the target, followed by a target fit. The **Rodman Instructions**, **d Cross**, **d Length**, and **d Height** are updated based on the target's current position relative to the stakeout point.
26. Move the target in the indicated directions by the indicated offsets. Repeat the process as needed until the target is over the stakeout point.

Standard Viewpoints

Window: ModelSpace

Menu: Viewpoint

Action: Displays submenu

Usage: Pointing to **Standard Viewpoints** displays submenu commands used to view your ModelSpace from standard viewpoints including: top, bottom, left, right, front, back, isometric, and right isometric.

The following commands are available from the **Standard Viewpoints** submenu:

Top, Bottom, Left, Right, Front, Back, [Isometric](#), [Right Isometric](#)

- The **Standard Viewpoints** submenu commands are self-explanatory. Also see the [Isometric](#) and [Right Isometric](#) commands.

Station Data (HDS3000)

Window: Scan Control

Menu: Window

Action: Expands Tab

Usage: The **Station Data** command expands the **Station Data** tab in the Scanner Control Panel.

Station Data Tab Options

HOME SCANWORLD

Select a Home ScanWorld (optional) from the dropdown. This ScanWorld is the repository of targets that can be selected in Station ID and in the **Set Horizontal from Target** dialog. This ScanWorld also receives any new targets of known coordinates introduced during the geo-referencing process.

STATION ID

Enter an identifier for the station position or select an existing identifier from the dropdown.

HI

Enter the measured height of the scanner above a known point.

- The scanner has a turret on the body to which a height is measured, from the ground point over which the scanner is centered.

X, Y, Z

Enter coordinates of the ground point.

SET HZ BUTTON

Opens the **Aim Scanner** dialog (used to set scanner horizontal/compass direction).

HZ FROM TARGET BUTTON

Opens the **Set Horizontal from Target** dialog.

APPLY BUTTON

Applies settings to the current ScanWorld's default coordinate system.

DEFAULT BUTTON

Resets all settings to defaults.

Steel Section

Window: ModelSpace

Menu: Create Object | Fit to Cloud

Action: Displays submenu

Usage: Pointing to **Steel Section** displays submenu commands used to fit a specified steel section to a picked point cloud.

The following steel shapes are available from the **Steel Section** submenu:

Angle, Channel, Tee, Rectangular Tube, Wide Flange

- For information on preparing point clouds for modeling, see the [Segmentation](#) entry in the *Modeling* chapter.
- For information on using parts tables, fit tolerances, and fit constraints, see the [Object Preferences](#) command and the [Modeling](#) entry in the *Getting Started* chapter.

To fit a steel section to the selected point cloud:

1. From the **Object Preferences** dialog, enable and set any desired fit constraints, fit tolerances, or parts tables for the steel type that you want to fit. See the *Object Preferences* entry in this chapter for details. If no preferences are set, the selected steel shape is fit directly to the selected cloud.
- To enable fit constraints, fit tolerances, or parts tables: From the **Create Object** menu, point to **Fit to Cloud Options**, and toggle **Show Table Matches**, and/or **Check Fit Tolerances** ON.
2. Using one of the methods described below, select one, two, or three points on the cloud of points you want to use to fit a steel section.

- All picks must be on major faces: the tops and bottoms of flanges and the sides of webs. Picks should not be on the edges of flanges or the top or bottom of webs or on the exposed cross section.

Method 1: Pick a point anywhere on the interior (not near the edge) of a face.

Method 2: Pick two points anywhere on the interior (not near the edge) of two perpendicular faces.

Method 3: Pick two points on the same face widely spaced and approximately along the steel section's axis.

Method 4: Pick three points on the same face. The first two points should be picked as in Method 2 above, and the third point can be anywhere on the same face provided that it is not in a line with the first two picks.

Note: For tee shapes whose orientation is ambiguous, the first pick is assumed to be on the web.

- When multiple point clouds are selected, the **Multiple Point Cloud Style** setting in the **Fitting** tab of the **Edit Preferences** dialog determines the results. If **Fit Single Object** (default) is selected, the clouds are first merged and then a single object is fit to the resulting merged point cloud. If **Fit Multiple Objects** is selected, an object is fit to each cloud individually.
3. From the **Create Object** menu, point to **Fit to Cloud**, and then point to **Steel Section** and select the type of steel section you want to fit. Unless you are fitting to a parts table item, the steel shape appears.
 - If the point cloud varies too much from the shape of the object you are trying to create, a **Fit Quality** warning message appears and gives you an option to keep or cancel the fit (see the *Fit Tolerances* area of the *Object Preferences* entry in this chapter). In this case it may help to further refine the point cloud (see the *Cloud Segmentation* entry in the *Modeling* chapter) and retry the fit.
 4. If parts tables are enabled, the **Select Table Item** dialog appears displaying possible matches. The best match is temporarily drawn in the ModelSpace and marked with an asterisk in the dialog.
 - Matches within the set range are marked with a “+” sign.
 - The **vs Table** column displays the RMS (root mean squared) error of the object's dimensions (from the initial fit) compared to the dimension in the table. The **vs Cloud** column displays the standard deviation of point errors relative to the object best-fit using the individual table item's dimensions. The remaining columns indicate the per-dimension difference versus the initial fit's dimensions.
 - Any enabled fitting constraints corresponding to parameters from the table are used in the initial fitting, but then the closest match in the table is used when creating the actual object.
 - To show matches from a different table, select a new table from the **Table** field, and then click **Refresh**. New potential matches are shown. Clicking **Refresh** without changing the table calculates any missing **vs Cloud** values.
 - To edit the range of potential matches, enter a new value in the **Match Table Items Within** field, and then click **Refresh**. The new range of potential matches is shown.

