
User Guide

Trimble RealWorks® 8.0

Contents

Welcome	15
----------------	-----------

What's New in RealWorks	17
--------------------------------	-----------

Installation of RealWorks	23
----------------------------------	-----------

System Requirements	25
Installing RealWorks	26
RealWorks Plant Tables	27
Storage Tank Application	28
Updating RealWorks	29
Modifying, Repairing and Removing RealWorks	30
License Files	31
Oil, Gas & Chemical Files	31
HASP	34
Contacting Trimble	39

Working Environment	41
----------------------------	-----------

User Interface	43
Menu Bar	44
Toolbars	45
Windows	46
WorkSpace	46
List	47
Property	48
Selection List	49
3D View	50
Tools and Commands	55
Toolbars	55
Menu Bar	62
Shortcut Keys	63
Customizing the User Interface	64
Windows	64
Toolbars	66

Organization of Data 69

Project Data	71
Scans Tree	72
Targets Tree	73
Models Tree	74
Images Tree	75
Project Cloud	76
Getting all Points	77
Getting Remaining Points	78
Active Group	79
Group and Object Nodes	80
In the WorkSpace Window	80
In the List Window	80

Basics of Operation 83

Opening Projects	85
Importing Projects	86
Connecting to Mobile Device	87
Project File Formats	89
Trimble 3D Scanning Files	89
RealWorks Files	90
TZF Files	91
TZS Files	93
Trimble Survey Project Files	95
JobXML, JOB and RAW Files	102
ASCII Files	104
Trimble TX5 and Other FLS Files	107
Surveying Network ASCII Files	108
SIMA ASCII Files	111
AutoCAD Files	112
IXF Files	114
CMF Files	115
PTX Files	116
PTS Files	117
Merging Projects	118
Scans Tree	118
Models Tree	119
Targets Tree	120
Images Tree	121
Saving Projects	123
Save	123
Save As	123
Publish	124
Publishing Projects	124

Page Configuration	125
Media	127
Links	129
Data Size Reduction	129
Enable Extract	130
Published Projects	130
Opening Scan Explorer	134
TZF Scans	135
Creating Sampled Scans	136
Modifying the Path for Input TZF Scan Files	139
Creating Thumbnails	141
Generating Point Color-Coding by Height	142
Importing Images	143
Selection Mechanism	144
WorkSpace Window	144
3D View Window	145
List Window	145
Clearing Selections	145
Undoing Operations	146
Redoing Operations	147
Closing Projects	148
Close	148
Close All	148
Capturing the Screen	149
Setting Preferences	150
General	150
Viewer	151
Units	152
Print	152

Exploring Data 153

Expanding and Shrinking the Project Tree	155
Locating an Item in the Project Tree	156
Finding an Item in the Project Tree	157
Exploring in the 3D View	158
Exploring in the Images Tree	159
Visualizing Data	160
Point Cloud	160
Geometry	162
Hiding all	164
Hiding all Except those Selected	165
Image	165
Inspection Map	169
Station	170
TZF Scan	177
Rendering Data	180
Point Cloud	181

Geometry	183
Lighting Directions	185
Navigating Through Data	186
Projection Modes	186
Displacement Modes	187
View Alignment	199
Standard Views	203
ColorBars	205
Creating	206
Importing	207
Deleting	207
Exporting	208
Editing	209
Hiding/Opening	210
Editing Data	211

Dragging and Dropping an Item	213
Cutting and Pasting an Item	214
Copying and Pasting an Item	216
Deleting an Item	218
Creating a New Group Node	219
Changing Color	220
From the Property Window	220
From the Menu Bar	221
Changing Name	222
Merging Point Clouds	223
Picking Items	224
Picking in the 3D Constraint Mode	224
Picking in the 2D Constraint Mode	227
Point Loading Manager	229

Loading Data	231
Processing Data	232
Basic Tools	233

Measurement Tool	235
Opening the Tool	236
Distance Measurements	237
Angular Measurements	250
Measuring a Position	257
Measuring an Orientation	258
Refining a Measurement	259
Saving a Measurement	260

Segmentation Tool	261
Opening the Tool	262
Drawing a Fence	263
Keeping In (or Out)	265
Creating the Segmentation Results	267
Sampling Tool	268
Opening the Tool	268
Sampling Methods	268
Frame Creation Tool	281
Opening the Tool	282
Selecting a Reference Frame	283
Building a Frame without Constraints	283
Building a Frame with Constraints	289
Creating the Built Frame	295
Shifting a Project	296
Converting to Meshes	297
Creating a Merged Mesh	298
Generating Key Plan	299
From TZF Scans	302
From Current View	303

Media Tools **305**

Video Creator	307
Opening the Tool	307
Defining a Navigation Path	308
Browsing Keyframes	329
Defining Video Parameters	330
Creating a Video	331
Capturing Screen (High Resolution)	333

Registration **335**

Auto-Extract Targets and Register	336
Opening the Tool	337
Target Types	338
Scan Creation	339
Reference Station	340
Registering Stations	341
Auto-Register Using Planes (Target-Less)	342
Opening the Tool	342
Reference Station	343
Registering Stations	344
Registration Report	347
Saving in RTF	348
Options	349

Target-Based Registration Tool	350
Opening the Tool	351
Selecting a Reference Station	352
Auto-Pairing Targets	354
Editing Targets	356
Adjusting Stations	356
Checking the Adjustment	357
Saving the Adjustment Result	366
Refine Registration Using TZF Scans	369
Opening the Tool	369
Reference Station	370
Refining Registration	371
Registration Report	372
Target Analyzer Tool	374
Opening the Tool	375
Selecting a Station	376
Selecting a TZF Scan	377
Focusing on Targets	378
Creating/Editing Targets	380
Updating Network	401
Applying the Result	401
Cloud-Based Registration Tool	402
Opening the Tool	402
Reference Cloud and Moving Cloud	403
Displacing the Moving Cloud	406
Picking Three Pairs of Points	410
Refining the Initial Registration	413
Checking the Registration Results	415
Saving the Registration Results	416
Orientation Tool	417
Opening the Tool	418
Define Horizontal Axis by Picking Two Points	419
Automatic Axis Definition	420
Rotating Counterclockwise 90°	421
Picking the Origin	421
Applying Transformation	422
Closing the Tool	422
Georeferencing Tool	423
Opening the Tool	424
Selecting Stations	425
Assigning Coordinates to Targets	426
Assigning Coordinates to Picked Points	428
Saving the Results	430
Bundle Adjustment	431
Name-Based Bundle Adjustment	432
Station Registration Parameters	434
Importing from TZF Files	434
Exporting to TZF Files	435
Exporting to RMX Files	436

Register Stations With Imported RMX Files	437
Modifying Stations	438
Forcing Leveled	438
Modifying the Instrument Height	439
Setting Over a Known Point	440
Removing a Known Point	442
Forcing Unleveled	443
Modifying Target Matching	444
Matching With	444
Matching Targets	445
Un-matching Targets	447
Renaming Targets	448
Creating Points	449
Creating Topo Points	449
Creating Point Cloud from Topo Points	450
Creating 3D Points	450
Creating a Registration Report (Target-Based)	453

OfficeSurvey

455

Cutting Plane Tool	457
Opening the Tool	458
Defining the Orientation of a Plane	459
Defining the Position of a Plane	464
Defining a Slice	467
Previewing a Single Slice	468
Previewing a Multiple Slice	469
Building Polylines	471
Saving the Cutting Result(s)	472
Contouring Tool	473
Opening the Tool	473
Defining an Elevation Range	474
Calculating Contours	477
Defining Principal Contours	480
Displaying Contours	481
Creating Contours	482
Manipulating Labels	483
2D-EasyLine Tool	484
Opening the Tool	484
Modeling Automatically Polylines	487
Modeling Manually Polylines	489
Editing Polylines	490
Saving Results	498
Mesh Creation Tool	499
Opening the Tool	499
Selecting a Projection Mode	500
Previewing Meshes	505
Creating Meshes	506

Mesh Editing Tool	507
Opening the Tool	508
Selecting an Element	509
Fencing an Area	512
Editing a Mesh	513
Mapping With a Texture	519
Applying Operations	521
Ortho-Projection Tool	522
Opening the Tool	523
Defining a Projection Plane	524
Modifying a Projection Plane	528
Checking a Projection Plane	533
Defining a Zone of Interest	536
Setting a Resolution	538
Rendering	539
Previewing an Ortho-Image	540
Printing an Ortho-Image	541
Splitting an Ortho-Image	542
Creating an Ortho-Image	542
Multi-Ortho-Projection Tool	543
Opening the Tool	543
Defining a Polyline	543
Defining a Zone of Interest	546
Defining Image Parameters	550
Previewing a Single Ortho-Image	553
Creating Ortho-Images	555
Image Rectification Tool	557
Opening the Tool	557
Choosing a Station	558
Defining a Projection Plane	559
Defining a Zone of Interest	567
Setting an Image Resolution	571
Previewing a Rectified Image	572
Splitting a Rectified Image	573
Creating a Rectified Image	574
Polyline Drawing Tool	575
Opening the Tool	575
Defining a 3D Plane	576
Drawing Polylines	577
Selecting Polylines	584
Editing Polylines	585
Deleting Polylines	590
Moving Polylines	591
Auto-Duplicating Polylines	596
Manually Duplicating Polylines	600
Creating Polylines	601
Image Matching Tool	602
Opening the Tool	602
Selecting an Image	603

Selecting Markers	605
Modifying Markers	610
Saving Markers	614
Matching an Image	614
Coloring Points	619
Applying the Matching	619
Volume Calculation Tool	620
Opening the Tool	620
Defining a Plane	621
Defining a Grid Resolution	629
Previewing a Volume	630
Editing a Volume	633
Saving a Volume in the Database	638
Profile/Cross-Section Tool	639
Opening the Tool	639
Selecting a Method	640
From Path	640
From Segments	648
Computing Cross-Sections	653
Displaying Cross-Sections	658
Editing Cross-Sections	659
Creating Profile and Cross-Sections	659
Feature Set Tool	660
Opening the Tool	660
Feature Code Libraries	660
Feature Codes	662
Feature Points	664
Setting Display Mode	669
Creating Feature Sets	669
2D-Polyline Inspection Tool	670
Opening the Tool	670
Selecting a Model for Inspection	671
Setting the Thickness Parameter	671
Viewing the 2D Inspection Results	672
Filtering the 2D Inspection Results	673
Saving the 2D Inspection Results	674
Twin Surface Inspection Tool	675
Opening the Tool	676
Defining a Projection	677
Determining a Resolution	683
Previewing the Inspections	684
Inspection Maps	685
Filtering the Inspections	688
Checking the Inspections	688
Saving the Inspections	689
Surface to Model Inspection Tool	690
Opening the Tool	690
Defining a Projection	691
Determining a Resolution	694

Previewing the Inspections	695
Inspection Maps	696
Checking the Inspections	701
Saving the Inspections	701
Inspection Map Analyzer Tool	702
Opening the Tool	703
Selecting Points & Polylines	705
Selecting Sections & Shifts	711
Selecting Volumes & Surfaces	718
Iso-Curves	729
Colored Meshes	735
Printing Inspection Maps	738
Profile Matcher Tool	739
Opening the Tool	740
Defining a Cutting Plane	742
Defining a Profile	744
Reversing the Profile	749
Creating the Profile	750
EasyProfile Tool	751
Opening the Tool	752
Selecting an Existing Profile	753
Setting the Section Size	754
Modifying Built Elements	755
Creating Profiles	758
Fitting Tool	759
Opening the Tool	759
Fencing a Set of Points	760
Fitting With a Geometry	761
Creating a Fitted Geometry	763
3D Inspection Tool	764
Opening the Tool	764
Previewing a 3D Inspection Cloud	765
Filtering a 3D Inspection Cloud	766
Creating a 3D Inspection Cloud	767
3D Inspection Analyzer Tool	768
Opening the Tool	768
Extracting Clouds From 3D Inspection Clouds	769
Auto-Splitting a 3D Inspection Cloud in a Cluster of Clouds	771
Creating the Extracted Cloud(s)	772

Modeling

773

Geometry Creator Tool	775
Opening the Tool	776
Defining a Plane	777
Defining a Sphere	784
Defining a Cylinder	786
Defining a Regular Cone	788

Defining a Circular Torus	790
Defining a 3D Point	793
Define a Segment	800
Creating an Extruded Entity	804
Creating a Box	805
Creating a Rectangular Torus	816
Cloud-Based Modeler Tool	819
Opening the Tool	819
Selecting a New Cloud Data	820
Defining a Set of Points on the Cloud Data	822
Choosing a Geometry Type	824
Using Constraints	824
Calculating a Geometry	860
Creating a Geometry	861
Geometry Modifier Tool	862
Opening the Tool	862
Modifying a Shape	863
Moving a Geometry	873
Moving a Geometry by Picking Entities	879
Pick a Plane, and then Pan	885
Moving a Geometry Along a User Defined Vector	894
Applying Constraints	898
Duplicator Tool	923
Opening the Tool	923
Choosing a Method	923
Resizing a Path	935
Defining Parameters	935
Reversing the Path Direction	938
Duplicating Items	939
Intersect Tool	940
Opening the Tool	940
Connecting Two/Three Entities	941
Connecting to a Series of Entities	948
Connecting a Plane to a Series of Planes	949
Plane Bounding Tool	951
Opening the Tool	952
Editing the Selected Plane Bounds	953
Selecting a Polyline for Bounding	953
Defining a Bounding Polyline	954
Editing a Bounding Polyline	960
Deleting a Bounding Polyline	964
Activating/Deactivating the Selection Mode	964
Moving a Bounding Polyline	965
Applying Bounds	967
Validating Bounds	969
3D Picking Tool	970
Picking an Entity with Direction	971
3D Direction Tool	973
Defining a 3D Direction Using Precise Methods	974

Defining a 3D Direction Using Visual Methods	976
Validating a 3D Direction	977
3D Point Tool	978
Picking a Point	978
Picking Three Planes	979
Picking a Plane and a Segment	981
Picking an Entity with Center	982
Projecting a 3D Point on a Plane	983
Editing Parameters	984
Validating a 3D Point	984
3D Axis Tool	985
Picking an Axial Entity	985
Picking Two Points	986
Picking Two Planes	987
Editing Parameters	988
Validating an 3D Axis	988
3D Radius Tool	989
Editing Parameters	989
Picking a Radial Entity	990
Picking Two Points of a Diameter	990
Picking Two Points of a Radius	990
Picking an Axis and a Point	991
3D Secant Tool	993
Picking a Cylinder to be Secant With	994
3D Plane Tool	996
Defining a 3D Plane in all Navigation Modes	998
Defining a 3D Plane in the Station-Based Navigation Mode	1000
Modifying a 3D Plane's Size	1003
Validating a 3D Plane	1003
Managing SteelWorks Catalogs	1004
Importing SteelWorks Catalogs	1004
Removing SteelWorks Catalog List	1005
SteelWorks Creator Tool	1006
Starting the Tool	1006
Selecting a New Cloud Data	1007
Defining a Set of Points on the Cloud Data	1009
Choosing a Section Type	1011
Selecting a Catalog File	1011
Selecting a Table	1012
Selecting a Reference	1012
2D Sections	1013
Defining a Length	1015
Creating Beams	1016
EasyPipe Tool	1017
Opening the Tool	1017
Selecting a New Cloud Data	1018
Defining a Set of Points on the Cloud Data	1020
Extracting an Initial Cylinder by Picking	1022
Selecting an Initial Cylinder for Tracking	1023

Starting Tracking Cylinders	1024
Continuing in Tracking Cylinders	1025
Deleting Extracted Cylinders	1026
Smoothing Extracted Cylinders	1027
Modeling Extracted Cylinders	1028
Creating Extracted Cylinders	1029
Exporting Pipe Center Lines	1030
As a DWG Format File	1031
As a DXF Format File	1032
As a DGN Format File	1033

Advanced Features **1035**

Setting as Home Frame	1037
Equalizing Point Cloud Luminance	1038
Equalizing Point Cloud Color	1039
Equalizing Image Color	1040
Coloring Points Using Station Images	1042
Deleting Geometries	1043

Exporting Data **1045**

Exporting Selection	1047
Google Earth (KMZ) Format	1048
BSF Format	1052
PDMS Macro Format	1052
E57 Format	1053
LAS Format	1054
AutoCAD PCG format (AutoDesk Revit MEP)	1055
Alias/WaveFront (OBJ) Format	1056
MicroStation (DGN) Format	1057
AutoCAD (DXF) Format	1059
ASCII Format	1060
LandXML Format	1061
Solids for AutoCAD	1062
Advanced Exports	1068
Object Properties	1068
Images	1069
Ortho-Images	1070
Measurements	1071
Feature Sets	1073
Exporting TZF/TZS Luminance Images	1074
Converting to BSF	1075
Converting to E57/PTX/PTS	1082
Inspection Maps	1085

Storage Tank Application 1093

Tank Calibration Tool	1095
Activating	1095
Defining a Dipping Plate	1096
Defining Body Parameters	1098
Defining Sump Volume Calculation Settings	1099
Generating Results	1100
Exporting	1101
Creating	1102
Tank Calibration Check	1103
Activating	1103
Filtering Sections	1103
Displaying Sections	1104
Editing Sections	1105
Applying	1108

Index 1109

Welcome

The **RealWorks** family of products is composed of **RealWorks Viewer**, **RealWorks Base**, **RealWorks Advanced**, **RealWorks Advanced-Modeler** and **RealWorks Advanced-Plant**.

RealWorks Viewer

This is a software tool for visualizing and exploring as-built data acquired by laser scanning technologies. In general, such a data set contains a 3D point cloud and optionally a collection of 2D images. This software tool allows you to load as many point clouds as needed. Each point of a point cloud can contain not only its 3D coordinates, but also other attributes such as intensity and surface normal. You can visualize a point cloud in 3D, rotate, pan or zoom in/out in order to explore it in detail. Visualization can be enhanced in different ways: points shaded by intensity, by color or by its normal, according to a view, etc. You can also compare a 3D point cloud with 2D images (if available). When images are registered with the 3D point cloud, you can visualize both data sets from the point of view from which the images were captured.