5. Select the appropriate part from the list and click **OK**. The select parts table item is fit.

Subtract From Patch

Window: ModelSpace

Menu: Edit Object | Patch

Action: Executes command

Usage: The **Subtract from Patch** command is used to create a hole in a selected patch.

To create a hole in a patch:

1. Draw a fence, or draw and select a polyline or polygon in the shape of the hole you wish to create.
 - The hole's edges cannot be self-intersecting and cannot cross the boundary of the patch. Also, holes cannot intersect each other.
2. Select the patch in which you want to create a hole.
3. From the **Edit Object** menu, point to **Patch**, and then select **Subtract**. The hole is created.

Surface Area

Window: ModelSpace

Menu: Tools | Measure

Action: Executes command

Usage: The **Surface Area** command calculates and displays the total surface area of the selected object.

To measure surface area:

1. Select the object you want to measure.
2. From the **Tools** menu, point to **Measure**, and then select **Surface Area**. The measurements results are displayed in square units. The measurement results are also displayed in the Output Box, if it is enabled.

Surface Deviation

Window: ModelSpace

Menu: Tools | Measure

Action: Opens dialog

Usage: The **Surface Deviation** command displays a dialog used to specify what output is generated after measuring the differences in elevation between a TIN mesh and a reference surface.

- The deviation is measured along the axis perpendicular to the active reference plane. The reference plane must be perpendicular to one of the major axes of the current UCS. The

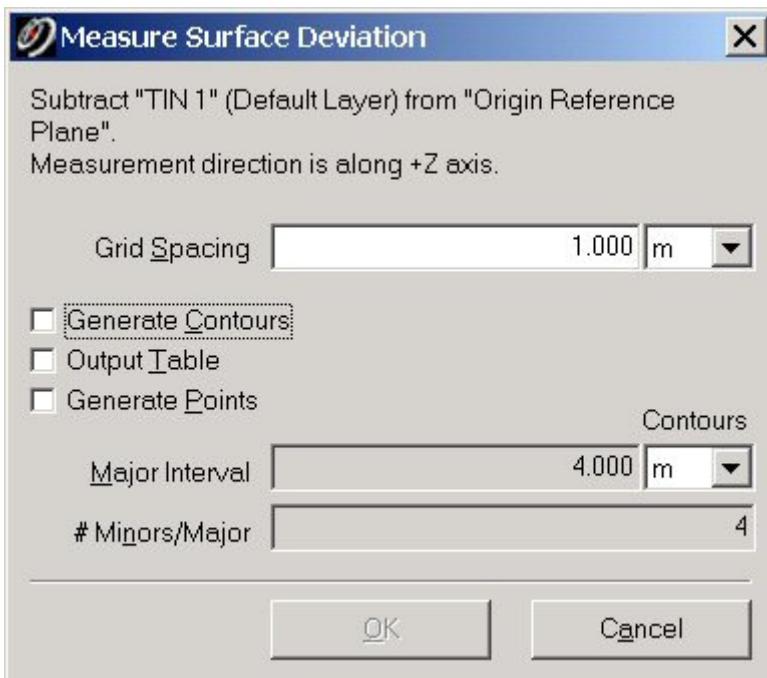
mesh(es) must be TIN relative to the reference plane.

- The reference surface may be the second TIN mesh selected, or the active reference plane if only one TIN mesh is selected.
- If a polyline/polygon is also selected, only the area within the polyline/polygon (projected onto the active reference plane) is measured.

To measure surface deviation:

1. Select the TIN whose surface is to be subtracted from the reference surface.
 - To measure surface deviation against the active reference plane, select only one TIN.
 - To measure surface deviation against another reference TIN, select a second TIN surface to act as the reference surface.
 - To measure surface deviation within a specific boundary, multi-select a polyline or polygon object.
2. From the **Tools** menu in the **ModelSpace** window, point to **Measure**, then select **Surface Deviation...** The [**Measure Surface Deviation**](#) dialog appears.
3. Change the **Grid Spacing** of the active reference plane. The deviations will be measured on the reference plane grid with this spacing.
4. Select the form of the measurement output to be generated.
 - **Generate Contours:** check this to generate contours representing the cut/fill regions. Set the **Major Interval** and **# Minors/Major** parameters. The contour elevations represent the differences in elevation in the current UCS.
 - **Output Table:** check this to display the **Measure Surface Deviation Results** dialog, which contains a table listing the individual deviation measurements. This measurement cannot be saved if this option is not selected.
 - **Generate Points:** check this to generate points whose coordinates and elevation in the current UCS correspond to the individual deviation measurements.
5. Click **OK**. The surface deviation is computed and the requested output is generated.

Measure Surface Deviation Dialog



- Grid Spacing:** Change the **Grid Spacing** of the active reference plane. The deviations will be measured on the reference plane grid with this spacing.
- Generate Contours:** Select this check box to generate contours representing the cut/fill regions. Set the **Major Interval** and **# Minors/Major** parameters. The contour elevations represent the differences in elevation in the current UCS.
- OutputTable:** Select this check box to display the **Measure Surface Deviation Results** dialog, which contains a table listing the individual deviation measurements. This measurement cannot be saved if this option is not selected.
- Generate Points:** Select this check box to generate points whose coordinates and elevation in the current UCS correspond to the individual deviation measurements.
- Contours:** If you select the **Generate Contours** check box to generate contours representing the cut/fill regions, set the **Major Interval** and **# Minors/Major** parameters. The contour elevations represent the differences in elevation in the current UCS.

Switch Alignment Start/End

- Window:** ModelSpace
- Menu:** Tools | Alignment and Section
- Action:** Executes command
- Usage:** The **Switch Alignment Start/End** is used to switch the assignment of the start station from one end of the selected alignment to the other.

- For more information about alignments and the sections process, see the [Sections Manager](#) chapter.

Synchronize Viewers

Window: Registration

Menu: Viewers

Action: Toggles menu item ON/OFF

Usage: The **Synchronize Viewers** command toggles the synchronization of the two Constraint viewers for simultaneous view navigation.

- When ON, viewpoint changes in one Constraint viewer are tracked in the second viewer. This is useful when adding pick point hints to create or update the initial alignment hints for cloud constraints.

Target All

Window: Scan Control

Menu: Scanner Control

Action: Executes command

Usage: The **Target All** command targets the entire image in the **Scan Control** window.

- Using the **Target All** command allows you to target the whole field of view in the **Scan Control** window without having to draw a target box manually.

To target the whole field of view in the Scan Control window:

1. From the **Scanner Control** menu, select **Target All**. (or click the **Target All** icon). The scan target appears in the **Scan Control** window.

Texture Map Browser

Window: ModelSpace

Menu: Edit Object | Appearance

Action: Opens dialog

Usage: The **Texture Map Browser** invokes the **Texture Map Browser** dialog, which provides access to the texture maps of the selected point cloud or mesh.

- For information on the texture mapping process, see the [Texture Mapping](#) chapter.

Texture Map Browser (Scan Control)

Window: Scan Control (ScanStation, ScanStation 2, HDS2500, HDS3000, HDS4500, HDS6000)

Menu: Project

Action: Opens dialog

Usage: The **Texture Map Browser** invokes the **Texture Map Browser** dialog, which

provides access to the texture maps of the selected point cloud or mesh, in the attached ModelSpace viewer (launching the viewer is needed).

- For information on the texture mapping process, see the [Texture Mapping](#) chapter.

Tilt

Window: ModelSpace

Menu: Tools | Reference Plane

Action: Toggles function ON/OFF

Usage: The **Tilt** command toggles the tilt function in **Drawing** mode, which allows you to interactively tilt the active Reference Plane around one of its major in-plane axes.

To tilt the Reference Plane:

1. From the **Tools** menu, point to **Reference Plane**, and then select **Tilt** (or click the **Tilt** icon). You are now in **Drawing** mode. The cursor changes to the **Drawing Tool** cursor and the **Tilt** function is ON.
2. In the ModelSpace, drag the **Drawing Tool** cursor to tilt the active Reference Plane.
 - The initial click in the ModelSpace determines which axis is tilted. Click close to the axis you want to tilt. The active Reference Plane is tilted around the other major axis.
3. Release the mouse button. The Reference Plane is tilted.

TIN Volume

Window: ModelSpace

Menu: Tools | Measure

Action: Opens dialog

Usage: The **TIN Volume** command displays a dialog that indicates how the prismatic volume between a TIN mesh and a reference surface will be computed.

- The volume is measured along the axis perpendicular to the active reference plane. The reference plane must be perpendicular to one of the major axes of the current UCS. The mesh(es) must be TIN relative to the reference plane.
- The reference surface may be the second TIN mesh selected, or the active reference plane if only one TIN mesh is selected.
- If a polyline/polygon is also selected, only the area within the polyline/polygon (projected onto the active reference plane) is measured.

To measure TIN Volume:

1. Select the TIN whose surface is to be subtracted from the reference surface.
 - To measure TIN Volume against the active reference plane, select only one TIN.
 - To measure TIN Volume against another reference TIN, select a second TIN surface to act as the reference surface.
 - To measure TIN Volume within a specific boundary, multi-select a polyline or polygon object.

2. From the **Tools** menu in the **ModelSpace** window, point to **Measure**, then select **TIN Volume**.... The **Measure TIN Volume** dialog appears.
3. Click **OK**. The prismoidal TIN volume is computed.
- The “Cut” (regions where the TIN is “above” the reference surface) and “Fill” (regions where the TIN is “below” the reference surface) are indicated graphically.

Toggle ScanWorld Leveled

Window: Registration

Menu: ScanWorld

Action: Toggles ScanWorld property

Usage: The **Toggle ScanWorld Leveled** command toggles the “leveled” property of the selected ScanWorld. A ScanWorld known to be “leveled” is registered such that its “up” vector remains the same. This command should be used with care, since an incorrect property may adversely affect the registration results.

- The name of a ScanWorld known to be leveled is followed by “(Leveled)” in the Registration viewer.

To toggle the leveled property of a ScanWorld:

1. In the **ScanWorlds’ Constraints** or **ModelSpaces** tab, select one or more ScanWorlds.
2. From the **ScanWorld** menu, select **Toggle ScanWorld Leveled**. The property is toggled for each selected ScanWorld.

Traverse

Window: Scan Control (ScanStation, ScanStation 2, HDS6000)

Menu: Window

Action: Expands Tab

Usage: The **Traverse** command expands the **Traverse** tab in the Scanner Control Panel.

- For an overview of the Traverse process, see the [Traverse \(ScanStation, ScanStation 2\)](#) section in the *Scanning with the ScanStation, ScanStation 2, and HDS3000* chapter, or the *Traverse (HDS6000)* section in the *Scanning with the HDS6000 and HDS4500* chapter.
- The Traverse procedure requires an enabled dual-axis compensator (ScanStation, ScanStation 2) or enabled tilt sensor (HDS6000). The scanner must be close to level.