Features also available in **RealWorks Viewer** are:

- Registration report and station visualization,
- **Sampling Tool**,
- **Segmentation Tool**,
- **Frame Creation Tool** and set as **Home Frame**,
- **Measurement Tool**,
- **Cutting Plane Tool**,

Tip: **RealWorks Viewer** is free and works with RWP format files. You can download it from the Trimble website. You only need to fill the form that will come after clicking the link to download.

Note: You cannot save with **RealWorks Viewer**.

Note: **RealWorks Viewer** only functions with **RWP** files. For more options relating to collaborating and sharing, see the **Publisher** function in certain modules.

RealWorks

This software tool provides you with a set of tools for processing 3D point clouds and 2D images in order to obtain the necessary information for your applications (or projects). Generally, this processing can be divided into three modes: **Registration**, **OfficeSurvey™** and **Modeling**.

- In **Registration**, you can register scans with respect to other scans and/or with respect to a set of survey points. The registration method is either target-based or cloud-based. When some targets have been used, you can first check and modify them. Then the **Target-Based Registration Tool** automatically registers them simultaneously. The results are validated through registration errors, which can be saved in a report. You can also use the **Geo-Referencing Tool** to put the scanned data into a known coordinate system.
- In **OfficeSurvey™**, you can extract different types of 2D drawings (polylines, contours, cross-sections, profiles, etc.) from the point clouds. These extracted results can then be exported into CAD systems including, but not limited to, AutoCAD® and MicroStation®. You can select and match 2D images to the point clouds; generate one (or multi) ortho-image(s) or collect survey points as Total Stations can do. You can generate triangular meshes from the point clouds and if required, carry out further editing of the result(s). You can determine the volume of a point cloud (or a mesh), the volume between two point clouds (or two meshes) or the volume between a point cloud and a mesh. If the volume information is not enough, you can compare two surfaces between each other (two point clouds or meshes together, a point cloud with a mesh or a point cloud/mesh with a model) and generate an inspection map. Since the metric information is still in the inspection map, you can extract measurements like surfaces, volumes, points, drawings like polylines, sections, shifts, colored meshes, iso-curves, etc. You can match a profile (2D curve, cross-section, polyline, etc.) at a specific point and in a given direction in a 3D scene; easily extract profiles along curbs, pavements, rail lines, cuttings, natural features, etc. or fit a set of points with a geometry which can be of planar, spherical or cylinder shape.
- In **Modeling**, you can create a geometry of following types: **Sphere**, **Plane**, **Cylinder**, etc. The creation can be based on a point cloud selection (or not). The created geometry can be then duplicated, modified, moved, etc. A sub-module including tools for modeling pipes and structural steelworks (with the notion of catalogs) has been added.

All **RealWorks** products can support a huge amount of points. The user is able to precisely control which points are loaded into memory and thus which are available for editing with all the regular tools. All **RealWorks** products include tools for managing Trimble FX data; importing TZS scans, automatically creating station(s) on import with link to TZS; creating scans, extracting targets, registering the created stations, creating sampled scans with spatial resolution, converting to Trimble LASERGen format.

What's New in RealWorks

In RealWorks 6.5.1

New Features in Existing Tools:

- In the **SteelWorks Creator Tool**, the user can now edit point cloud(s) thanks to a **Point Cloud Manager**.
- In the **Scan/Target Creator Tool**, the **Spherical Target** extraction is not limited to spheres for which the radius is equal to 38.10 mm.
- In the **Measurement Tool**, three new types of measurements have been added. The user can now measure a distance in a horizontal plane, along a vertical axis or a clearance distance.
- The user is able to configure the mouse buttons for **Zoom In** or **Zoom Out**.
- The user is able to open or drag and drop TZS format files within **RealWorks**.
- In the **Export Inspection Vertical Slices** (or **Horizontal Slices**) feature, the user can choose the unit of measurement to represent values.

New Tools:

- The **Center Line Tool**.
- Import of Trimble CMF format files.
- Rendering cloud issued from TZS format files according to the height of points along the **Z** (or **Elevation**) axis.
- Automatic extraction of **Spherical** and **Black and White Flat Targets** and register the stations they issued from.
- The user can create a plane containing the vertical axis by picking two 3D points.
- He (or she) can also make vertical an axial geometry or horizontal a plane with two new tools in the **Geometry Modifier Tool**.
- Etc.

In RealWorks 6.5.3

New Tools:

- The user can now convert TZS format files to PTX (or PTS) format files.

In RealWorks 7.0

New Features in Existing Tools:

- In the **Segmentation Tool**, the user can now draw a **Circular Selection**.
- In the **Frame Creation Tool**, the user can set an axial entity's axis as **X** (or **Y** or **Y**) **Axis** of the frame to create.
- In the **Measurement Tool**, some additional measurement methods have been added.
- In the **Scan/Target Creator Tool**, the user can create a **Point Target** by picking a point from a loaded TZS format file.
- The user can use the **Find Best Extrusion** facility in the **3D Plane Tool** to fit a plane in order to calculate an **Extrusion** in the **Cloud Based Modeler Tool** or **2D Section** in **SteelWorks Tool**
- Etc.

Enhancement of Existing Tools:

- In the **3D Inspection Analyzer Tool**, a histogram has been added to the **From** and **To** sliders.
- The **Open Wizard for ASCII Files** options change when importing an ASCII format file into a current open project.
- In the **Geometry Creator Tool**, the user can apply two new constraints to a **Segment: Pick Two Planes** and **Pick Axial Geometry**.
- In the **Geometry Creator Tool**, the user can apply a new constraint to a **3D Point: Pick Two Axial Entities**.
- Etc.

New Tools:

- The user can display the 3D position and the name of each station in the **3D View**.
- And if required, display the Network Visual(s) of a single (or all stations).
- The user can color the point cloud using station-based images.
- A new licensing system has been implemented: HASP
- The user can open a Trimble TSPX format file, a common project format file between **RealWorks** and **Trimble Business Center**.
- Etc.

In RealWorks 7.1

New Features in Existing Tools:

- **RealWorks** supports now a new type of file called **Trimble Scan File** which gathers two kinds of formats: **TZS** and **TZF**.
- Filtering by zone is enabled when creating sampled scans from **TZF/TZS** scans or when converting from **TZF/TZS** scan files to **BSF**.

Enhancement of Existing Tools:

- Conversion of proprietary scanner data files: **TZS**, **FLS**, **iQscan** and **PTX** to **TZF**.
- New naming convention for ortho-images when splitting them into several pieces.
- In the **Fitting Tool**, the user can now fit a set of points with a **Vertical Cylinder** or a **Horizontal Plane**.

New Tools:

- **Storage Tank Tool**,
- **Storage Tank Check Tool**.

In RealWorks 7.2.1

Enhancement of Existing Tools:

- In the **Auto-Extract and Register** feature, the user can choose and set a station as **Reference Station**.
- In the **Target-Based Registration Tool**, targets are matched automatically. The **By-hand** method has been removed.
- **Scan/Target Creator Tool: Polygonal Selection** by adding (or subtracting, or intersecting) has been removed, TZF scans can be open directly from the **Scan Target Creator Tool**, and the load percentage for scans is now 20 million points.
- The **Scan/Target Creator Tool** has been removed from 7.2
- The **Complete Station By Adding New Scans** feature has been removed.
- When opening a **TZS** format file, the conversion to the **TZF** format is now obligatory. Declining the conversion will close the current open project.
- Import of **Riegl** file format (3dd) has been removed.
- **TZF** scans cannot be open in a **2D Viewer**.
- **Discontinuity Display** rendering removed from the 3D View toolbar.
- The **Export as Point Cloud for AutoCAD (PTC)** feature has been removed from 7.2.

New Tools:

- **Generate Key Plan from TZF Scans,**
- **Generate Key Plan from Displayed Cloud.**
- Extract Active Sub-Project.
- Able to open **Trimble TX5** files (with fls or IQscan extensions).

In RealWorks 7.2.2

- Compatible with Windows 8
- In **RealWorks Viewer**, the items below have been removed:
 - **Export Selection** and **Advanced Export** features,
 - **Save** and **Save As** features,
 - **TZF** Scans features,
 - Import features.

In RealWorks 8.0

Enhancement of Existing Tools:

- A station can now contain several **TZF** scans within, with one set as **Main Scan**.
- The ten last opened files are listed at the bottom of the **File** menu. As a shortcut, each of them can open by selecting it from this menu.
- **Generate Key Plan from TZF Scans**: The feature is now called **Generate Default Key Plan (Top View)**. The option used to compute the **Key Plan** is not based on the **Elevation** information but on the **Cloud Rendering** option chosen within **RealWorks**.
- **Target-Based Registration Tool**:
 - Targets matched together and target groups are renamed. Instead of keeping their default name or **TargetX** (when they are extracted) and **mTargetX** where **X** is an order, they are renamed as **XXX**. **XXX** starts at 001 and is incremented by one.
 - Stations that are not selected as the input of the **Target Based Registration Tool** are now removed from the **Registration Details** dialog.
- **Target Analyzer Tool**:
 - The **Re-fit** and **Fit** buttons are replaced by an icon.
 - The **Flat Target** icon has been changed in the **Fitting Tool**.
- **RealWorks** no longer needs to disable the **Aero** function in **Windows**.

New Tools:

- **Auto Register using Planes (Target Less)**
- **Refine Registration Using TZF Scans**
- **Publish projects**.
- **Rename targets that are matched manually together**.
- **Import (or export) Station Registration Parameters from (or to) TZF files**.

- **Graphics Card:** 1GB, or higher 3D Open GL 3.2
- **Sub-Project Manager** and **Partial Loading Manager** are both removed from **RealWorks 8.0** as the notion of **Sub-Project** and **Partial Loading Manager** do not exist anymore. A **Point Loading Manager** system has been added.
- **TZF Scans** can be exported to **E57** format as regular grid data sets.
- **PTX** format file with several scans in the same station: A **TZF Scan** is created for each scan in the same station in the **RealWorks** project.
- Features removed from RealWorks 8.0:
 - **Erase Points** from the **Edit** menu,
 - **Reduce Number of Points When Moving** from the **Preferences/Viewer** dialog.
 - Project files saved in a version older than 5.0 are not supported anymore.
- The file formats listed below are removed from the **Open** dialog:
 - **PPF** Trimble proprietary PointScape project file format,
 - **DCP** Trimble proprietary PocketScape project file format,
 - **SOI** Trimble proprietary format for scanned files.
- **RealWorks Plant** as licensing module removed

CHAPTER 1

Installation of RealWorks

In this chapter, the user will find information about system requirements, program installation, activation and how to contact Trimble.

System Requirements

To run **RealWorks**, the system requirements are as shown.

- **Operating System:** Microsoft® Windows® Seven® (64-bit OS) and 8 (64-bit OS)
- **Processor:** 2GHz or higher
- **RAM:** minimum 8 GB - 16 (or 32) GB recommended
- **Graphics card:** 1GB, or higher 3D OpenGL 3.2
- **Mouse:** Three buttons with wheel
- DVD-Rom drive
- **SSD** drive recommended

Notes:

- Always update graphics card drivers before using **Trimble 3D Spatial Imaging** office software.
- **Trimble 3D Spatial Imaging** office software is designed to provide superior data processing and editing performance. To ensure the best possible software use experience, and an optimum productivity/cost-of-equipment ratio, Trimble highly recommends that users acquire the most powerful hardware configuration available at the time of purchase of the computer equipment on which the software is intended to be used.

Caution: If your graphics card is not compliant with OpenGL 3.2, an error message appears. If this case occurs, please, update your graphics card's driver to get the latest version of OpenGL.

Tip: If your computer has two graphics cards: an integrated graphics card with low performance and a dedicated graphics card with a high performance processor. Please enforce the use of the high performance processor graphics card with **RealWorks**.

Installing RealWorks

This section explains how to install **RealWorks** on a standalone computer. Before you install **RealWorks**; close all Windows programs, and ensure that your computer has sufficient operating system requirements and memory capabilities (for more information, see the **System Requirements** section). Make sure that you have local or domain administration rights.

To Install RealWorks:

1. Turn on your computer and start Microsoft Windows.
2. Insert the **RealWorks** software CD in your computer's CD-ROM drive. If Autorun is enabled, the Setup program should start automatically.
3. At the **Welcome to RealWorks 8.0 Setup** dialog, press **Next**. The **System Information and Current Program Version** dialog appears. System information and the current program version are listed.
4. Click **Next**. The **License Agreement** dialog appears.
5. Read carefully the terms of the license agreement.
6. If you do not accept the terms, check the "I do not accept the terms of the license agreement" option and the install procedure will close.
7. If you accept all the terms, check the "I accept the terms of the license agreement" option and click **Next**. The **Choose Destination Location** dialog opens.
8. In the **Choose Destination Location** dialog, choose **Next** to accept the default install directory C:\Program Files\Trimble\RealWorks 8.0.
9. If you wish to install in a different directory, choose the **Change** button. After you have chosen the install directory, press **Next**. The **Setup Type** dialog appears.
10. In the **Setup Type** dialog, choose the type of **Setup** you prefer. **Complete** will install the program with all features. **Custom** will require you to choose the options to install. By default, **Complete** is selected. You have choice between two license types per software product: **Trimble OG&C's** license file or **HASP** license file.
 - If you have a **HASP** license file, keep the **Complete** option and click **Next**. The **Select Program Folder** dialog appears.
 - If you have a **Trimble OG&C's** license file, choose the **Custom** option and click **Next**. The **Select Features** dialog opens.
 - a) Keep the **Application Files**, **Program DLLs** and **Help Files** options checked.
 - b) Check the **Trimble OG&C License** option.
 - c) Click **Next**. The **Select Program Folder** dialog appears.

11. You should select a program folder inside which **Setup** will add program icons. You can keep the given program folder, type a new one in the **Program Folder** field or select an existing one from the **Existing Folders** field.
12. Press **Next**. The **Ready to Install the Program** dialog appears.
13. Press **Install**. The **Setup Status** dialog opens and files are installed. The **Setup Type** dialog appears.
14. Select the type(s) of file you wish **RealWorks** to take in charge. Four types are available: **NEU** and **ASC** are an ASCII format file extension, **JXL** is an extension of text files exported from Trimble Survey Controller™, Survey Manager™ or Survey Pro™ software. **TZS** is an extension of files from Trimble's **LASERGen**. **TSPX** is a Trimble Survey Project file.
15. Click **Next**. The **InstallShield Wizard Complete** dialog appears. You have successfully installed **RealWorks** on your computer.
16. Click **Finish** to complete the installation*.

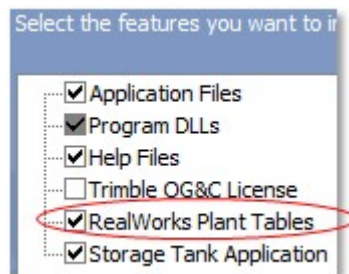
Notes:

- If the **Setup** program does not start automatically, choose **Run** from the **Start** menu. The **Run** box opens. In the **Open** field, enter the CD-ROM drive letter, followed by ".\setup". For example, enter d:\setup.
- (*) Sometimes, you may need to restart your computer.

Caution: If there is already a version of **RealWorks** installed on your computer and if a session is open. Please, exit **RealWorks**. This avoids the **Setup** of a new version of **RealWorks** to interfere with the current version.

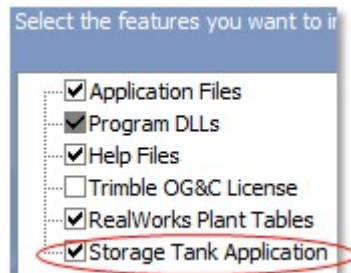
RealWorks Plant Tables

RealWorks includes some catalog files. These files are automatically installed when choosing **Complete** as **Setup Type**. If you do not want them, you need to first choose **Custom** as **Setup Type**, and then un-check the **RealWorks Plant Tables** option in the **Select Features** page.



Storage Tank Application

The **Storage Tank Application** is an option which is automatically installed when choosing **Complete** as **Setup Type**. If you do not want this option, you need to first choose **Custom** as **Setup Type**, and then un-check the **Storage Tank Application** option in the **Select Features** page.



Updating RealWorks

This section explains how to update **RealWorks** on a standalone computer. The update only applies when you move from one version (of **RealWorks**) to the higher.

To Update RealWorks:

1. Turn on your computer and start Microsoft Windows.
2. Insert the **RealWorks** CD in your computer's CD-ROM drive. If **Autorun** is enabled, the **Setup** program should start automatically.
3. If the **Setup** program does not start automatically, run it from the **Start** menu as described in the previous section.
4. At the **Welcome to RealWorks 8.0 Setup** dialog, press **Next**.
5. The **System Information** dialog appears and system information and the current program version are listed. Press **Next**. The **Ready to Install the Program** dialog opens.
6. Click **Install**. The **Setup Status** dialog appears.
7. Once the update done, you are prompted to either view the Help file or to launch **RealWorks** by checking the option.
8. Click **Finish**.

Modifying, Repairing and Removing RealWorks

This section explains how to modify, repair and remove **RealWorks** from a standalone computer. Modification, repair and removal (of **RealWorks**) apply only when you want to make a change of an option in an existing installation, or simply to uninstall **RealWorks**.

To Modify, Repair and Remove RealWorks:

1. Turn on your computer and start Microsoft Windows.
2. Insert the **RealWorks** CD in your computer's CD-ROM drive. If **Autorun** is enabled, the **Setup** program should start automatically.
3. If the **Setup** program does not start automatically, run it from the **Start** menu as described in the previous section.
4. At the **Welcome to RealWorks 8.0 Setup** dialog, press **Next**.
5. The **System Information** dialog appears and system information and the current program version are listed. Press **Next**. The **Modify, Repair or Remove the Program** dialog opens.
6. Choose any of the following options:
 - Select **Modify** and click **Next**. You can add new components in your program or select currently installed components to remove.
 - Select **Repair** and click **Next**. All program components installed in the previous setup will be re-installed.
 - Select **Remove** and click **Next**. All installed program components will be removed.
7. If **Modify** has been selected, the **Select Features** dialog appears.
 - a) Un-check the components to clear and click **Next**.
 - b) Or check the components to install and click **Next**.
 - c) Click **Finish** to end the maintenance.
8. If **Remove** has been selected, the **Confirm Uninstall** dialog appears; click **Finish** to end the uninstall procedure.