Traverse Tab Options

REGISTRATION

Displays the current Registration that represents the Traverse.

OPTIONS

Displays a pop-up menu with additional Traverse operations.

RECHECK BACKSIGHT...

Re-acquires the backsight target to see if the scanner has drifted since the backsight was first acquired.

REPORT...

Displays the **Traverse Report** dialog, which displays the parameters and quality of the traverse.

DISPLAY...

Launches the **ModelSpace for Traverse** ModelSpace viewer with a top-down view of the current Traverse.

APPEND CURRENT STATION

Re-opens a closed Traverse and adds the current ScanWorld to the Traverse. This can be used if the Traverse is closed prematurely.

MODIFY CURRENT STATION...

Displays the **Modify Traverse Info** dialog, which is used to edit existing Station, Backsight, and Foresight settings.

REMOVE CURRENT STATION

Removes the current ScanWorld from the Traverse. This can be used if the ScanWorld has been erroneously added to the Traverse.

CLOSE WITH LAST STATION

Closes the Traverse using the last ScanWorld added to the Traverse.

OPEN REGISTRATION...

Launches the Registration viewer for the current Traverse.

<, STATION / BACKSIGHT / FORESIGHT, >

Selects the target settings to display in the panel.

ID

The ID of the Station or Target (Backsight or Foresight). The ID is used to match against known, assumed, or acquired coordinates during Traverse.

HI / HT

The Height of Instrument (Station) or Height of Target (Backsight or Foresight).

X Y Z / N E EL

The position of the known or assumed coordinate with the matching ID in the **Known Coordinates** ScanWorld.

TYPE

The type of the target to be acquired.

ACQUIRE

Acquires the target (Backsight or Foresight) at the current pick point.

ADD

Adds the ScanWorld to the Traverse. The Station, Foresight, and Backsight settings must be entered and the Foresight and Backsight targets must be in the ScanWorld's ControlSpace.

CLOSE

Adds the ScanWorld to the Traverse and then closes the current Traverse. The Station, Foresight, and Backsight settings must be entered and the Foresight and Backsight targets must be in the ScanWorld's ControlSpace.

To perform a traverse:

1. Set up the scanner over a known point. Set up one target over another known point and another target over the next scanner position.
2. Connect to the ScanStation/ScanStation 2 and enable the dual-axis compensator, or connect to the HDS6000 and enable the tilt sensor.
3. Import or add the known or assumed coordinates, using **Add/Replace Coordinate**, **Import Coordinate List**, or **Import Coordinate List (Leica System 1200)**.
4. From the **Window** menu, select **Traverse**. The **Traverse** panel appears in the control panel.
5. Select **Station** in the pull-down.
6. Select the station **ID** from the list.
7. Measure and enter the **HI**.
8. Select **Backsight** in the pull-down, or push the **>** button to advance to the **Backsight** entry.
9. Select the backsight **ID** from the list.
10. Measure and enter the **HT**.
11. Select the target **Type**.
12. Pick a cloud point on the target over the known point and push **Acquire**. The backsight is acquired. If the target already exists in the ControlSpace, the target need not be re-acquired.
13. Select **Foresight** in the pull-down, or push the **>** button to advance to the **Foresight** entry.
14. Select the foresight **ID** from the list, or enter a new **ID**.
15. Measure and enter the **HT**.
16. Select the target **Type**.
17. Pick a cloud point on the target over the known point and push **Acquire**. The foresight is acquired. If the target already exists in the ControlSpace, the target need not be re-acquired.
18. Push the **Add** button. The ScanWorld is added to the Traverse.
 - To end the traverse, push the **Close** button instead. The foresight ID must be that of a known coordinate.
 - If you want to close the traverse but push **Add** instead of **Close**, click on the options button next to the **Registration** field and select **Close with Last Station** from the pop-up menu.
 - To edit the settings for this position, click on the options button next to the Registration field and select **Modify Current Station**.
19. Move the scanner to the next position, advance to the next ScanWorld, and repeat. The previous station ID is now the backsight ID, and the previous foresight ID is now the station ID.

- If you have closed the traverse already but need to add another station to the traverse and continue the traverse, click on the options button next to the **Registration** field and select **Append Current Station** from the pop-up menu.

Trim Edges

Window: ModelSpace

Menu: Create Object | Segment Cloud

Action: Executes command

Usage: The **Trim Edges** command creates a new segment of the selected point cloud that consists of all the points near the cloud's boundary.

- For an overview of the segmentation process, see the [Segmentation](#) entry in the *Modeling* chapter.

To trim the edges of the selected cloud:

1. Select the cloud whose edges you want to trim.
2. From the **Create Object** menu, point to **Segment Cloud**, and then select **Trim Edges**. The points along the boundary of the cloud are segmented.

Turn Viewpoint

Window: ModelSpace

Menu: Viewpoint

Action: Displays submenu

Usage: Pointing to **Turn Viewpoint** displays submenu commands used to rotate your ModelSpace View 90 degrees to the left, right, up, or down, changing the point of focus.

- To move your *viewpoint* 90 degrees to the left, right, up, or down while maintaining your point of focus, see the [Move Viewpoint](#) command.

The following commands are available from the **Turn Viewpoint** submenu:

Left, Right, Up, Down

Undo

Window: ModelSpace, Navigator, Registration, Scan Control, Image Viewer

Menu: Edit

Action: Executes command

Usage: The **Undo** command reverses the most recent action.

- Most user actions in *Cyclone* can be reversed once performed (there are some exceptions).
- *Cyclone* provides for theoretically unlimited undo events. An undo event is possible after you perform an action or provide some input that makes a change in the object database.
- Once you've undone an action, you can move forward in the history and redo the action (see the [Redo](#) command), until there are no more actions to redo (you get back to where you started undoing). However, if you undo, and then perform a new action instead of a redo, all events that have not been redone are discarded.
- Due to *Cyclone*'s concurrent modeling feature, someone else may change an object in your ModelSpace, which may break your chain of undo and redo events if the events affect a common object.

Unextrude

Window: **ModelSpace**

Menu: **Edit Object | Extrude**

Action: Executes command

Usage: The **Unextrude** command removes the selected extrusion, leaving only the original patch.

To unextrude an extrusion:

1. Select the extrusion you want to unextrude.
- It does not matter which side of the extrusion is picked - the original patch is always the result.
2. From the **Edit Object** menu, point to **Extrude**, and then select **Unextrude**. The extrusion is deleted, leaving only the original patch.

Unfreeze Registration

Window: **Registration**

Menu: **Registration**

Action: Executes command

Usage: The **Unfreeze Registration** command allows constraints in a previously frozen registration to be modified.

- After modifying constraints, a new registration must be performed, and a new ScanWorld/frozen registration must be created in order to affect changes.
- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.

Note: The **Unfreeze Registration** command removes the created ScanWorld from above the Registration. The operation is disabled if a ModelSpace exists under the created ScanWorld. To unfreeze the Registration, first delete the ModelSpace under the created ScanWorld in the Navigator.

Ungroup

Window: ModelSpace

Menu: Edit | Group

Action: Executes command

Usage: The **Ungroup** command restores a group to its original components.

- To remove a single component without affecting the rest of the group, see the [Remove from Group](#) command.

To restore the original components of a group:

1. Select the group.
2. From the **Edit** menu, point to **Group**, and then select **Ungroup**. The components are restored and may be selected individually.

Unify Clouds

Window: ModelSpace

Menu: Tools

Action: Displays dialog

Usage: The **Unify Clouds** command displays the **Unify Clouds** dialog, which combines multiple point clouds into a single efficient cloud.

- The **Unify Clouds** command is typically used in a ModelSpace containing a registered set of scans and a large number of point clouds, performance is improved by the cloud unification process.
- After the point clouds are unified, you will generally want to use the **Set ScanWorld Default Clouds** command to designate the point cloud as the default point cloud that is loaded whenever a new ModelSpace is created from the parent ScanWorld.

To unify clouds:

1. From the **Tools** menu, select **Unify Clouds**. A warning prompt appears.
2. Click **Unify**. The **Unify Clouds** dialog appears.
3. Select unification options, and then click **Unify**.

Note: Any points not included in the unification process are removed from the ModelSpace when the cloud is unified.

- To reduce the number of points in the resulting unified cloud, select **Reduce Cloud: Average Point Spacing** and enter a value for the average distance between any two points.
- 4. Click **Unify**. The point clouds are combined into a single efficient cloud.

Unify ModelSpace

Window: **Navigator**

Menu: **Tools**

Action: Displays dialog

Usage: The **Unify ModelSpace...** command displays the **Unify Clouds** dialog, which combines multiple point clouds into a single cloud in the selected ModelSpace or ModelSpace View.

- In a ModelSpace containing a registered set of scans and a large number of point clouds, performance is improved by the cloud unification process.
- After the point clouds are unified, you will generally want to use the **Set ScanWorld Default Clouds** command to designate the point cloud as the default point cloud that is loaded whenever a new ModelSpace is created from the parent ScanWorld.

To unify a ModelSpace:

1. In the **Navigator** window, select the ModelSpace or ModelSpace View containing the clouds you want to unify.
2. From the **Tools** menu, select **Unify ModelSpace**. A warning prompt appears.
3. Click **Unify**. The **Unify Clouds** dialog appears.
4. Select unification options, and then click **Unify**.

Note: Any points not included in the unification process are removed from the ModelSpace when the cloud is unified.
- To reduce the number of points in the resulting unified cloud, select **Reduce Cloud: Average Point Spacing** and enter a value for the average distance between any two points.
5. Click **Unify**. The point clouds in the selected ModelSpace or ModelSpace Views are combined into a single efficient cloud.

Update Constraint

Window: **Registration**

Menu: **Constraint**

Action: Executes command

Usage: The **Update Constraint** command removes any constraints that have been deactivated (constraint objects have been made invisible in the ControlSpace) from the current Registration.

- In order to update constraints for a completed (frozen) Registration, the Registration must be reactivated by invoking the **Unfreeze Registration** command.
- For more information on the registration process, see the [Registration](#) entry in the *Getting Started* chapter.

Update Initial Cloud Alignment

Window: **Registration**

Menu: **Cloud Constraint**

Action: Executes command

Usage: The **Update Initial Cloud Alignment** command recomputes the correspondences between the current pick points used in the selected Cloud Constraint and resets the initial cloud alignment.

- This command is useful if the **Optimize Alignment** function is unsuccessful because of an inaccurate initial alignment hint.
- For more information on the cloud registration process, see the [Cloud Registration](#) entry in the *Registration* chapter.

To update initial cloud alignments:

1. From the **Constraint List** tab, select the cloud constraint to update, then select **Show Constraint** from the **Constraint** menu. The ControlSpaces of the constrained ScanWorlds are opened in the Constraint viewers, and the pick points of the cloud constraint are displayed.
2. Modify the set of pick points, adding or removing pick points as needed.
3. From the **Cloud Constraint** menu, select **Update Initial Cloud Alignment**. The pick points and transformation of the cloud constraint are updated to match the new pick points, if possible.