Tip: You can also use the **Add/Remove** tool via your Windows® control panel.

Note: Updating, when carried out frequently, may generate residual files on your hard disk. To minimize such eventualities, we recommend that you completely remove **RealWorks** from your hard disk and perform a new installation procedure.

License Files

There are at least two license types the user can use with **RealWorks: OG&C** and **HASP**.

Oil, Gas & Chemical Files

A **RealWorks Oil, Gas and Chemical** license file can be of two types: a local license or a server license. Before being able to use **RealWorks Oil, Gas and Chemical 8.0**, first check the **Custom** option in the **Setup Type** dialog when installing **RealWorks** and then register it.

Registering the RealWorks Power, Process and Plant Software

To Register RealWorks Power, Process and Plant Software:

1. After opening **RealWorks Power, Process and Plant 8.0**, the following message "**Connection problem with license server: the software must be shut down**" appears. Close the message. The **Trimble Power, Process and Plant - Client License Configuration Tool** dialog appears with the **Configuration** tab open by-default.
2. Click on the **Create Registration** tab to open it.

The screenshot shows a Windows-style dialog box titled "Trimble Process Power Plant - Client License Configuration Tool". It has four tabs: "Configuration", "Create Registration" (which is selected), "Server Diagnostics", and "System Information". The "Create Registration" tab contains a form with the following fields:

- Company Name:
- Site:
- Address:
- Address (optional):
- City:
- State:
- Zip:
- Country:
- Administrator:
- Email:
- Phone:
- IP Address:
- Drive:

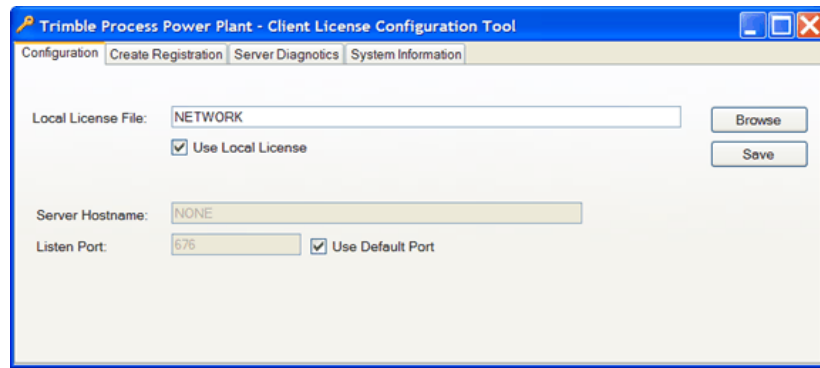
At the bottom right of the form are two buttons: "Email" and "Save".

3. Complete the form by entering the necessary information.
4. Click **Email** to send the completed form.
5. If required, click **Save** to save the completed form as a backup.
6. Click on the **Close** button. The **Trimble Power, Process and Plant - Client License Configuration Tool** dialog closes.

Entering a Trimble Power, Process and Plant License

To Enter a Trimble Power, Process and Plant License:

1. Click **Start** and then select **All Programs > Trimble > RealWorks 8.0> Trimble PPP License**. The **Trimble Power, Process and Plant - Client License Configuration Tool** dialog appears with the **Configuration** tab opens by-default.



2. Do one of the following:
 - If you have a network license, do as described below:
 - a) Un-check the **Use Local License** option. The **Local License File** field and the **Browse** button both become dimmed and the **Server Hostname** field is enabled.
 - b) Enter a server name in the **Server Hostname** field.
 - c) If required, click **Ping** in the **Server Diagnostics** dialog to diagnose the server.
 - If you have a local license, do as described below:
 - a) Keep the **Use Local License** option checked.
 - b) Click **Browse**. The **Open** dialog opens with the "license files (*.lic)" set as default in the **Files of Type** field.
 - c) Navigate to the drive/folder to locate the license file.
 - d) Click on the license file to select it. Its name appears in the **File Name** field.
 - e) Click **Open**. The license file name appears as well as its path in the **Local License File** field.
3. In the **Trimble Power, Process and Plant - Client License Configuration Tool** dialog, click **Save**.

4. Click **Close**.

Swapping a Local License for a Network License and Vice Versa

To Swap a Local License for a Network License and Vice Versa:

1. Open **RealWorks Power, Process and Plant 8.0**.
2. From the **Help** menu, select **License**. The **Trimble Power, Process and Plant - Client License Configuration Tool** dialog opens.
3. Do as described in the "Entering a Trimble Power, Process and Plant License" topic.
4. Click **Save** and **Close**.

HASP

A **HASP** license file can be either a single-user license file or a network license file installed on your organization's network. To be able to use a **HASP** license file, you need to check the **Complete** option in the **Setup Type** dialog when installing **RealWorks**.

Locating a License File

To Locate a HASP License File:

1. In **RealWorks**, select **License** from the **Help** menu. The **License Manager** dialog opens.
2. Click on the **Search Type** pull down arrow.
3. Choose one of the following items.
 - **Automatic**: This is the default search option. This allows searching for a license by first checking whether a HASP hardware key is connected to a USB port on your computer. If not found, it checks whether a "Detachable License" is installed on your computer. If not found, it checks whether a multi-user license is available on a HASP network key. Optionally, you can specify a network server in the **License Server** drop-down list; otherwise, all network servers are searched.
 - **Local Hardware**: This option allows searching for a license installed on a HASP hardware key connected to a USB port on your computer.
 - **Local Software**: This option allows searching for a "Detachable License" when using a multi-user license installed on your computer.
 - **Local Automatic**: This option enables to search for a license by first checking whether a HASP hardware key is connected to a USB port on your computer. If not found, it checks whether a checked-out instance of a multi-user license is installed on your computer.
 - **Network**: This option enables to search for a multi-user license on a HASP network key installed on the server specified in the **License Server** drop-down list. If the correct server is not already included in the list, select the **Search Network** option from the **License Server** drop down list to locate it.
4. Click the **OK** button. The **License Manager** dialog closes.

Single-User License

A **Single-User License** is a **HASP** hardware key the user has to plug into a USB port of his computer.

Upgrading

You can update your current license if you need to add more features or extend your warranty period.

To Upgrade a Single-User License:

1. Connect your **HASP** hardware **USB** key into a **USB** port of your computer.
2. Ensure that no other **HASP** hardware **USB** keys are connected and **RealWorks** is running.
3. In **RealWorks**, select **License** from the **Help** menu. The **License Manager** dialog opens.
4. Check that the key ID displayed in the **Features** list matches the key ID for this upgrade.
5. Click the **Upgrade** button. The **Upgrade License** dialog opens.
6. Enter the 19-digit code.
7. Click the **Upgrade** button. The **Upgrade License** dialog closes.

Note: You need to have an Internet connection on your computer. It is required to check your key ID.

Multi-User Licenses

A **Multi-User License** is a **HASP** network license key. Generally speaking, it is a 19-digit code the user gets when buying such license.

Detaching

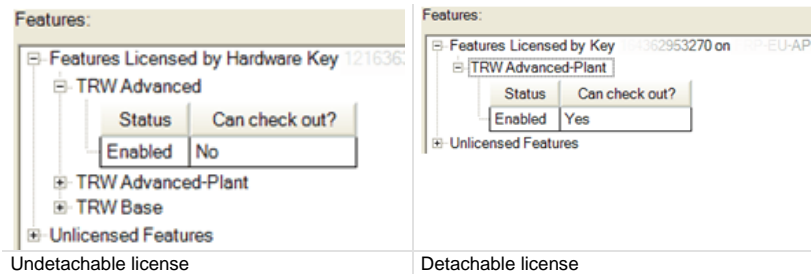
To be able to use a **HASP** network license key, the user needs to be connected to his organization network. But if required, he can detach it from the network and attach it to his machine for a limited period of time allowing by this way running **RealWorks** running without a network connection.

Checking for Checkout Support

You need to check if your HASP network key (multi-user license) is installed on your network and is configured to support a license checkout.

To Check for a License Checkout Support:

1. In **RealWorks**, select **License** from the **Help** menu. The **License Manager** dialog opens.
2. In the **Features** panel, you may check:
 - The feature(s) licensed by your key ID,
 - For feature, the ability to be checked-out (or not).



3. Click the **Ok** button. The **License Manager** dialog closes.

Configuring a Computer to Allow License Checkout

To Configure a Computer to Allow License Checkout:

1. In **RealWorks**, select **License** from the **Help** menu. The **License Manager** dialog opens.
2. In the **License Manager** dialog, click the **HASP Admin Control Center** link located in the lower-left corner of the dialog. The **Sentinel HASP Admin Control Center** page is displayed in a browser window.
3. In the **Administration Options** panel, select **Configuration**. The **Configuration for HASP License Manager on "Computer_Name"** appears.
4. Select the **Detachable Licenses** tab.
5. Check the **Enable Detaching of Licenses** option.
6. Click the **Submit** button.

Tip: The **HASP Admin Control Center** page can also be reached by first clicking the **Start** button on the taskbar and then **All Programs / Trimble / Trimble RealWorks 7.0 / HASP Admin Control Center**.

Checking-out

To Check-out a Multi-User License:

1. In **RealWorks**, select **License** from the **Help** menu. The **License Manager** dialog opens.
2. Click the **Check Out** button. The **Check Out License** dialog opens.
3. Enter the number of days.
4. Or first, click on the **Expiration** pull down arrow.
5. And then, select an expiration date from the calendar.
6. Click the **OK** button. The **Check Out License** dialog closes.
7. Click the **Ok** button. The **License Manager** dialog closes.

Note: The **Check Out** option must be selected on the host machine from which the license will be detached.

Updating

To Update a Multi-User License:

1. Click the **Start** button on the taskbar and then **All Programs / Trimble / Trimble RealWorks 7.0 / Network License Update**. The **Update Network License** dialog opens.
2. Enter the 19-digit code.
3. Click the **Update** button.

Notes:

- The user should choose the "**Related Tools and Utilities / Update Network License**" from the installation DVD to automatically install the **Update Network License** utility on his computer.
- You need to have an Internet connection to proceed to the update.

Contacting Trimble

For information regarding this (or other) Trimble software product(s), please visit the Trimble website at www.trimble.com.

For contacting Trimble support: please refer to list below.

- For **Survey Division**: trimble_support@trimble.com
- For **Power, Process and Plant Division**: ppp_support@trimble.com

CHAPTER 2

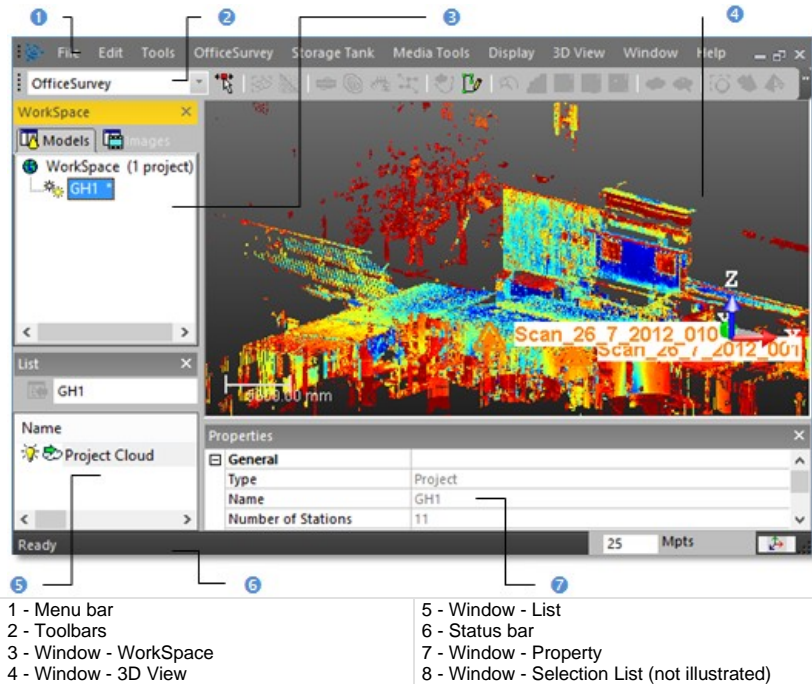
Working Environment

RealWorks includes three modules (**OfficeSurvey™**, **Registration** and **Modeling***); each corresponds to a processing mode. We are not going to develop here each of the processing modes, but note that the working environment which appears after starting **RealWorks** changes according to the module you set.

Note: (*) The **Modeling** processing mode is not present in **RealWorks (Base)** and **Advanced**.

User Interface

When you start a **RealWorks** session, you can see the main window with its working sub-window components. After you open a project, these components will activate so you can start working with them. You can customize the settings of the different components inside the main window. By default, a **RealWorks** session looks like the following example.



Menu Bar

RealWorks provides you with a set of tools and commands. The menu bar, always open and displayed on the top of the user interface, contains all available tools and commands that you can use. This bar is composed of main menus that you have to drop-down in order to reach the tool (or command) you wish to use. For some tools (or commands), you may need to go to the sub-menu to reach them.

Some main menus will be automatically added (or deleted) according to the processing modes you are actively using. Many tools and commands can also be reached from the toolbars or the pop-up menus.

Toolbars

The **RealWorks** user interface is composed of different toolbars. Each of them has specific uses. By default, all toolbars are not open when you start **RealWorks**. Those that are opened are displayed either horizontally under the menu bar or vertically beside the **WorkSpace** window. You can move each toolbar to any location within the user interface, open or close them.

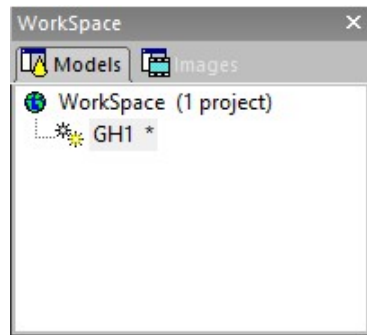
Windows

RealWorks main window has several components, which are described hereafter.

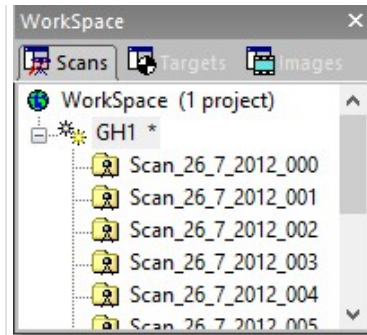
WorkSpace

The **WorkSpace** window (always opened when the user interface appears) is the one located under the toolbars and comprises a set of tabs. This window is used for organizing data hierarchically in a tree called **Project Tree**. This main tree is subdivided into sub-trees called **Scans**, **Targets**, **Models** and **Images**. Each of them is used for organizing certain types of data from a loaded project.

To display a sub-tree, click the corresponding tab. Note that only one tab can be displayed at any given moment. The **Images** tab and the **Models** tab appear respectively two and three times - one in each processing module (**OfficeSurvey™**, **Registration** and **Modeling***) for the **Images** tab and only in the **Registration** and **Modeling*** processing modules for the **Models** tab. You can move this window to any location within the user interface, or close, reduce and restore it.



The WorkSpace window in OfficeSurvey™ and Modeling*

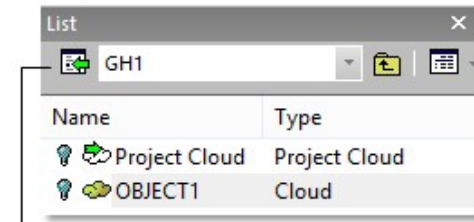


The WorkSpace window in Registration

Note: (*) The **Modeling** processing mode is not present in **RealWorks (Base)** and **Advanced**.

List

This window, always opened when you start **RealWorks**, is located under the **WorkSpace** window. It is used to display the content of a selected group of the **Project Tree**. When a **RealWorks** session begins, this window has the name **List**. Once you load a project and select a group of the **Project Tree** under the displayed tab, this window will rename itself according to the contents of the displayed tab. Each object node shown in this window is identified by its icon, its name and its other properties. The **List** window contains a toolbar called **Database** below the title bar. You can move the **List** window to any location in the user interface, or close, reduce and restore it.



The Database toolbar

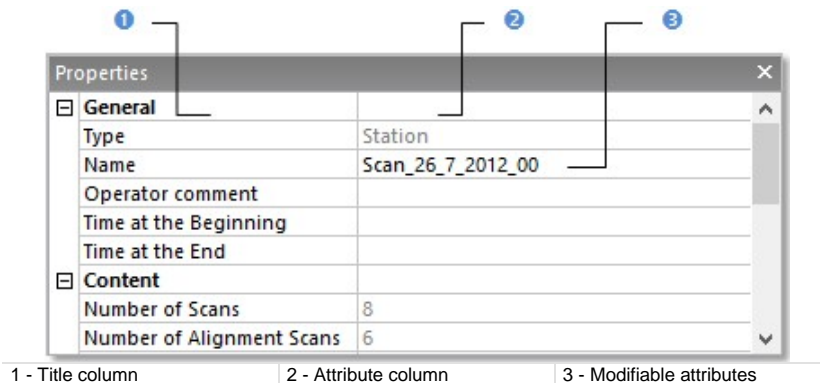
Note: Because the number of points for a point cloud is often large and the symbol (**Digit Grouping Symbol**) used for grouping the digits makes in that number unreadable; you now can customize this symbol by first opening the **Regional and Language Options** in the **Control Panel** (of **Windows®**) and then selecting **Customize** and **Digit Grouping Symbol**. This change will be memorized and used for the next session of **RealWorks** and will affect the display of the numbers of points in that window.

Caution: The notion of **Number of Loaded Points** in the **List** window has been removed in **RealWorks 8.0**.

Property

This window is used to list property information of a selected object and is divided into two columns. The left one lists the titles of each property and the right one shows property values (either fixed or modifiable). Properties shown in gray are fixed and those in black are modifiable. Properties are classified by category such as **General**, **Content**, **Geometry**, etc.

You can shrink each category of properties by hiding its content. To do this, left-click on the **Shrink** button. By default, the **Property** window is not displayed in the user interface. You have to open it by using the command from the main **Window** menu or from the pop-up menu after selecting an object. You can move this window to any location in the user interface, or close, reduce and restore it.



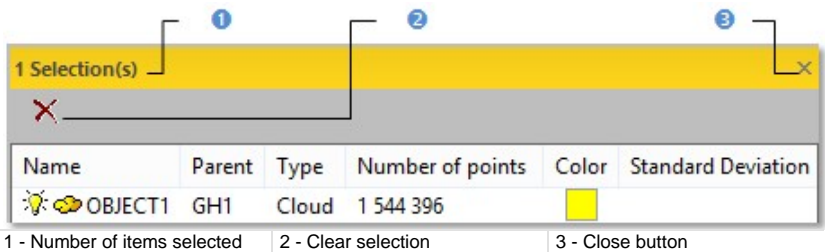
Note: Because the number of points for a point cloud is often large and the symbol (**Digit Grouping Symbol**) used for grouping the digits makes in that number unreadable; you now can customize this symbol by first opening the **Regional and Language Options** in the **Control Panel** (of **Windows®**) and then selecting **Customize** and **Digit Grouping Symbol**. This change will be memorized and used for the next session of **RealWorks** and will affect the display of the numbers of points in the **Property** window.

Tip: You can select and copy any value from the **Property** window by using the **Ctrl + C** keys.

Caution: The notion of **Number of Loaded Points** in the **Property** window has been removed in **RealWorks 8.0**.

Selection List

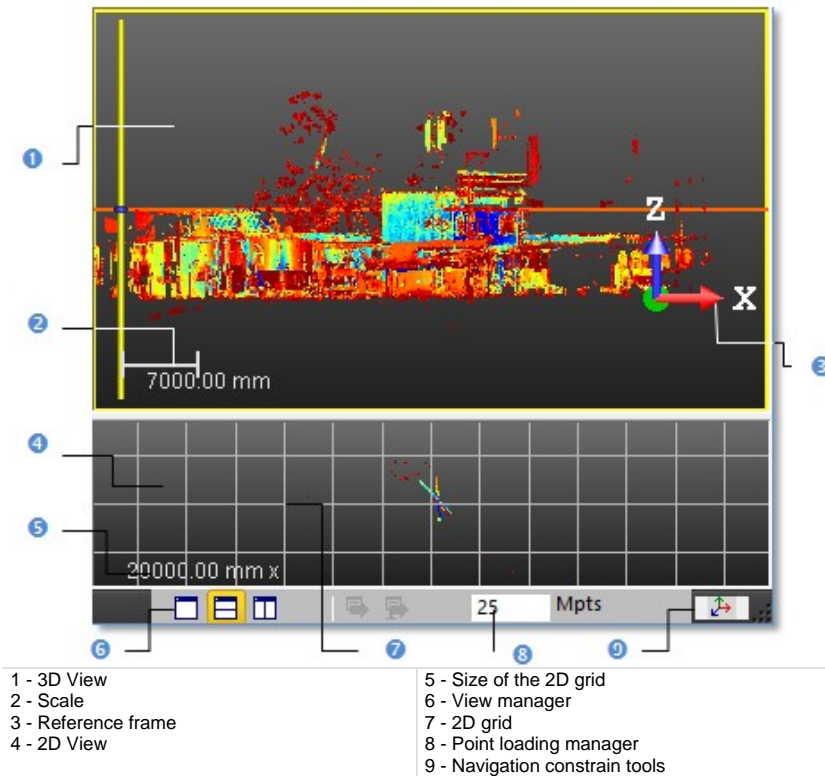
The **Selection List** window is used to list all selections done from the **Project Tree** or **from the 3D View**. By default, the **Selection List** window is not displayed within the user interface. You have to open it by using the command from the **Window** menu. You can move this window to any location in the user interface, or close it.



Caution: The notion of **Number of Loaded Points** in the **Selection List** window has been removed in **RealWorks 8.0**.

3D View

The **3D View** is always open at the right side of the user interface under the toolbars. You can move, reduce and restore it in the same way as for the other windows of **RealWorks** but you cannot close it. This window is mainly used for displaying (or hiding) the 3D representation of a selection from the **Project Tree**. An orthonormal **Reference Frame** and a scale are displayed respectively at the bottom right and bottom left corners of this window. By default, the window background is blue with a gradient effect. You can customize it to suit your preference.



View Manager





RealWorks includes a **View Manager** that enables to navigate through the different aspects that may have the **3D View** when using a certain type of tools. The **3D View** can be split in two or three sub-views. Each sub-view can be a **3D View**, a **3D View** locked in 2D or graphs. This **View Manager** appears as a toolbar at the bottom right corner of the **3D View** and is composed of two sets of icons.

Note: In the sub-view mode (two horizontal sub-views, two vertical sub-views or three sub-views), the sub-view with a yellow edge is the active sub-view. In the one view mode, the view in full is always the active view.

Tip: Almost all the icons can be reached from the toolbar at the bottom right corner of the **3D View** or by selecting **Layout** from the **3D View** menu.






Changing Sub-View Configuration

The first set is composed of the icons described in the table below. It enables to change the configuration of the two or three sub-views as follows:

This icon	Enables
 Make Full	To expand the selected sub-view to full view.
 Split Horizontally	To tile two sub-views horizontally
 Split Vertically	To tile two sub-views vertically
 Split 3 Views	To tile into three sub-views

Hiding and Displaying Sub-Views

The second set is composed of the icons described in the table below. It enables to hide or to expand in full a sub-view. It also enables to restore the default layer.

This icon	Enables
 Display Main View	To display the main 3D View in full
 Display Sub-View 1	To display the first planar sub-view in full
 Display Sub-View 2	To display the second planar sub-view in full.
 Restore Default Layout	To restore the default layout.
 Hide View	To hide the current view.

Notes:

- The **Hide View** icon can only be selected from the pop-up menu (or from the **3D View / Layout** menu).
- A sub-view once expanded in full cannot be hidden. That's why the **Hide View** icon is dimmed.

Two Sub-View Configuration

In that configuration, only **Make Full**, **Split Horizontally** and **Split Vertically** are available. When a sub-view has been selected and expanded in full, the **Display Main View** and **Display Sub-View 1** icons become enabled.

Three Sub-View Configuration

In that configuration, all are available: **Make Full**, **Split Horizontally**, **Split Vertically** and **Split 3 Views** are available. When a sub-view has been selected and expanded in full, the **Display Main View**, **Display Sub-View 1** and **Display Sub-View 2** icons become enabled.

2D Grid

In the **2D View** mode, there is by default a **2D Grid** superposed on the current sub-view. This grid helps the user to have a metric scale of objects displayed within the sub-view.

Hiding/Showing

To Hide (or Show) the 2D Grid:

1. From the **3D View** menu, select **2D Grid**. A sub-menu drops down.
2. Select **Hide 2D Grid** from the sub-menu to hide the 2D grid.
3. Or select **Show 2D Grid** from the sub-menu to display the 2D grid.

Tip: You can right-click in the sub-view and select **Hide 2D Grid** (if the **2D Grid** is displayed) or **Show 3D Grid** (if the **2D Grid** is hidden).

Notes:

- Hiding the **2D Grid** from the current sub-view will display the scale.
- Whatever the view (**3D View** or current sub-view) you select, you should be in **Isometric** to be able to display the scale.

Changing a Size

There are seven pre-defined and square sizes: **0.1x0.1**, **1x1**, **5x5**, **10x10**, **50x50**, **100x100** and **1000x1000**. All are expressed in the current unit of measurement. The current size is displayed at the bottom left corner of the sub-view.

To Change a Size:

1. In the **3D View** menu, select **2D Grid**. A sub-menu drops down.
2. Select a pre-defined size from the sub-menu.

Tip: You can right-click anywhere in a sub-view and select **2D Grid** from the pop-up menu. A sub-menu drops down from which you can select a pre-fined size.

Customizing a Size

You can define a size which be either square (the same resolution in Horizontal and in Vertical) or not.

To Customize a Size:

1. In the **3D View** menu, select **2D Grid**. A sub-menu drops down.
2. Select **Customize** from the sub-menu. The **2D Grid Size** dialog opens.
3. Input a distance value in the **Horizontal Size** field.
4. Input a distance value in the **Vertical Size** field.
5. Click **OK**. The **2D Grid Size** dialog closes.

Tip: You can right-click anywhere in a sub-view and select **Customize** from the pop-up menu.

Navigation Constraint Tools

When you manipulate a scene in the **3D View**, you can apply a set of constraints like e.g. rotating it horizontally. All available constraints are split into two categories: temporary constraints or permanent constraints.

Note: We will not discuss in detail these two constraint modes here. For more information see the Navigation Modes section.

Tools and Commands

Commands are actions which can apply to the selected object(s) or to a whole set of displayed objects, while tools are a set of actions logically organized together to fulfill a function of the software. All tools and commands can be found in the menu bar, and most of them can also be found in various toolbars. In the next topics, we will discuss in more detail the organization of the tools and commands in the menu bar and the toolbars.

Toolbars

Icons are the graphic representation of tools and commands in the toolbars. They are organized into different toolbars according to their similarity in terms of function.

Main

Under the **Main** toolbar, you can find the following list of icons:



Open



Connect to Mobile Device



Open Scan Explorer



Save



Print



Undo



Redo

Database

Under the **Database** toolbar, you can find the following list of icons:



Locate



Go Up To Parent



Thumbnails



Icons



Details

Display

Under the **Display** toolbar, you can find the following list of icons:



Display Cloud



Display Geometry



Hide Cloud



Hide Geometry





























Hide All



Lighting Direction

3D View

Under the **3D View** toolbar, you can find the following list of icons:

-  White Color
-  Cloud Color
-  Station Color
-  Scan Color
-  Grey Scaled Intensity
-  Color Coded Intensity
-  True Color
-  Wireframe
-  Hidden Lines
-  Surface
-  Textured
-  Point Size > 1 Pixel
-  Point Size > 2 Pixels
-  Point Size > 3 Pixels
-  Point Size > 4 Pixels
-  Point Size > 5 Pixels
-  Normal Shading
-  Perspective
-  Isometric
-  Examiner
-  Walkthrough
-  Station-Based
-  Display Stations
-  Display Station's Labels
-  Change Cloud Color
-  Change Shape Color









View Alignment

Under the **View Alignment** toolbar, you can find the following list of icons:

-  Zoom On Selection
-  Center On Point
-  Zoom All
-  Zoom In
-  Zoom Out
-  Front
-  Back
-  Left
-  Right
-  Top
-  Bottom
-  Front of Selected Object
-  Back of Selected Object
-  Left of Selected Object
-  Right of Selected Object
-  Top of Selected Object
-  Bottom of Selected Object
-  Go to Shooting Position

Tools [in Registration]

Here is a list of icons that you can find in the **Tools** toolbar when you are in the **Registration** mode:

-  Auto-Extract Targets and Register
-  Auto-register using Planes (Target-Less)
-  Target-Based Registration Tool
-  Refine Registration Using TZF Scans
-  Cloud-Based Registration Tool
-  Georeferencing Tool
-  Target Analyzer Tool
-  Orientation Tool
-  Generate Key Plan From Current View
-  Registration Report (Target-Based)
-  Measurement Tool












Tools [in OfficeSurvey]

Here is a list of icons that you can find in the **Tools** toolbar when you are in the **OfficeSurvey™** mode:

-  Show Manipulator Tool
-  Segmentation Tool
-  Sampling Tool
-  Cutting Plane Tool
-  Contouring Tool
-  Profile/Cross-Section Tool
-  EasyProfile Tool
-  2D-EasyLine™ Tool
-  Polyline Drawing Tool
-  2D-Polyline Inspection Tool
-  Volume Calculation Tool
-  Twin Surface Inspection Tool
-  Surface To Model Inspection Tool
-  Inspection Map Analyzer Tool
-  Fitting Tool
-  Mesh Creation Tool
-  Mesh Editing Tool
-  Ortho-Projection Tool
-  Multi-Ortho-Projection Tool
-  Image Rectification Tool
-  Feature Set Tool
-  Measurement Tool
-  Image Matching Tool
-  Profile Matcher Tool
-  3D Inspection Tool
-  3D Inspection Analyzer

Tools [in Modeling]

Here is a list of icons that you can find in the **Tools** toolbar when you are in the **Modeling** module:

-  Segmentation Tool
-  Sampling Tool
-  Measurement Tool
-  Cloud-Based Modeler Tool
-  Geometry Creator Tool
-  Geometry Modifier Tool
-  Intersect Tool
-  Duplicator Tool
-  Plane Bounding Tool
-  EasyPipe Tool
-  SteelWorks Creator Tool

Window

Here is a list of icons that you can find in the **Window** toolbar:

-  Tree Window
-  List Window
-  Property Window
-  Selection List Window
-  Cascade
-  Tile Vertically
-  Tile Horizontally

Working Frame Tool

Here is a list of icons that you can find in the **Working Frame Tool** toolbar which only appears when you select the related command:

-  Frame Creation Tool

Menu Bar

The menu bar is a series of aligned menu titles. This series changes depending on the processing mode you are in. You drop down a menu by clicking on its title, and then you can select a command.

- **File:** This menu is a standard menu that contains **Open**, **Close**, and other file related commands. This menu also contains tools for loading Trimble FX controller files, converting TZS to BSF, etc. It is available in the **Registration** mode.
- **Edit:** This menu gives access to object editing operations such as cut, copy and paste, undo, redo, find, etc. and access to advanced functions and preferences.
- **Tools:** This menu contains common tools that you can use no matter which processing mode you are in.
- **Plant*:** This menu contains tools for modeling pipes and structural steelworks.
- **OfficeSurvey:** This menu contains all available tools related to the **OfficeSurvey™** mode.
- **Registration:** This menu contains all available tools related to the **Registration** mode.
- **Modeling*:** This menu contains all available tools related to the **Modeling** module.
- **Media Tools:** This menu contains tools for computing video and capturing snapshots.
- **Display:** This menu gives access to object display and hide functions in the **3D View** window.
- **3D View:** This menu gives access to different visualization parameters (rendering, view, etc.) in the **3D View** window.
- **Window:** This menu allows user to organize the user interface.
- **Help:** This menu gives access to online help.

Note: (*) The **Plant** and **Modeling** menus are not present in **RealWorks Base**.

Shortcut Keys

You can use shortcuts to carry out the following common tasks in **RealWorks**.

Press:	To:
CTRL + O	Open a file
CTRL + S	Save a project
CTRL + Z	Undo the last action
CTRL + Y	Redo the last action
CTRL + X	Cut the selected object
CTRL + C	Copy the selected object
CTRL + V	Paste the selected object
Del.	Delete the selected object
CTRL + F	Access to the Find function
CTRL + M	Merge two selected clouds
Home	Zoom all

You can use the following shortcuts when the view manager toolbar opens at the bottom of the **3D View** window.

Press:	To:
F11	Set the active sub-view in full mode
CTRL + F11	Replace the current view by the next one (only available in full mode)
SHIFT + F11	Hide all open windows except the 3D View window or display them if hidden

Customizing the User Interface

By default, all windows except the **Property** and **Selection List** windows are open within the user interface. Toolbars are also open and are displayed either horizontally under the menu bar or vertically along the left side of the **3D View**.

Windows

You can display (or hide) any window as required. Note that you cannot close the **3D View** but only reduce (or resize) it.

Displaying

To Display a Window:

1. Select **Window** from the menu bar.
2. Select a window (to display) from the drop-down menu.


Tip: You can right-click anywhere on any open toolbar and select a window (to display) from the pop-up menu by checking it.

Hiding

To Hide a Window:

1. Select **Window** from the menu bar.
2. Select a window (to hide) from the drop-down menu.

Tips:

- You can also click on the **Close**  button or right-click on the **Title Bar** (of a **Floating** window) and select **Hide** from the pop-up menu.
- You can right-click anywhere on any open toolbar and select a window (to hide) from the pop-up menu by unchecking it.

Note: An open window has its icon highlighted in the drop-down menu.

Undocking

To Undock a Window:

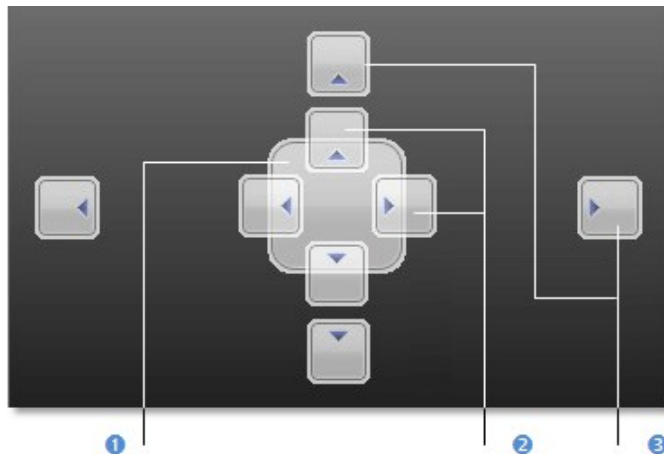
1. Move the pointer somewhere over an area of the window (to undock). A good place to point is the title bar.
2. Press and hold the mouse button while you drag the window to a suitable location in your working environment.
3. Release the mouse button to drop the window to its new location. It will remain in this new place until you move it again (or close it).

Tip: A window, once moved from its **Docking** position, becomes **Floating**. You can check its status by right-clicking on the **Title Bar**.

Docking

To Dock a Window:

1. Move the pointer somewhere over an area of the window (to dock). A good place to point is the **Title Bar**.
2. Press and hold the mouse button while you drag the window. A diamond guide appears.



1 - Guide Diamond

2 - Four arrows pointing toward the four sides of the editing panel

3 - Four additional arrows pointing toward the four edges of the user interface

3. When the window reaches the location to dock, move the pointer over the corresponding portion of the guide diamond. The desired area is then shaded.
4. Release the mouse button to drop the window to its new location. It will remain in this new place until you move it again (or close) it.

Tip: You can also right-click on the **Title Bar** of an undocked window and select **Docking** from the pop-up menu. The window is then re-docked to its initial position.

Toolbars

You can display (or hide) any toolbar as required.

Displaying

To Display a Toolbar:

1. In the **Window** menu, select **Toolbars**.
2. Select a toolbar (to display) from the drop-down submenu.

Tip: You can right-click anywhere on any open toolbar and select a toolbar (to display) from the pop-up menu.