Update Original ModelSpace

Window: **ModelSpace**

Menu: **ModelSpace: File | Launch**

Navigator: **File**

Action: Executes command

Usage: Executing this command copies edited objects back to the ModelSpace from which the current ModelSpace was originally created.

Note: This command is active only for ModelSpaces created via the [File | Launch | Copy Selection to New ModelSpace](#) or [Copy Fenced to New ModelSpace](#) commands.

To update the original ModelSpace from a subordinate ModelSpace View:

1. From the **File** menu, point to **Launch** and then select **Update Original ModelSpace**. A dialog appears.
2. Choose whether or not to mark the current ModelSpace for deletion after the current and original ModelSpaces have been merged.
 - If you select **Yes**, the current ModelSpace is marked for deletion and the original ModelSpace is updated to reflect changes.
 - If you select **No**, the current ModelSpace is left intact, and the original ModelSpace is updated to reflect changes.
 - If you select **Cancel**, the update process is aborted and you are returned to the current ModelSpace.
 - The updated source ModelSpace contains all objects that were either edited or created in the descendant ModelSpace, i.e., objects from the source ModelSpace that have not changed are not duplicated in the resulting merged ModelSpace.

To update the original ModelSpace from the Navigator window:

1. Select the ModelSpace that was created using the **File | Launch | Copy Selection to New ModelSpace** command.
2. From the **File** menu, select **Update Original ModelSpace**. A dialog appears.
3. Choose whether or not to mark the current ModelSpace for deletion after the current and original ModelSpaces have been merged.
 - If you select **Yes**, the current ModelSpace is marked for deletion and the original ModelSpace is updated to reflect changes.
 - If you select **No**, the current ModelSpace is left intact, and the original ModelSpace is updated to reflect changes.
 - If you select **Cancel**, the update process is aborted and you are returned to the current ModelSpace.
 - The updated source ModelSpace contains all objects that were either edited or created in the descendant ModelSpace, i.e., objects from the source ModelSpace that have not changed are not duplicated in the resulting merged ModelSpace.

Update Simulated Scanner Position

Window: **ModelSpace**

Menu: **Tools | Scanner**

Action: Executes command

Usage: The **Update Simulated Scanner Position** command changes the position of the simulated scanner to match the current viewpoint of the ModelSpace from which it was launched.

- For general information on the **Scan Control** window, see the [Scanning](#) entry in the *Scanning with the ScanStation, ScanStation 2, and HDS3000* chapter.

Use Fit Constraints

Window: ModelSpace

Menu: Create Object | Fit to Cloud Options

Action: Toggles menu item ON/OFF

Usage: The **Use Fit Constraints** command enables/disables user-specified constraints used to fit a geometric object to a segmented point cloud in the ModelSpace viewer.

- In order to use fit constraints, the **Fit Constraints** box must be selected and the desired constraints must be set within the **Object Preferences** dialog. See the [Object Preferences](#) command.
- For an overview of fitting objects in *Cyclone*, see the [Modeling](#) entry in the *Getting Started* chapter.

Use Parts Table

Window: ModelSpace

Menu: Create Object

Action: Toggles menu item ON/OFF

Usage: The **Use Parts Table** command enables/disables the use of object-specific parts tables when inserting objects in the current ModelSpace viewer.

- When enabled, object types that have **Catalog/Parts Table** selected in the **Object Preferences** dialog present controls to select a table and part when inserting that object type.
- When disabled, no tables are used by any operation.

User Coordinate System

Window: ModelSpace

Menu: Edit Object | Handles | Constrain Motion to

Action: Executes command

Usage: The **User Coordinate System** command constrains the movement of objects relative to the specified axis of the user coordinate system.

- The **User Coordinate System** command works in conjunction with axis constraints. For information on constraining object motion to an axis, see the [Constrain Motion to](#) entry in this chapter.
- For more information on using user coordinate systems, see the [Save/Edit Coordinate Systems](#) command.

- Some handles may not be able to move according to the constraints (e.g., rotation and other custom handles).
- For more information on using handles, see the [Handles](#) entry in the *Modeling* chapter.

User Feedback...

Window: All windows

Menu: Help

Action: Opens dialog

Usage: The **User Feedback** dialog is provided for users to provide Leica Geosystems HDS with feedback on the use of *Cyclone* in your work sessions, such as suggestions for future enhancements.

To submit user feedback:

1. From the **Help** menu, select **User Feedback**.... The **User Feedback** dialog appears.
2. Enter personal data (name, company, email, phone, etc.) in the appropriate fields. It is important that you provide this data so we can contact you to follow up if necessary. The **Date** and **Time** fields are populated automatically with the current date and time.
3. Enter a summary of the feedback being reported in the **Summary** field, followed by a detailed narrative in the **Detailed Description** field. Provide as much detail as possible.
4. Click **Create Feedback** to open a **Save** dialog; enter a file name for the .txt file (or you can accept the default, *feedback.txt*), select the folder into which you want to save the file, and click **Save**. The report is created and saved to the designated file and location.
5. E-mail the newly created .txt file to support@hds.leica-geosystems.com.

Verify Camera Calibration

Window: Scan Control

Menu: Scanner Control | Camera Calibration

Action: Opens dialog

Usage: The **Verify Camera Calibration** dialog is used to test the accuracy of the current camera calibration.

- For a general overview of camera calibration, see the [Camera Calibration](#) entry in the *Scanning with the HDS2500* chapter.
- This command is only available for the HDS2500.

To Verify Camera Calibration:

1. Place the scanner in a dimly lit room (unlit is preferable) 5-10 meters from the wall that the scanner is facing.
2. Open the **Scan Control** window and connect to the scanner.