Hiding

To Close a Toolbar:

1. In the **Window** menu, select **Toolbars**.
2. Select a toolbar (to hide) from the drop-down submenu.

Tip: You can right-click anywhere on any open toolbar and select a toolbar (to hide) from the pop-up menu.

Moving

To Move a Toolbar:

1. Move the pointer somewhere over an area of the toolbar that does not display a button (or drop-down list). A good place to point is the title bar.
2. Press and hold the mouse button while you drag the toolbar to a suitable location in your window.
3. Release the mouse button to drop the toolbar to its new location. It will remain in this new place until you move it again or close it.

CHAPTER 3

Organization of Data

In **RealWorks**, a project contains original scanned data and images, and all objects created from the scanned data. In order to make such data visible to users, we organize them into a **Project Tree** under **WorkSpace**.

Project Data

Each **Project Tree** is composed of four sub-trees called **Scans**, **Models**, **Targets** and **Images**. At any given time, only one of them is displayed.

When a project is loaded into **RealWorks**, it is immediately inserted under the **WorkSpace** as a named project. Under the project, you can find two types of node called **Group** node and **Object** node. An object node is always a leaf node, while a group node could be either an internal or a leaf node. The organization and manipulation of the group and the object nodes in the **WorkSpace** window and the **List** window are similar to those of the respective **File** and **List** windows of Microsoft Windows Explorer.

Scans Tree

The **Scans Tree** is the first sub-tree of the **Project Tree**. It is only available in the **Registration** processing mode. You can display it by selecting its related tab from the **WorkSpace** window. This tree is used for organizing scanning results called **Stations**, **Scans**, or **Images**. It may have as many levels as a project requires. **Scans** and **Images** are always the leaves of the tree, while stations are internal nodes. It is important to note that the content of a **Station** (**Scans** or **Images**) cannot be moved to other stations, nor can their position be changed inside a station. This is to preserve the scanning order.

Caution: Images from the **Scans Tree** cannot be deleted and scans from which all points have been deleted are erased.

Targets Tree

The **Targets Tree** is the second sub-tree of the **Project Tree** in the **Registration** processing mode. You can display it by selecting its related tab from the **WorkSpace** window. It is used for organizing the registration entities (**Targets**, **Survey Points**, **Topo Points**, etc.) matched or not.

Note: Refer to the **Registration** section for more details on the exact definition of the registration entities and how they are organized and used.

Models Tree

The **Models Tree** is the first sub-tree of the **Project Tree**; it is available in both the **OfficeSurvey™** and **Modeling** processing modes. You can display it by selecting its related tab from the **WorkSpace** window. This sub-tree is used for organizing models of a scene. The organization can be logical, spatial or discipline-based (or simply a combination thereof), depending on the purposes of a project.

The **Models Tree** may have as many levels as a project requires. You can create, re-organize, delete, browse, search, locate or visualize objects in this sub-tree. Each object node of this sub-tree may contain a point cloud, a geometry or both. We call them the two representations of this object. By default, only one representation is displayed:

- A cloud object is displayed by its cloud representation,
- A shape object is displayed by its geometric representation,
- By default, an object with both representations is displayed by its geometric shape representation. The user must explicitly ask to display its cloud representation.

For a project saved in a version of **RealWorks** older than 8.0, there is at least one **Sub-project** attached at the root of the **Models Tree** and only one is active at a time (the one in bold). After saving an old project in 8.0, all **Sub-projects** are replaced by groups named "From "Sub-Project" name".



1 - The Models Tree in projects older than 8.0.

2 - The Models Tree in projects saved in 8.0.

Note: A project created and saved directly in 8.0 has no notion of groups (coming from **Sub-Projects** conversion).

Images Tree

The **Images Tree** is the fourth and last sub-tree of the **Project Tree**. You can display it by selecting its related tab from the **WorkSpace** window whatever the processing mode - **OfficeSurvey™**, **Registration** and **Modeling** - you are in. This tree is used to organize (or browse) images taken by a laser scanner's on-board video camera (or other digital cameras). For example, you can group together a set of images and use them for texturing a part of the scene.

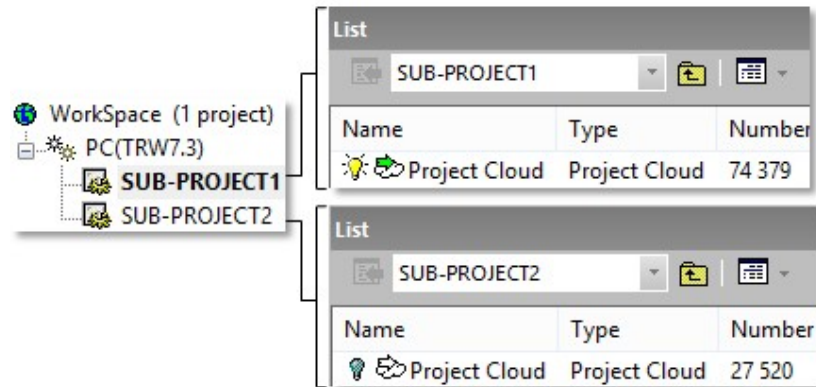
Note: You can view an image (or group of images) as thumbnail(s) in the **List** window either by selecting first **Database Browsing**, then **Thumbnails** from the **Window** menu or by clicking on its corresponding icon in the **Database** toolbar.

Project Cloud

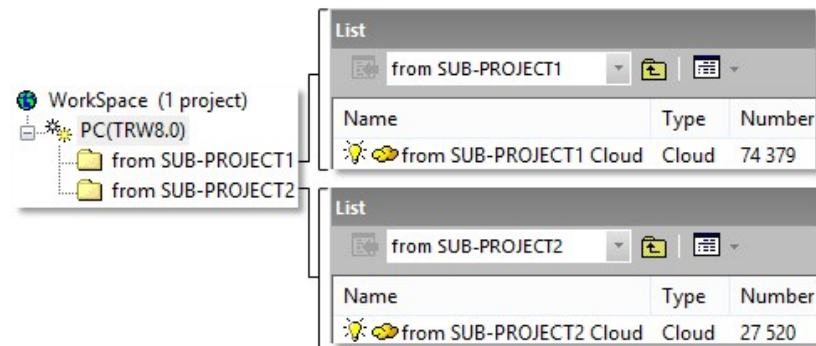
A **Project Cloud** is a cloud node attached to the **Models Tree** in either the **OfficeSurvey** processing mode or the **Modeling** processing mode. The aim of the **Project Cloud** is to allow you to quickly find all points (or all unused points) of the project.

Note: The contents of the **Project Cloud** are automatically displayed in the **3D View** after getting all points or getting remaining points.

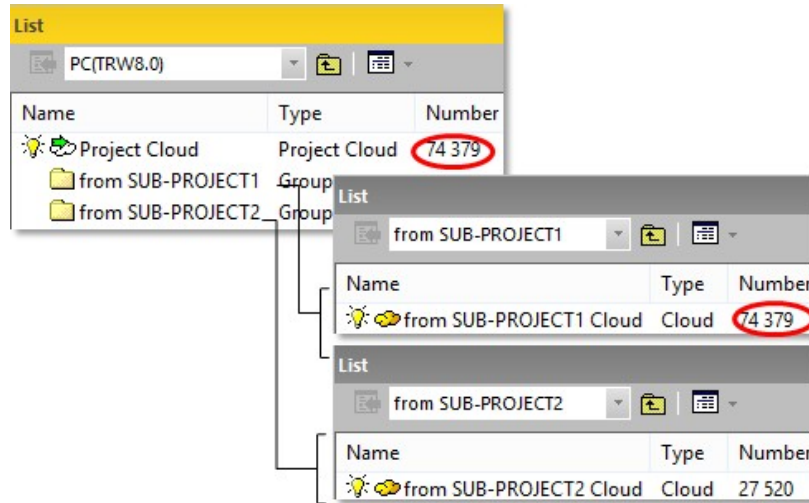
Before 8.0, each **Sub-project** of a project has its own **Project Cloud**. Only points belonging to the **Project Cloud** of the active **Sub-Project** are loaded.



After saving (a project) in 8.0, the **Project Cloud** of each **Sub-project** is converted to a **Cloud** with the same number of points. This **Cloud** has as name "From 'Sub-Project' name Cloud".



The newly saved project has a unique **Project Cloud** with the same number of points as the **Project Cloud** of the active **Sub-Project** of the old project.



Getting all Points

When you load a project for the first time, the **Project Cloud** of the project is empty. You have to load it with points. You have the choice of getting all points or only unused ones. Once the **Project Cloud** is loaded with all points, you can see the number of points in the corresponding attribute column.

To Get all Points:

1. Load first a project in **RealWorks**.
2. Select a project under the **WorkSpace**.
3. Select **Project Cloud** in the **List** window.
4. From the **Edit** menu, select **Get All Points**.

Tip: You can also select the **Project Cloud** from the **Project Tree**, right-click to display the pop-up menu and then select **Get All Points**.

Note: When you create a new project by importing **TZF** format file(s), you may notice the number of points in the **Project Cloud** is equal to zero. This number remains in this state until you perform your first extraction (of points) from the **TZF Scan(s)**.

Getting Remaining Points

After you have segmented points of a project and organized them into different groups, there may remain some non-segmented and/or un-organized points. At various moments, you may need to find/display these points. Once the **Project Cloud** is loaded with all unused points, you can see the number of points in the corresponding attribute column.

To Get Remaining Points:

1. Load first a project in **RealWorks**.
2. Select a project under the **WorkSpace**.
3. Select **Project Cloud** in the **List** window.
4. From the **Edit** menu, select **Get Remaining Points**.

Tip: You can also select a **Project Cloud** from the **Project Tree**, right-click to display the pop-up menu and then select **Get Remaining Points**.

Active Group








An **Active Group** is a group that you have selected. Note that an active group can only be a project, a station, a group of models or a group of images. Selecting an active group can be done in the **Project Tree** or in the **List** window. By default, all new created objects will be put under this active group.

Group and Object Nodes

The user will find information related to the identification of **Group** and **Object** nodes in the **WorkSpace** and **List** windows.

In the WorkSpace Window

Each item of data displayed in the **WorkSpace** window is identified by its icon, name and order in the **Project Tree**. Below is a list of icons appearing in the four sub-trees.






-  WorkSpace
-  Project
-  Project, Opened
-  Unmatched target folder
-  Matched target folder
-  Station
-  Group of objects

In the List Window

Each item of data displayed in the **List** window is identified by its icon, name, attributes and order in the **Project Tree**. Lists given hereafter are not exhaustive and are given only as a guide.



















Scans Tab

Here is a list of icons that you can find in this window when you select the **Scans** tab.

-  Station, Unleveled
-  Station, Leveled
-  Station, Leveled and Setup
-  Scan
-  Image
- Etc.

Models Tab

Here is a list of icons that you can find in this window when you select the **Models** tab:

-  Project cloud
-  Group of models
-  Model as cloud
-  Model of box shape
-  Model of cylinder shape
-  Model of fitted cylinder shape
-  Model of plane shape
-  Model of fitted plane shape
-  Model of circular torus shape
-  Model of fitted circular torus shape
-  Model of sphere shape
-  Model of regular cone shape
-  Model of fitted Polyline
-  Model of fitted composite curve
-  Model of fitted mesh
-  Model of point-to-point distance measure
-  Model of angular measure
-  Model of 3D point measure
- Etc.

Images Tab

Here is a list of icons that you can find in this window when you select the **Images** tab:



Image, Imported



Image, Matched



Image, OrthoPhoto

Targets Tab

Here is a list of icons that you can find in this window when you select the **Targets** tab:



Target, Spherical



Target, Trimble Flat



Target, Fitted Sphere

Etc.

CHAPTER 4


Basics of Operation

This chapter is dedicated to the basic operations of opening and importing of project files.

Opening Projects

A project file can be opened by using the **Open** dialog or using the **Recent Files** in the **File** menu. The ten last opened files are listed at the bottom of the **File** menu. As a shortcut, you can open any of them just by selecting it in this menu.

To Open a Project File:

1. From the **File** menu, select **Open** . The **Open** dialog opens with the **Add to Project** option dimmed*.
2. Select a file type from the **File of Type** field.
3. Navigate to the drive/folder where the file is located.
4. Click on the file to select it. Its name appears in the **File Name** field.
5. Click **Open**. The **Open** dialog closes.

Tip: You can use the shortcut key **Ctrl + O** (or click **Open** in the **Main** toolbar) to pop-up the **Open** dialog.

You can also drag and drop to open a project file into **RealWorks**. If **RealWorks** is not already open, this operation will open it. Only one project file can be dragged and dropped at a time. If it is already open, you can drag and drop a set (of project files).

When a set of project files is opened through the **File / Open** menu, a project and a set of stations (one per project file) are created and rooted under the **Project Tree**. When that set has been dragged and dropped in, a set of projects (one per project file) and a unique station (related to the project file) are then created.


Notes:

- For files of certain types, you cannot drag and drop a set of projects into **RealWorks** when there is no opened session.
- Projects are ranked by alphabetic order in the **Project Tree** in the **Workspace**. They are ranked from their opening order in the **List** window.
- You can abort the opening of a project by pressing **Esc**.
- When you open a project previously saved in **RealWorks** format or in **PointScape** format or in **JobXML** format for which images are missing, a warning message appears and all missing images are listed.

Importing Projects

A project file can be imported into an existing project by using the **Open** dialog.

To Open a Project File:

1. From the **File** menu, select **Open** . The **Open** dialog opens with the **Add to Project** option dimmed*.
2. Select a type of file from the **File of Type** field.
3. Navigate to the drive/folder where the file is located.
4. Click on the file to select it. Its name appears in the **File Name** field.
5. Keep the **Add to Project** option checked.
6. If there are several projects, click on the pull-down arrow.
7. Choose a project from the drop-down list.
8. Click **Open**. The **Open** dialog closes.

Tip: You can use the shortcut key **Ctrl + O** (or click **Open** in the **Main** toolbar) to pop-up the **Open** dialog.

Connecting to Mobile Device

A fast way to open (or import) a file from a Trimble data collector such as a Recon™, TCU™ or TSC2™ in **RealWorks** is to connect and synchronize the Trimble data collector with a desktop computer (or laptop). Only a file of **RAW** (from the Trimble Survey Pro™), **JOB** (from the Survey Controller™ software) and **JobXML** (from the Trimble Survey Controller™, Survey Manager™ or Survey Pro™ software) extensions can be opened (or imported) in that way.

Microsoft® ActiveSync® is a software program that comes with your data collector when you purchase it. This program allows you to synchronize the information on your data collector with the information on your desktop computer (or laptop). Synchronization is done by comparing data between these two computers and updates both of them with the most recent information. ActiveSync® is already integrated into the operating system on your data collector. However, you must install ActiveSync® on your desktop computer (or laptop). You can install the software from the CD that was shipped with your data collector or you can download the current version of ActiveSync® from the Microsoft® website.

To Connect to a Mobile Device:

1. Connect a Trimble data collector to your desktop computer (or laptop). For more details, please refer to the documentation that comes with your data collector.
2. Power the Trimble data collector On.
 - If there is no project open, the **Connection to Mobile Device** dialog opens and the **Add to Project** option is grayed out. You are restricted to opening a file.
 - a) Navigate to the drive/folder where the file is located.
 - b) Click on the file to select it. The **Open** button becomes active.
 - c) Click on the **Open** button.
 - If there is one or more projects open in **RealWorks**:
 - a) From the **File** menu, select **Connect to Mobile Device**. The **Connection to Mobile Device** dialog opens and the **Add to Project** option is available and default checked.
 - b) Navigate the Drive / folder where the file is located.
 - c) Click on the file to select it. The **Open** button becomes active.
 - d) Keep the **Add to Project** option checked.
 - e) Click on the pull down arrow and choose a project from the drop down list.
 - f) Click on the **Open** button.

Tip: Instead of selecting **Connect to Mobile Device** from the **File** menu, you can also click on it related icon in the **Main** toolbar.

Note: The path to the file to open (or import) as well as the dialog box size are persistent. This means that they remain unchanged till the user changes them.

Project File Formats

RealWorks supports a numerous of file formats. There are those that are Trimble's (software) proprietary formats, those coming from Trimble's instruments (or from competitors), those generating from a third-party software, etc.

Trimble 3D Scanning Files

A **Trimble 3D Scanning File** is a file with one of the following extensions: ***.rwp**, ***.raw**, ***.jxl**, ***.asc**, ***.neu** and ***.tzf**. Among all these file formats, some can be imported into an existing project and others cannot. A file of the following extension **rwp** cannot be imported. Only those of **asc**, **neu** and **tzf** extensions or those coming from the Trimble **Survey Controller™**, **Survey Manager™** or **Survey Pro™** software (JobXML and related) can be imported.

Below are listed all the extensions and the application from which each extension is from.

- **RWP** Trimble proprietary **RealWorks** project file format.
- **ASC** a well-known **ASCII** coordinates files.
- **NEU** a Neutral file format identical to **ASC**.
- **TZF** Trimble Scan File format file,
- Etc.

Note: There is no direct entry for a file with the **RAW** extension in the "Type of File" field in the **Open** dialog. You need to choose **Trimble 3D scanning File** as format if you have a file with one of the extensions.

RealWorks Files

RWP is a proprietary format of Trimble. It is a project file format. The **RWI** folder is a folder linked to the **RWP** format file. It contains all data files of a project (**RWC** and **RWV** for versions of **RealWorks** before 8.0 and **RWCX** and **RWV** for **RealWorks** 8.0). **RWC** and **RWCX** are cloud format files. **RWV** is an image format file.

A project file saved in a version of **RealWorks** older than 5.0 is not supported anymore in 8.0. When you try to open such a file, an error message appears.

When you open a project saved in a version of **RealWorks** older than 8.0, all the project files are converted to the 8.0 format. The conversion takes a certain amount of time and is temporary.

If you decide to save the project under the same name (after converting), the conversion becomes definitive and cannot be cancelled. The project will be not accessible with older versions of **RealWorks**. The conversion (of all project files to 8.0 format) is only required one time. The next time you want to open the project, no conversion is required and the loading of the project is accelerated.