3. From the **Scanner Control** menu, point to **Camera Calibration** and select **Verify Camera Calibration**.... A progress dialog appears, describing accuracy as the process tests 50 points.
 - To cancel at any time during the process, click **Cancel**. For a reliable evaluation, testing at least 20 points before canceling is recommended.
 - Acceptable accuracies for HDS2500 camera calibration are average errors of around 2 pixels. Max errors of 8 pixels may be common and acceptable, depending on the scene.

Verify TIN

Window: ModelSpace

Menu: Tools | Mesh

Action: Executes command

Usage: The **Verify TIN** command checks to see if the selected mesh conforms to the definition of a Triangulated Irregular Network (TIN).

- A TIN is a mesh in which there are no vertically overlapping faces.
- TIN verification can only be performed on a *mesh object* as opposed to a point cloud that is currently viewed as a mesh. To create a mesh object from a point cloud, see the [Create Mesh](#) command.

Vertex (From Intersections)

Window: ModelSpace

Menu: Create Object | From Intersections

Action: Executes command

Usage: The **Vertex** command creates a vertex at the planar or linear intersection of either three selected planar objects or two linear objects.

To create a vertex at the intersection of objects:

1. Select three planar objects or two linear objects.
 - Objects such as cylinders that define a centerline are considered linear.
2. From the **From Intersections** submenu, select **Vertex**. The vertex is created.
 - When creating a vertex at the intersection of three planes: Even if the selected objects are not physically touching, the vertex is created at the planar intersections of the selected objects.
 - When creating a vertex at the intersection of lines: If the lines do not intersect, the vertex is placed at the midpoint of the closest approach of the linear objects.

Vertex (From Pick Points)

Window: ModelSpace

Menu: Create Object | From Pick Points

Action: Executes command

Usage: The **Vertex** command creates a vertex at a picked point.

To create a vertex at a picked point:

1. Pick one or more points.
2. From the **Create Object** menu, point to **From Pick Points**, and then select **Vertex**. A vertex is created at each pick point.

View All

Window: ModelSpace, Registration

Menu: ModelSpace: Viewpoint

Registration: Viewers

Action: Executes command

Usage: The **View All** command adjusts the viewpoint so that the entire scene is visible in the current window.

View Half-Space

Window: ModelSpace

Menu: Tools | Cutplane

Action: Toggles menu item ON/OFF

Usage: The **View Half-Space** command hides all geometry below the active Cutplane.

- This can be useful to see "inside" the surfaces that are left visible.
- To toggle whether the geometry above/below the Cutplane is hidden, use the **Flip** command in the [Cutplanes](#) dialog.

View Interim Results

Window: Registration

Menu: Registration

Action: Executes command

Usage: The **View Interim Results** command creates a temporary ModelSpace to display an interim registration of the selected ScanWorlds, Constraints, ModelSpaces, or ControlSpaces.

- For more information on the cloud registration process, see the [Cloud Registration](#) entry in the *Registration* chapter.
- The **View Interim Results** command allows for the visual inspection of the current state of registration for any selected subset of registration objects (ScanWorlds, Constraints, ModelSpaces, or ControlSpaces), without having to first freeze the ScanWorld and create a ModelSpace (see [Create ScanWorld/Freeze Registration](#) and [Create ModelSpace](#) commands).

To view interim results:

1. Select the ScanWorlds, ControlSpaces, ModelSpaces, and/or Constraints to view.
2. From the **Registration** window, select **View Interim Results....** A ModelSpace appears, displaying the content from the selected items.
 - When a ScanWorld is selected, the contents of the ScanWorld (as defined by preferences set in the **Initialization** tab of the **Edit Preferences** dialog) are shown in the temporary ModelSpace. The current registration transformation for the ScanWorld determines its relative position in the new ModelSpace.
 - When a ModelSpace or ControlSpace is selected, its contents are shown in the temporary ModelSpace. The current registration transformation for the space's ScanWorld determines its relative position in the new ModelSpace.
 - When a constraint is selected, all objects referenced by the selected constraint are shown in the temporary ModelSpace. The current registration transformations for the objects' ScanWorlds determine their relative positions in the new ModelSpace.
 - When a single cloud constraint is selected, the ControlSpaces of the constraint are shown, using the current cloud constraint alignment to determine their positions relative to each other in the temporary ModelSpace. This is the recommended method for visually checking the alignment of the cloud constraints before attempting global registration. Similarly, the initial alignment of a cloud constraint (in *not aligned* state) can be visually inspected before running the **Optimize Cloud Alignment** command on it.
3. When finished, close the temporary ModelSpace viewer. A dialog appears with options for saving the temporary ModelSpace.

View Lock

Window: **ModelSpace**

Menu: **Viewpoint**

Action: Displays submenu

Usage: Pointing to **View Lock** displays submenu commands used to temporarily disable the ability to pan, zoom, or rotate the ModelSpace viewpoint.

The following commands are available from the **View Lock** submenu:

[Pan](#), [Zoom](#), [Rotation](#), [Clear Locks](#)

View Mode

Window: ModelSpace, Registration, Scan Control
Menu: ModelSpace: Edit | Modes
 Registration: Edit | Modes
 ScanControl: Viewers
Action: Executes command
Usage: The **View Mode** command enters **View** mode and selects the **View** mode cursor, which is used to manually adjust the ModelSpace viewpoint.

View ModelSpace

Window: Registration
Menu: ScanWorld
Action: Executes command
Usage: The **View ModelSpace** command is used to display a ScanWorld's ModelSpace in one of the Constraint viewer windows.