If you decide to not save the project, the conversion is not applied and is lost (if you close **RealWorks**). The next time you open the project in 8.0, the conversion is required again.

Caution: If a cloud format file (**RWCX**) is missing in the project you attempt to open, an error message appears and the project will not be open.

TZF Files

TZF is a proprietary format of Trimble. A file with such a format is essentially a **Trimble Scan File**.

- A project and a station* will be created and rooted under the **Project Tree**. The project is named **ProjectX** where **X** is its order. The station takes the name of the **TZF** format file.
- A **TZF Scan** is also created and put under the station.
- The project is not saved. The user has to save it manually.
- Once saved, a project file and a folder are created. Both are named according to the name given by the user, with a **RWP** extension for the first and a **RWI** extension for the second. The **RWI** folder is empty of content.


Note: The processing mode will automatically switch to **Registration**. The **Scans Tree** is selected by default.

Tip: A **TZF** format file can be either opened as a single project or imported into an existing project.

Note: (*) A **Leveled Station** is created and rooted in the **Scans Tree** for each **TZF** format file tagged as **Leveled**, once open (or imported) into **RealWorks**.

You can preview a **TZF** format file as a **Thumbnail** in **Windows Explorer**. You need to first enable the **Thumbnail Preview** in **Windows Explorer** and then to set the icon view size to **Medium Icons**, **Large Icons** or **Extra Large Icons**.

Some new information has been added to **TZF** format files, in **RealWorks 8.0**. This information, related to the instrument itself and to the scanning settings, like **Starting/Final Temperature** (Internal) (in **Celsius** and **Fahrenheit**), Atmospheric Correction **PPM** (Parts **Per Million**), **Grid Steps**, **Scanner Levelling** and etc. appear when you display the properties of a **TZF Scan** (only if the **Property** window is open).

Properties	
[-] General	
Type	TZF Scan
Name	Scan 02
Number of Points	25 440 867
[-] Scan Information	
File Path	F:\04 - Trimble Testing Files\Multi
Type	Full Scan
Density	Level 1
Date of Creation	04/06/2013 11:12:35
Date of Completion	04/06/2013 11:14:22
Operator Name	
Starting Scan Temperature (Internal)	23.4°C / 74.1°F
Final Scan Temperature (Internal)	
PPM	
Instrument Name	
Instrument Serial Number	94500304
Instrument Firmware Version	Trimble TX8 2013-00 (1058)
Percentage of Scan Completion	100%
Warning during Scan	No Warning
Horizontal Grid Step	753.982 µRad; 23mm @ 30m / 0.7
Vertical Grid Step	753.982 µRad; 23mm @ 30m / 0.7
Scanner leveling	True
[-] Others	
Image Size (w x h)	34 719 444 (8334 x 4166)
Preview	

TZS Files

A **TZS** format file is a **Trimble Scan File**. For each **TZS** format file, a warning dialog appears and prompts you to proceed to the conversion to the **TZF** format (or not).

Notes:

- The processing mode will automatically switch to **Registration**. The **Scans Tree** is selected by default.
- A **TZS** format file can be either opened as a single project or imported into an existing project.

Caution: You cannot open **TZF** format files in **RealWorks** 7.0 or lower.

Refusing to Convert to the TZF Format

To Refuse to Convert to the TZF Format:

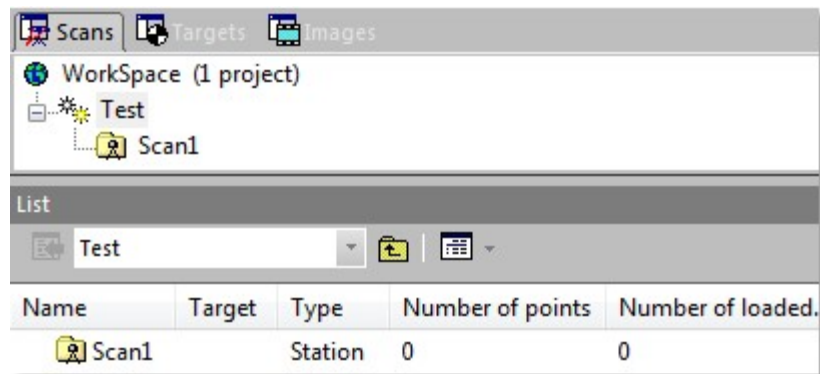
- In the warning dialog, click **No**. If there is an open project, the project will close. If there is no open project, nothing happens (no project is created).

Converting to the TZF Format

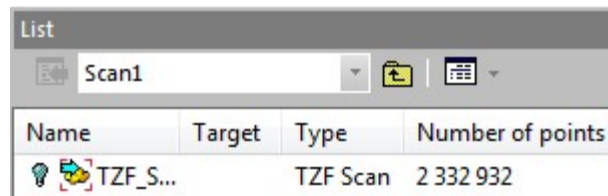
To Convert to TZF Format:

1. In the warning dialog, click **Yes**. An Information dialog appears.
2. Click **OK**. The **Save** dialog opens. The default folder is the folder where the **TZS** format file is.
3. Keep the default folder.
4. Or navigate to a drive/folder where you want to store the project.
5. Keep the default name which is **ProjectX**.
6. Or input a name in the **File Name** field. The **RWP** extension is automatically added.
7. Click **Save**. The conversion is then performed.

Once completed, a project and a station are created and rooted under the **Project Tree**. The project has the name given by the user. The station has the name of the **TZS** format file.



A **TZF Scan** is created is under the station.



A project file and a folder are created. Both are named according to the name given by the user, with a **RWP** extension for the first and a **RWI** extension for the second. Under the **RWI** folder, a scan file with the **TZF** extension is also created.

Trimble Survey Project Files

A **Trimble Survey Project**, once created and saved in **Trimble Business Center** (2.5 and above), is composed of a **TSPX** format file, a **SDF** format file, two **VSE** format files and a folder which contains the JXL format file, images and TSF format files. Such project is shared by **Trimble Business Center** and **RealWorks**. This lets the user to combine the survey capabilities of **Trimble Business Center** with the manipulation capabilities of **RealWorks** for processing scanned data and then manipulating them.

Once a **TSPX** format file has been open in **RealWorks**, a **RealWorks** project file (with **RWP** as extension) and **RWI** folder are created and are added to the previous list of files.

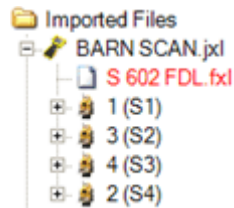
Notes:

- Any modification done within **RealWorks** is not taken back into **Trimble Business Center**.
- Only one **Trimble Survey Project** can be open in **RealWorks** at once.

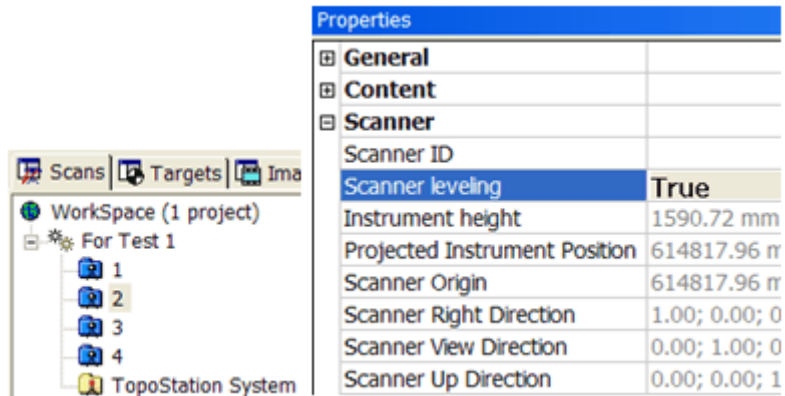
Warning: Do not modify the items which compose a **Trimble Survey Project**. If you decide to move the project to another location on your hard disk, move the directory that includes all the files, not only the **TSPX** format file.

Stations

In **Trimble Business Center**, each station has a station node representation in the **Project Explorer** panel. The representation matches the survey instrument type and is followed by the point ID and an "S" number in parenthesis that uniquely identifies the station.

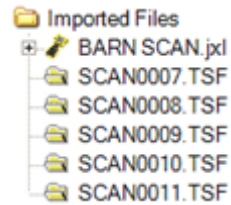


In **RealWorks**, the **Observation** information is not read. As a result, each station has a "Blue Folder" representation in the **Project Tree**. The blue color means that the station is "Leveled" but not setup over a **Known Point**.

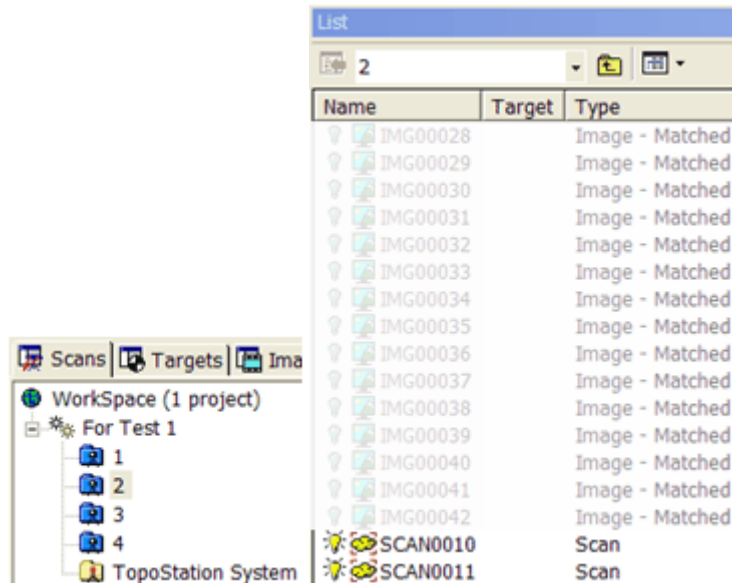


Scans

In **Trimble Business Center**, all scans (**Trimble Scanner Files (TSF)**) are put as a list after the imported data file node under the **Imported Files** node in **Project Explorer** panel.

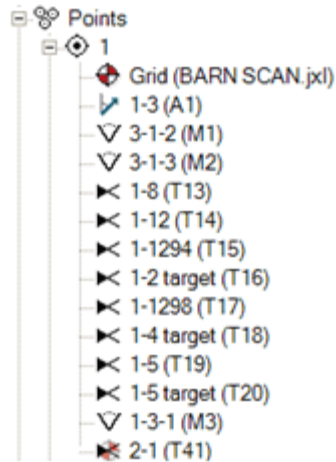


In **RealWorks**, each scan resulting from the measurement from a single station point of view is stored under that station in the **Scans Tree**.

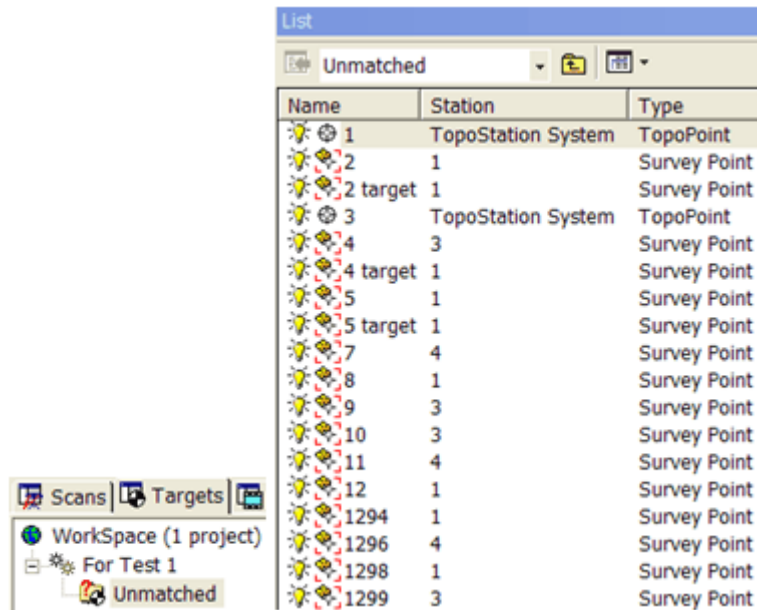


Points

In **Trimble Business Center**, each point in the project as a node representation and all are put under the **Points** node in the **Project Explorer** panel. Observations to and from the point are listed as observation nodes beneath it.



In **RealWorks**, each station contains all the measurements made from that station in the field. All points are set as unmatched and put under the **Unmatched** folder in the **Targets Tree**. A point resulting from the measurement from a single station is read as a **Survey Point** and is stored under that station. A point resulting from the measurement of a set of stations is read as a **TopoPoint** and stored under the **TopoStation System** folder.



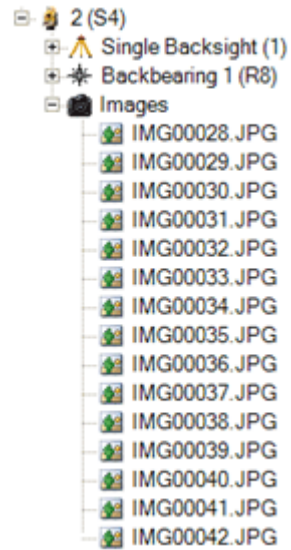
Name	Station	Type
1	TopoStation System	TopoPoint
2	1	Survey Point
2 target	1	Survey Point
3	TopoStation System	TopoPoint
4	3	Survey Point
4 target	1	Survey Point
5	1	Survey Point
5 target	1	Survey Point
7	4	Survey Point
8	1	Survey Point
9	3	Survey Point
10	3	Survey Point
11	4	Survey Point
12	1	Survey Point
1294	1	Survey Point
1296	4	Survey Point
1298	1	Survey Point
1299	3	Survey Point

Note: Key-in points, GNSS points, etc. are also read as **TopoPoints** and stored under the **TopoStation System** folder.

Tip: You can display (or hide) the **Topo Point** (or **Survey Point**)'s **3D Labels** by selecting **Rendering/Display 3D Labels** from the **3D View** menu. The **Display 3D Labels** feature once selected has a check mark on its side.

Images

In **Trimble Business Center**, images taken from a station point of view are stored under that station in the **Project Explorer** panel.



In **RealWorks**, images taken from a station point of view are stored as "Matched Images" under that station in the **Scans Tree** and in a list in the **Images Tree**.

List		
2		
Name	Target	Type
IMG00028		Image - Matched
IMG00029		Image - Matched
IMG00030		Image - Matched
IMG00031		Image - Matched
IMG00032		Image - Matched
IMG00033		Image - Matched
IMG00034		Image - Matched
IMG00035		Image - Matched
IMG00036		Image - Matched
IMG00037		Image - Matched
IMG00038		Image - Matched
IMG00039		Image - Matched
IMG00040		Image - Matched
IMG00041		Image - Matched
IMG00042		Image - Matched
SCAN0010		Scan
SCAN0011		Scan

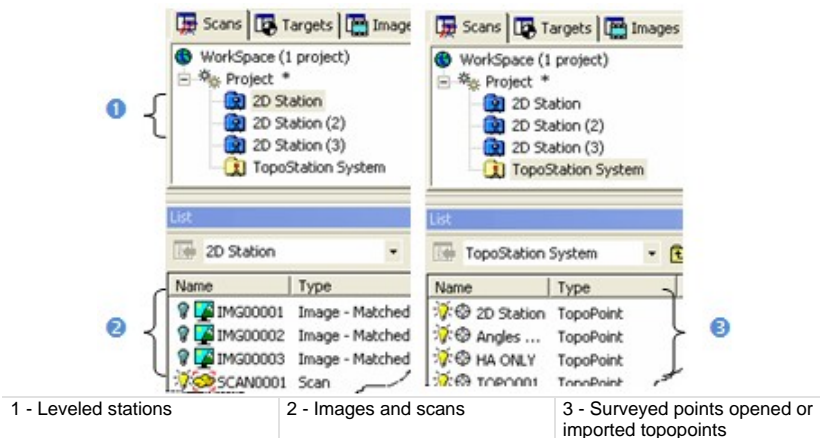
List			
For Test 1			
Name	Station	Type	Size
IMG00001	1	Image - Matched	2048 x 1536
IMG00002	1	Image - Matched	2048 x 1536
IMG00003	1	Image - Matched	2048 x 1536
IMG00004	1	Image - Matched	2048 x 1536
IMG00005	1	Image - Matched	2048 x 1536
IMG00006	1	Image - Matched	2048 x 1536
IMG00007	1	Image - Matched	2048 x 1536
IMG00008	1	Image - Matched	2048 x 1536
IMG00009	1	Image - Matched	2048 x 1536

JobXML, JOB and RAW Files

A **JobXML** file (with *.jxl extension) is a text file exported from Trimble Survey Controller™, Survey Manager™ or Survey Pro™ software in an XML based format. Some dependency files may be related to the JobXML file (such as Scan files with TSF extension (*.tsf stands for Trimble Scanning File) and JPEG images (*.jpg extension)). If Trimble Data Transfer© is installed on your computer, you can also open a JOB file (with *.job extension) which is the Survey Controller™ native binary file format. If Trimble Data Transfer© is not detected on your computer, the "Job Files (*.job)" file of type line in the **Open** dialog is not available. RAW file (with *.raw extension) is a SurveyPro™ native ASCII (o TXT) file format. A base converter is provided with **RealWorks** installer.

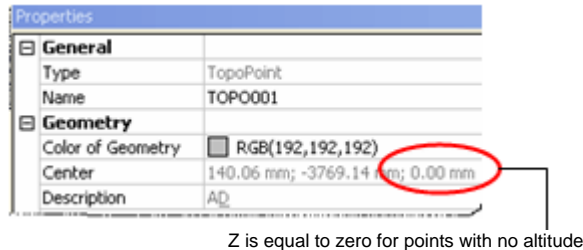
Objects in the JobXML (or Job or RAW) format file are opened (or imported) within the **Scans Tree** as follows:

- Points surveyed from one station setup are imported as standard survey points within each station,,
- GPS points and keyed-in points are imported as topo-points within the **TopoStation System** folder,
- Each station of the Instrument is opened (or imported) as leveled station,
- Registered scans using the scanning capabilities of the Instrument are put under their respective station,
- Images taken from the Instrument are registered under the respective station if the stationing is carried out when shooting the images. Images are put with no link to the station under the **Images Tree** if stationing has not been carried out when shooting the images.