To view a ModelSpace:

1. From the **ModelSpaces** tab, select the ModelSpace you want to view.
2. From the **ScanWorld** menu, select **View ModelSpace**. The ModelSpace appears in one of the two Constraint viewer windows.
 - The first ModelSpace that you open appears in the subwindow on the bottom left side of the **Registration** window. The next appears on the right, and then each subsequently viewed ModelSpace is alternately loaded into the left and right windows.
 - The Constraint viewer windows are used to view and select the objects to add a manual constraint, and to verify existing constraints.

View Object As...

Window: ModelSpace
Menu: View
Action: Opens dialog
Usage: The **View Object As...** command opens the **View As** tab of the **View Properties** dialog, which is used to set various viewing and rendering properties.

Dialog Options

APPLY TO

Select the class of object to which you want to apply changes. Each class of object has its own set of editable properties.

RESET

Click this button to reset the “view as” properties for the selected class.

COLOR MAP (POINT CLOUDS AND MESH OBJECTS ONLY)

Click this button to invoke the **Color Map Parameters** dialog. See [Edit Color Map](#) in the Commands chapter.

APPLY

Click this button to apply any changes to the "view as" properties for the selected class.

View As Options

Choose one from the list of available options. (The list varies depending on the object class selected in the **Apply To** field.)

LOD - FULL RANGE

When selected, the full set of low-, medium-, and high-resolution level of detail (LOD) is displayed.

LOD - HIGHEST COMPLEXITY

When selected, only high resolution LOD is displayed for maximum rendering detail.

LOD - LOWEST COMPLEXITY

When selected, only low resolution LOD is displayed for faster rendering.

WIREFRAME (MESH OBJECTS ONLY)

When selected, mesh objects are displayed in wireframe mode, rather than as a solid.

PER-FACE NORMALS (MESH OBJECTS ONLY)

When selected, each triangle contained within a mesh object is shaded separately based on its normal.

RAW POINTS (POINT CLOUDS ONLY)

When selected, points are displayed with no meshing.

BASIC MESH (POINT CLOUDS ONLY)

When selected, the cloud is displayed as a mesh with as many triangles as possible.

COMPLEX MESH (POINT CLOUDS ONLY)

When selected, the cloud is displayed as a mesh with only those triangles that are likely to form the actual surface.

CENTERLINE

When selected, only the object’s centerline is displayed.

CENTER POINT (SPHERES ONLY)

When selected, only the centerpoint of the sphere is displayed. Centerpoints are displayed graphically as a vertex.

Other Options

Choose any or all of the available options by checking the box. (The list varies depending on the object class selected in the **Apply To** field.)

SHOW OBJECT'S CLOUD

Select this option to display the point cloud that defines the object.

POINT NORMALS (POINT CLOUDS ONLY)

Select this option to display normals associated with scan points.

SHOW OBJECT'S CENTERLINE

Select this option to display an object's centerline.

SHOW SOLID VALVE (VALVES ONLY)

Select this option to display valves as solid objects rather than as wireframe objects.

W/O OBJECT'S CENTER POINT (SPHERES ONLY)

Select this option to view spheres as solids without displaying center point vertices.

SHOW ELEVATION LABEL (CONTOURS ONLY)

Select this option to display labels for contour lines containing elevation data.

SHOW CONTENTS WITH POINTS (EXCLUSION VOLUMES ONLY)

Select this option to display both objects and points within an exclusion volume.

View Slice

Window: ModelSpace

Menu: Tools | Cutplane

Action: Toggles menu item ON/OFF

Usage: The **View Slice** command toggles whether or not the active Cutplane slices through the data, hiding all geometry beyond the specified thickness of the Cutplane slice. (See the *Set Slice Thickness* entry in this chapter).

- The results resemble a cross-section that can be used for quick 2D plots directly from point clouds and surfaces.

Virtual Surveyor...

Window: ModelSpace

Menu: Tools

Action: Opens dialog

Usage: The **Virtual Surveyor** command is used to select a file for Virtual Surveyor data input/output and then proceed to the **Virtual Surveyor** dialog.

- For information on the Virtual Surveyor process, see the [Virtual Surveyor](#) chapter.

Volume

Window: ModelSpace

Menu: Tools | Measure

Action: Executes command

Usage: The **Volume** command calculates and displays the volume of the selected object.

- To measure a volume of a mesh, use the **Mesh Volume** command instead.

To measure an object's volume:

1. Select the object whose volume you want to measure.
2. From the **Tools** menu, point to **Measure**, and then select **Volume**. The measurements results are displayed in cubic units. The measurement results are also displayed in the Output Box, if it is enabled.

Zoom

Window: ModelSpace

Menu: Viewpoint

Action: Displays submenu

Usage: Pointing to **Zoom** displays submenu commands used to zoom the viewpoint in or out, or zoom into a fence.

The following commands are available from the **Zoom** submenu:

In, Out, Into Fence, Into Selection

Zoom Submenu Commands

IN

Zooms in so objects near the center of the ModelSpace View appear approximately 1.5 to 2 times larger.

OUT

Zooms out so objects near the center of the ModelSpace View appear approximately 1.5 to 2 times smaller.

INTO FENCE

Zooms in until the data fence fills the field of view.

INTO SELECTION

Zooms in until the current selection fills the field of view.

Zoom (Lock)

Window: ModelSpace

Menu: Viewpoint | View Lock

Action: Toggles option ON/OFF

Usage: The **Zoom** command toggles the ability to zoom the viewpoint in or out while in **View** mode.

- When **ON**, you cannot zoom the viewpoint.