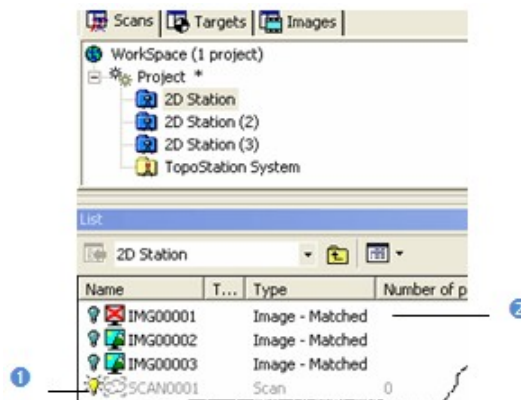


A warning message appears when:

- Points are surveyed with no altitude. These points are then opened (or imported) with altitude 0 in **RealWorks**,



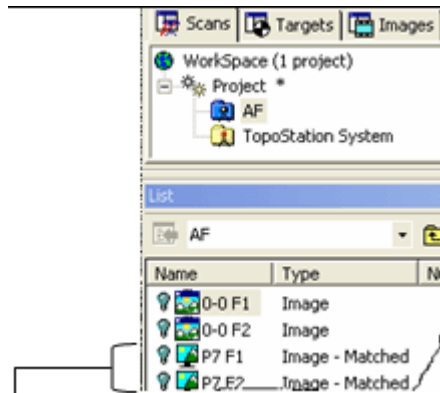
- Linked images/scans are missing from the reading folder. When opening (or importing) a JobXML (Job, Raw) format file, it refers to external files that contain scanning data and images respectively with the TSF (Trimble Scanning File) extension and the JPG extension. These files are supposed to be in the same folder than the JobXML (or Job) file. Missing TSF files are shown as null size scans and Missing images as broken link images.



1 - Missing TSF file opened (or imported) as null size scan

2 - Missing image file opened (or imported) as broken link image

- Some object fields are missing,
- Images have been taken around zenith. These images are opened (or imported) in the **Images Tree** as "Unmatched" images (through they are still linked to their shooting station).



Images taken around zenith are registered as "Matched"

Tip: You can display (or hide) the **Topo Point** (or **Survey Point**)'s **3D Labels** by selecting **Rendering/Display 3D Labels** from the **3D View** menu. The **Display 3D Labels** feature once selected has a check mark on its side.

ASCII Files

An **ASCII** format file may have **ASC**, **NEU** or **XYZ** as extension. There are in general two sections in such a file. The first section is called **Header** in which specific information about the nature of the file is stored. The second section is a list of **3D Points**. Each line contains a point represented by its X, Y, and Z coordinates plus, optionally, other attributes such as intensity or color.

Tip: An **ASCII** format file can be either opened as single project or imported into an existing one.

Note: A file to open (or to import) containing some corrupted lines will be ignored.

With Wizard

This opens a **Wizard** which allows the user to choose the parameters to fit the **ASCII** format file to open.

To Open With the Wizard:

1. In the **Open** dialog, keep the **Open Wizard for ASCII Files** option checked.
2. Click **Open**. The **Neutral Point Import** dialog opens.

This dialog is composed of six parts: five for adjusting the parameters to import the data and one for visualization.

Header	You can ignore the first lines of an ASCII file by selecting the number of lines to skip. These lines can be headers, comments or X,Y,Z coordinates that you do not want to keep.
Separator	Separator between attributes of a point can be a Comma, Tabulation or other.
Units	You have the choice between the Metric system and US/British system for the values of X, Y, Z coordinates.
Select Content	<p>This part enables to display (or to not display) the information about the Intensity, the Color and the Normal of loaded points.</p> <p>If the Intensity option has been chosen, the user can customize the intensity range. This means that intensity values larger than the value in the Max Intensity field were replaced by 255 and those that are between 0 and the Max Intensity value are mapped to the values from 0 to 255.</p>
Data Type	<p>According to the option (or combination of options) chosen in the Select Content panel, this part identifies fourteen different combinations of the attributes of a point. When the user chooses:</p> <p>§ No option, only the "Single Coordinates X,Y,Z" option is available.</p> <p>§ Intensity, the Max Intensity field and the "Coordinates and Intensity" option become enabled.</p> <p>§ Color, the "Coordinates and Color" option is enabled.</p> <p>§ Normal, the "Coordinates and Normal" option is enabled.</p> <p>§ Intensity and Color, the "Coordinates, Intensity and Color" and "Coordinates, Color and Intensity" options are available.</p> <p>§ Intensity and Normal, the "Coordinates, Normal and Intensity" and "Coordinates, Intensity and Normal" options are available.</p> <p>§ Color and Normal, the "Coordinates, Color and Normal" and "Coordinates, Normal" and "Color3" options are available.</p> <p>§ Intensity, Color and Normal, the "Coordinates, Intensity, Color and Normal", "Coordinates, Intensity, Normal and Color", "Coordinates, Color, Normal and Intensity", "Coordinates, Color, Intensity and Normal", "Coordinates, Normal, Intensity, and Color" and "Coordinates, Normal, Color and Intensity" are all available.</p> <p>If you select one of the options when you load an ASCII file without Intensity, Color or Normal RealWorks indicates that the Intensity, Color or Normal cannot be found and displays this error with three question marks between two brackets.</p>
Preview	Only the first thirty points are listed in the Preview panel.

3. Choose the parameters to fit the file to open.
4. Click **OK**. The **Neutral Point Import** dialog closes.

Notes:

- You can use the shortcut key **Ctrl + O** or click **Open** in the **Main** toolbar to pop-up the **Open** dialog.
- (*) We assume that there is no project opened. If there is one opened, the **Add to Project** option is enabled and default checked. You can then import such a file into the already opened project.

Without Wizard

When you load an **ASCII** format without the **Wizard**, **RealWorks** attempts to determine which separator is used and the different attributes of a point.

To Open Without the Wizard:

1. In the **Open** dialog, uncheck the **Open Wizard for ASCII Files** option.
2. Click **Open**. The **Open** dialog closes.

Note: An **ASCII** format file, when dragged and dropped into an open session of **RealWorks**, is loaded without the **Wizard**.

Trimble TX5 and Other FLS Files

RealWorks supports the **Trimble TX5** file format originating from the **Trimble TX5** 3D scanning system. A file in such format (with **iQscan** and **FLS** extensions) can be opened in **RealWorks**. The **Add to Project** option in the **Open** dialog is grayed out.

For each **Trimble TX5** format file, a dialog appears and informs you that you need to first create and save a project into the Trimble **RWP** format.

- A project and a station will be created and rooted under the **Project Tree**. The user has to give a name to the project while the station takes the **Trimble TX5** format file name.
- A **TZF Scan** is also created and put under the station.
- A project file and a folder are created. Both are named according to the name given by the user, with a **RWP** extension for the first and a **RWI** extension for the second. Under the **RWI** folder, a scan file with the **TZF** extension is also created.

Notes:

- The processing mode will automatically switch to **Registration**. The **Scans Tree** is selected by default.
- A **Trimble TX5** format file can be either opened as a single project or imported into an existing project.

Caution: Do not rename the extension (**FLS**) of the folder which contains the **FLS** format file to open. Otherwise, an error dialog opens and warns you that the **FLS** format file has been removed (or deleted) from its previous location.

Surveying Network ASCII Files

One of the key features of **RealWorks** is its ability to open (or import) surveyed data produced by other data collectors such as Total Stations, Field Stations, etc. Each such file will be opened (or imported) alone as a topographic station with points converted to topopoints (or as a station like other scanning stations in the selected project with points converted to 3D points).

A file with the **CRD** extension is a coordinate file with five data fields (Point number, Northing, Easting, Elevation and Description) in binary form. A file with the **CR5** extension is also a coordinate file but owned by TDS. A file with the **TXT** extension is an ASCII text file. Each line of the text file can contain any combination of Point number, Northing, Easting, Elevation and Description. All point information should be on one line with the values separated by a comma, space or other delineateers. All these parameters can be customized during the loading phase in **RealWorks**.

The **Surveying Network Import** dialog which appears after selecting a file to open is composed of six parts: five for adjusting the parameters to import the data and one for visualization. When you load a surveying network file, **RealWorks** attempts to determine which separator is used and the different attributes of a point. But you can customize these parameters:

Import:	You can import as topopoints or as 3D points.
Header:	You can ignore the first lines of a Surveying Network ASCII format file by selecting the number of lines to skip. These lines can be headers, comments, or X, Y, Z coordinates that you don't want to keep. The number of lines that you can skip is limited to 12. You can do the same for Column char and Comment line char.
Separator:	Separator between attributes of a point can be a semicolon, comma, tabulation or other.
Units:	You have the choice between several units: Millimeter, Meter, U.S. Survey Foot and International Foot. The U.S. Survey Foot, defined by the National Bureau of Standards NBS, corresponds to a value of 1200/3937m (or 0.3048006096m). The International Foot corresponds to a value of 0.3048m.
File Format:	You can choose between two types of contents: § Point Number, X, Y, Z, Description, § Point Number, Northing, Easting, Elevation, Description.

Note: The **Open Wizard for ASCII Files** option in the **Open** dialog becomes active and is default-checked if the file to be imported has a TXT extension; and remains inactive (grayed out) for files with CRD and CR5 extensions.

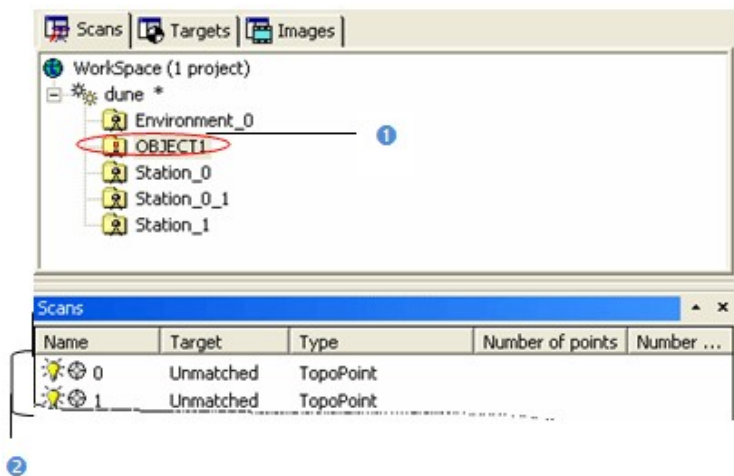
Tip: You can display (or hide) the **Topo Point** (or **3D Point**)'s **3D Labels** by selecting **Rendering/Display 3D Labels** from the **3D View** menu. The **Display 3D Labels** feature once selected has a check mark on its side.

Opening

If there is no project opened, you can only open a **Surveying Network ASCII** format file as a **Topographic Station**, the **Add to Project** and **In a Station (Fill with GeomPoints)** options respectively in the **Open** and **Surveying Network Import** dialogs are dimmed.

To Open a Surveying Network ASCII Format File:

1. In the **Surveying Network Import** dialog, customize the opening the parameters to fit the survey network file to open.
2. Click **OK**. The **Surveying Network Import** dialog closes.



1 - A topographic station

2 - A set of TopoPoints

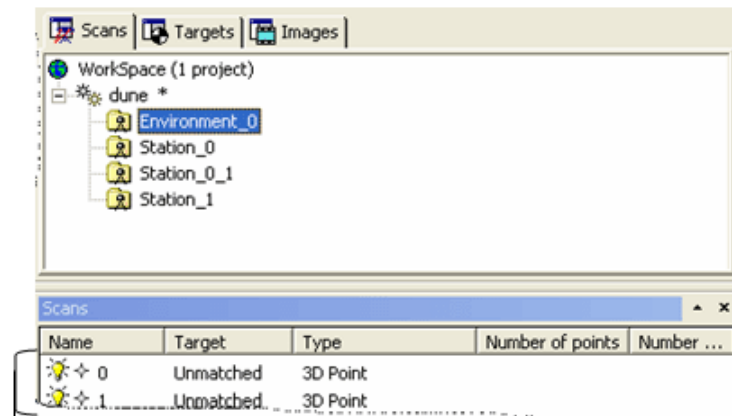
Note: The processing mode will automatically swap to **Registration**. The **Scans Tree** is by default selected.

Importing

If there is an opened project, you can choose between importing a **Surveying Network ASCII** format file in a station as unmatched **3D Points** inside or as a **Topographic Station** with unmatched **TopoPoints** inside.

To Import a Surveying Network ASCII Format File:

1. In the **Surveying Network Import** dialog, do one of the following:
 - Add the surveying network file as a station filled with topopoints.
 - a) Check the **As a TopoStation (Fill with TopoPoints)** option.
 - b) Customize the parameters for the **Surveying Network ASCII** format file to open.
 - c) Click **OK**. The **Surveying Network Import** dialog closes.
 - Add the surveying network file as 3D points in a station.
 - a) Check the **In a Station (Fill with GeomPoints)** option.
 - b) Click on the **In a Station (Fill with GeomPoints)** pull down arrow.
 - c) Choose a station from the drop down list.
 - d) Customize the parameters for the **Surveying Network ASCII** format file to open.
 - e) Click **OK**. The **Surveying Network Import** dialog closes.



A set of 3D Points

Notes:

- The processing mode will automatically swap to **Registration**. The **Scans Tree** is by default selected.

- When importing a **Surveying Network ASCII** format file, you need to have at least a station within your project. Otherwise the **In a Station (Fill with GeomPoints)** option is dimmed.

SIMA ASCII Files

RealWorks supports **SIMA ASCII** format files (Japanese survey file format). A file in such format (with sim extension) can be opened (or imported) in **RealWorks**. If no project is open, the **Add to Project** option in the **Open** dialog is grayed out and you are restricted to opening. If there is an opened project, the **Add to Project** option is enabled and default checked. Each file will be opened (or imported) as a topographic station and each point converted to an unmatched Topopoint.


Note: **RealWorks** will swap for the **Registration** processing mode after opening such a file.

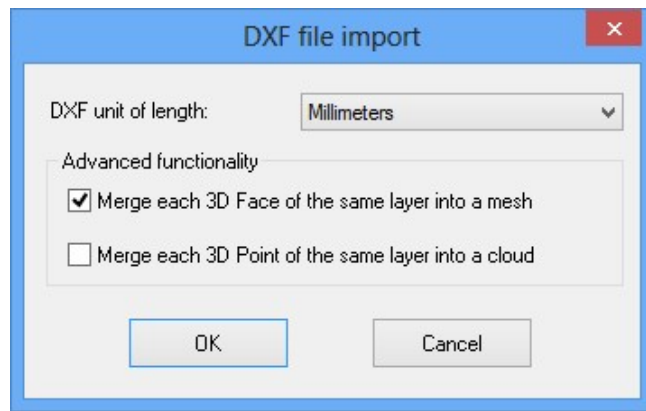
Tip: You can display (or hide) the **Topo Point's 3D Labels** by selecting **Rendering/Display 3D Labels** from the **3D View** menu. The **Display 3D Labels** feature once selected has a check mark on its side.

AutoCAD Files

RealWorks can open drawings in **DXF** (Drawing eXchange Format) or **DWG** (DraWinG) file format. The **DXF** file format is an **ASCII** file format which describes CAD data defined by AutoDesk. This file format facilitates the exchange of CAD data between two different programs. The **DWG** file format is the binary file format from AutoCAD and AutoCAD LT.

To Open a DXF (or DWG) Format File:

1. From the **File** menu, select **Open** . The **Open** dialog appears.
2. Select **AutoCAD Files (*.dxf; *.dwg)** from the **File of Type** field.
3. Navigate to the drive/folder where the DXF (or DWG) format file is located.
4. Click on the DXF (or DWG) format file's name to select it.
5. Click **OK**. The **DXF File Import** (or **DXF File Import**) dialog opens.

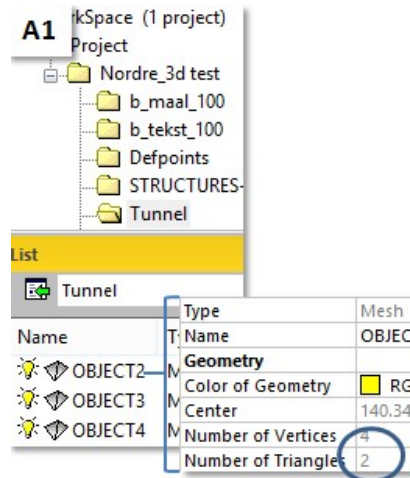


6. Click on the **DXF Unit of Length** (or **DXF Unit of Length**) pull down arrow.
7. Specify a unit of measurement to apply from the drop down list.
8. Choose between **Merge Each 3D Face of the Same Layer into a Mesh** and **Merge Each 3D Point of the Same Layer into a Cloud**.
9. Or check both options.
10. Click **OK**. The **DXF File Import** (or **DXF File Import**) dialog closes.

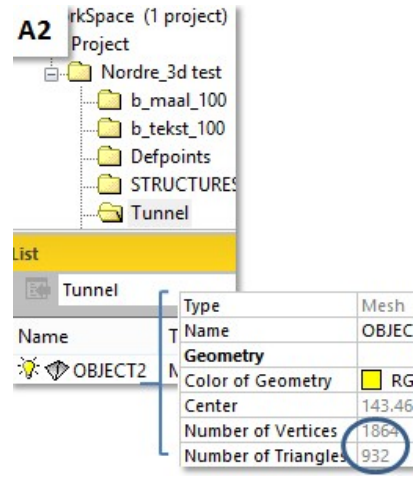
Caution: A warning message appears if the **DWG** (or **DXF**) format file to open (or import) contains entities with no equivalent in **RealWorks**. These entities will not be opened (or imported) in **RealWorks**.

Note: If there is no project open, you can only open a **DWG** (**DXF**) format file. The **Add to Project** option in the **Open** dialog is dimmed.

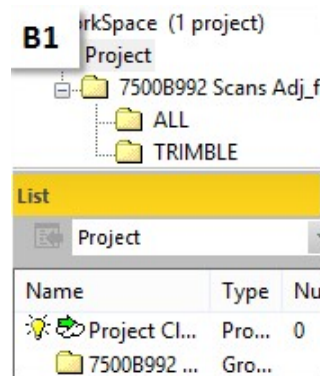
A set of model groups is created and put under a project rooted under the **Models Tree**. Each model group contains all 3D faces (or 3D points) of the same layer - each opened as a mesh of two triangles (see A1) (or as a 3D point object (see B1 and B2)).



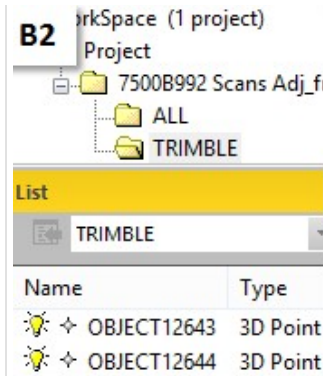
A1 - A mesh has been created for each 3D face



A2 - A mesh has been created for all 3D faces of the same layer

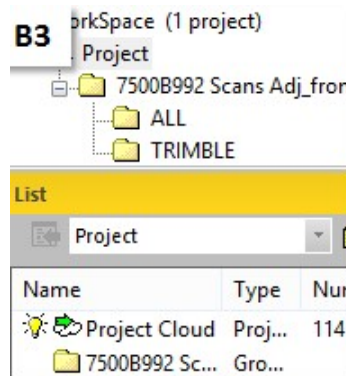


B1 - The Project Cloud is empty of points

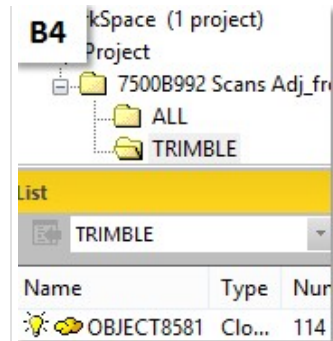


B2 - Each 3D point is opened as an object of 3D point

There are two options in the **DXF File Import** (or **DWG File Import**) dialog. The first when checked allows merging of all **3D Faces** of the same layer into a mesh (see A2) and the second all **3D Points** into a **Cloud** (see B3 and B4)).



B3 - The Project Cloud contains points



B4 - All 3D points of the same layer are opened and merged as a unique point cloud

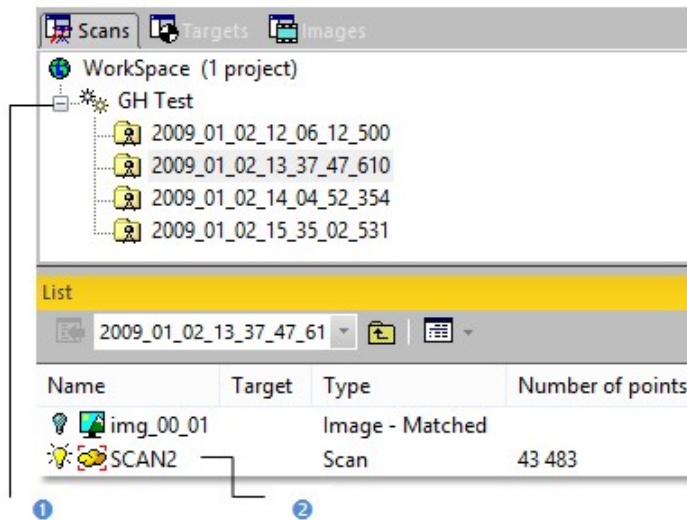
Note: For all 3D Points of the same layer, a station is created and rooted under the Scans Tree.

IXF Files

A file of IXF format (Optech's laser scanning systems - ILRIS - data format) with ixf extension can be opened (or imported) in RealWorks. If no project is open, the Add to Project option in the Open dialog is grayed out and you are restricted to opening. If there is an opened project, the Add to Project option is enabled and default checked. Each file will be opened (or imported) as station(s) put in the Scans Tree and as points put in the current Project Cloud in the Models Tree.

CMF Files

A **CMF** format file is a file provided by a Trimble CPW 8000 (or Trimble CX) instrument. For each **CMF** format file, a project will be created and rooted under the **Project Tree**. Inside the project, all points of the **CMF** format file will be put a scan under a station named by the **CMF** format file.



1 - The created station named by the file name 2 - All points are put in a scan

A **RealWorks** project file and a folder, respectively with the **RWP** and **RWI** extensions, are created. A set of **RWCX** format files (one per scan) and a set of **RWV** format files (one per image and when available) are created under the **RWI** folder.

Notes:

- A **CMF** format file can be either opened as a single project or imported into an existing project.
- The **CMF** library does not need to be installed apart as it is included in the **RealWorks** installer.

PTX Files

PTX is a file extension for laser scanning files. It is **ASCII** based. If there is no project, an information dialog appears for each **PTX** format file. This dialog informs the user that he needs to first create and save a project into the Trimble **RWP** format.

- A project and a station will be created and rooted under the **Project Tree**. The user has to give a name to the project while the station takes the **PTX** format file name.
- A **TZF Scan** is also created and put under the station.
- A project file and a folder are created. Both are named according to the name given by the user, with a **RWP** extension for the first and a **RWI** extension for the second. Under the **RWI** folder, a scan file with the **TZF** extension is also created.

Notes:

- The processing mode will automatically switch to **Registration**. The **Scans Tree** is selected by default.
- A **PTX** format file can be either opened as a single project or imported into an existing project.
- The project created within **RealWorks** is saved in the database.

PTX format files may contain several scans in the same station. **RealWorks** converts all the scans and creates **TZF Scans**, one per scan, in the same station in the **RealWorks** project.

PTS Files

PTS is a file extension for laser scanning files. It is **ASCII** based. A project and a station will be created and rooted under the **Project Tree**. The project is named **ProjectX** where **X** is its order. The station has as name the **PTS** format file name. Under the station, a scan named **ScanX** where **X** is its order is created.

Notes:

- The processing mode will automatically switch to **OfficeSurvey**. The **Models Tree** is selected by default.
- The created project is not saved in the database; the user has to save it manually, otherwise it will be lost.

Once the project is saved, a **RealWorks** project file and a folder are created. Both are named according to the name given by the user, with a **RWP** extension for the first and a **RWI** extension for the second. Under the **RWI** folder, a scan file with the **RWCX** extension is also created.

Merging Projects

You can merge several projects into a single project. After merging, a new project with the name **Merge Project** is created.

To Merge Projects:

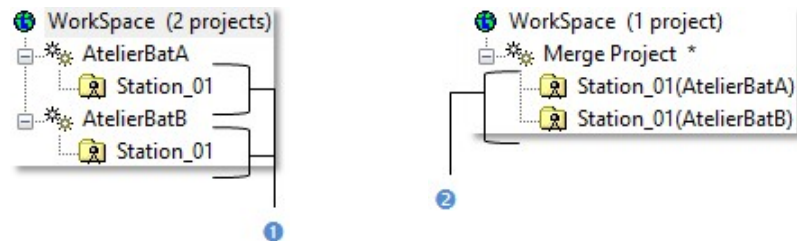
1. Select at least two projects from the **Project Tree**.
2. In the **File** menu, select **Merge Projects** from the drop-down menu.

Note: For each project that has been modified and not yet saved, a dialog appears and prompts you to save it.

Caution: There is no undo once projects are merged together other than to not save it.

Scans Tree

Stations from different projects are merged under the same project and are renamed according to the project they belong to.

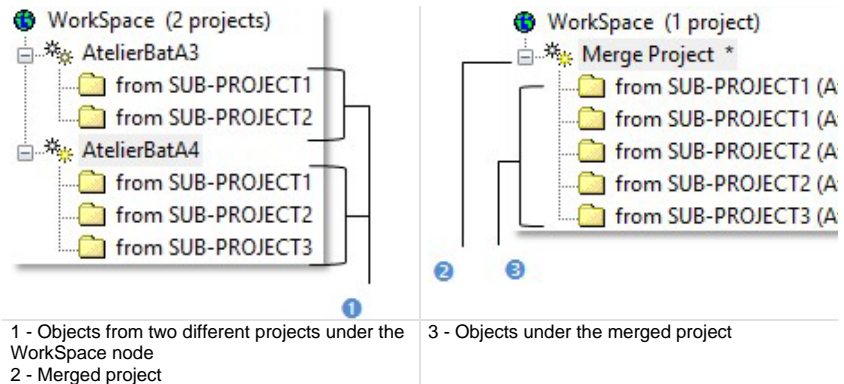


1 - Stations from two different projects under the WorkSpace window

2 - Stations under the merged project

Models Tree

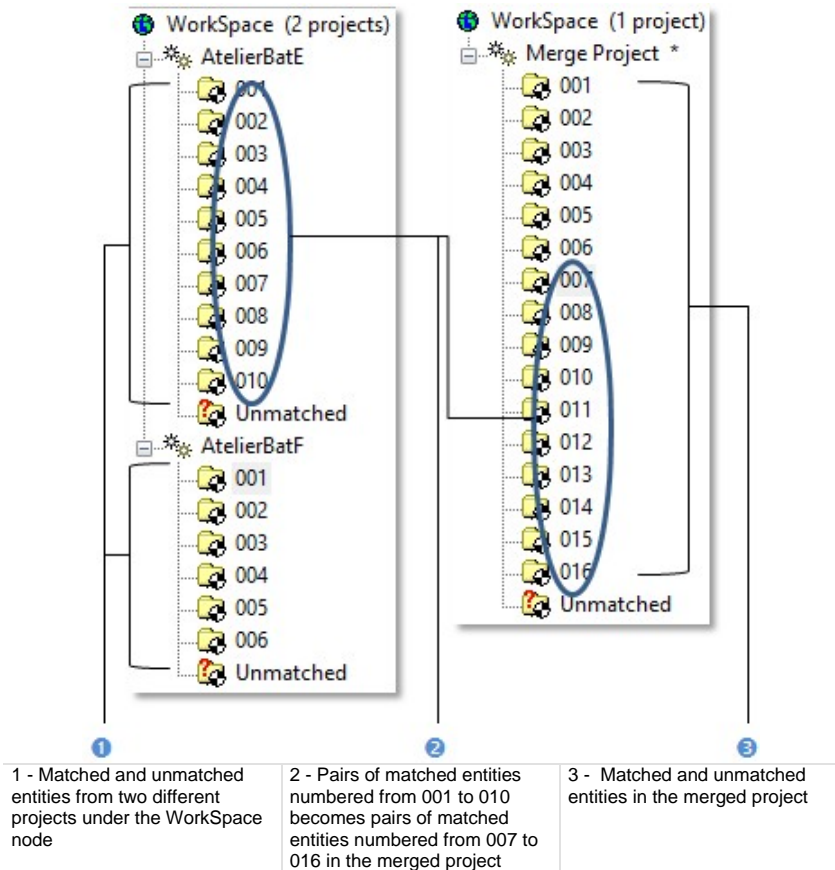
Objects (Group, Cloud, etc.) from different projects are merged under the same project and are renamed according to the project they belong to.



Note: The **Project Cloud** of the merged project is the sum of all **Project Clouds**.

Targets Tree

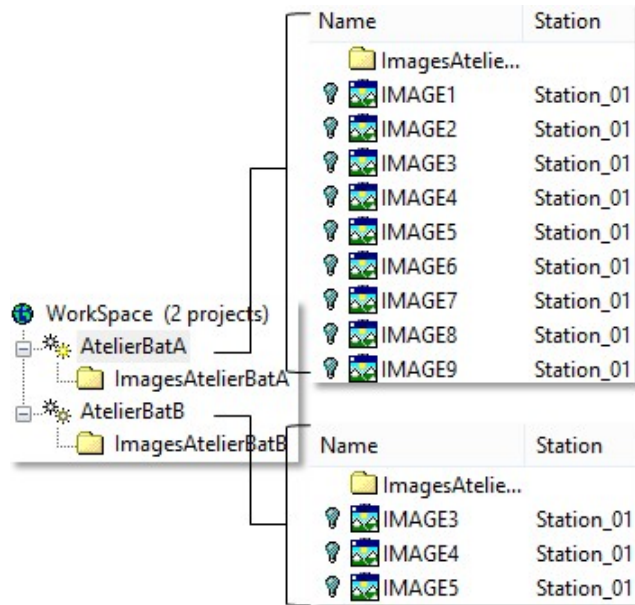
Unmatched entities are merged under the same group (called **Unmatched**) and matched entities (gathered in pairs) are put by order under the merged project. Pairs sharing the same name and the same order than those in the first loaded project (first from the **Project Tree**) have their number changed in order to continue the numbering*.



Caution: (*) Only pairs are renumbered, not the entities inside.

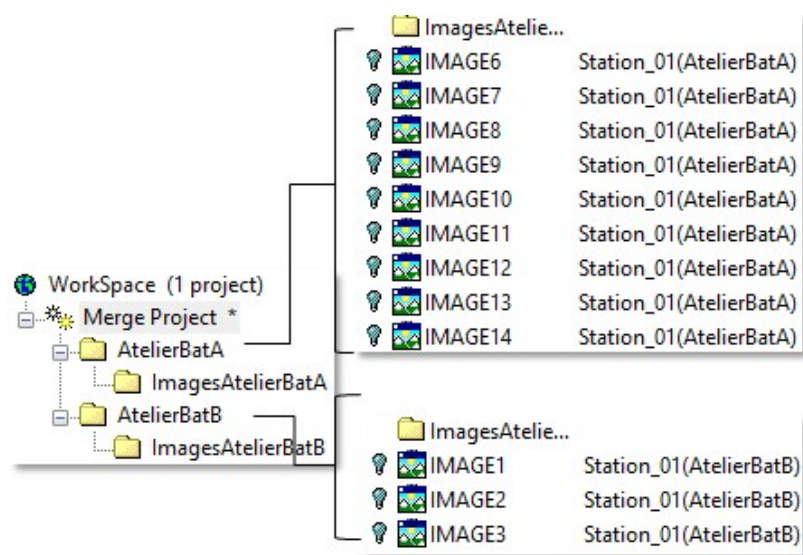
Images Tree

In the **Images Tree**, images of each tree are placed under a folder named after the old project. If all images are named **IMAGEX** where **X** is an order, all are renamed. The first image, from the first selected project and at the root of the **Image Tree**, is renamed to **IMAGE1**, the second **IMAGE2**, and so on.



Images Tree BEFORE merging

In the example below, **IMAGE3** from **AtelierBatB** is renamed **IMAGE1**. The project to which each image belongs is indicated between brackets. If all images are different; like e.g. A, B, C, etc. All keep their name.



Images Tree AFTER merging

Note: Feature code libraries having the same name are also merged (Feature codes of same name belonging to a library having the same name are duplicated).

Saving Projects


A project, which has not been saved, has an asterisk beside its name in the **Project Tree**. You can save the project into the existing project file by using the **Save** command or into a new project file by using the **Save As** command.

Caution: When a project has already been updated from an older version of **RealWorks** to the current version, saving it under the same name will make it inaccessible under older versions of **RealWorks**.

Caution: You cannot save (or save as) a project in **RealWorks Viewer**.

Save

To Save:

1. Select an unsaved project from the **Project Tree**.
2. From the **File** menu, select **Save** .

Tip: You can also use the shortcut key **Ctrl + S** or select **Save** from the **Main** menu.

Save As

To Save As:

1. Select either a saved project or an unsaved project from the **Project Tree**.
2. From the **File** menu, select **Save As**. The **Save** dialog opens.
3. Navigate to the drive/folder where you want to store the project.
4. Enter a name in the **File Name** field. The extension is added automatically.
5. Click **Save**. The **Save** dialog closes.

Publish

The input of the feature is a single project with at least one **TZF** format file within. Otherwise, the feature is grayed out. The output of the feature, both a published project and an embedded version of **Trimble Scan Explorer**, can then be distributed via a media like a USB flash drive or DVD-Rom. To be able to view a published project with the embedded version of **Trimble Scan Explorer**, the user needs to have **Internet Explorer 8** or later and **.Net Framework 4** and **Visual C++ Redistributable 2010 X86**.

Publishing Projects

To Publish a Project:

1. Select a project from the **Project Tree**.
2. From the **File** menu, select **Publish**. The **Publish 'Project_Name'** dialog opens.
3. Setup the Page Configuration,
4. Select the Media to add,
5. Define a Link pointing to e.g. a document.
6. Apply a Data Size Reduction;
7. If required, Enable Extract data in the published project,
8. Click **Publish** to start publishing.

Tip: You can abort the publication in progress by clicking **Cancel**.

Page Configuration

The default name in the **Title** field is the name of the published project. It's up to you to give a name other than this default one. The default (or defined) **Title** is the one which appears at the top left corner of the published page. A **Logo** is an image file that may have the following formats: bmp, jpg, png and gif. It also appears at the top left corner of the published page.

The **Output** folder is the folder where all the published files are located. The default path to the **Output** folder is C:\Users\User_Name\Documents. The default **Background Color** is yellow. The default **Font Color** is dark grey.

To Setup the Page Configuration:

1. Click on the **Page Configuration** tab.
2. Do one of the following:
 - Define a Title,
 - Attach a Logo file,
 - Define an Output Folder,
 - Set a Background Color,
 - Set a Font Color.


Title

To Edit a Title:

- Input a new name in the **Title** field.

Logo


To Add a Logo:

1. Click **Open** . The **Open** dialog opens.
2. Navigate to the drive/folder where the **Logo** file is located.
3. Click on the file to select it. Its name appears in the **File Name** field.
4. Click **Open**. The **Open** dialog closes. The path to the **Logo** file is displayed in the **Logo** line.

Tip: You can manually enter the path of a **Logo** file.

Output Folder

To Define an Output Folder:

1. Click **Open** . The **Browse For Folder** dialog opens.
2. Locate a drive/folder to store the published files.
3. If required, create a new folder by clicking **Make New Folder**.
4. Click **Ok**. The **Browse For Folder** dialog closes. The path to the defined folder is displayed in the **Output** line.

Tip: You can manually enter the path of the **Output** folder.

Note: The defined **Output** folder will not be taken into account if you do not publish your project (by clicking on the **Publish** button).

Background Color

There are nine **Standard Colors**: **White**, **Gray**, **Black**, **Red**, **Green**, **Blue**, **Yellow**, **Orange**, and **Purple**.

To Set a Background Color:

1. Click on the **Background** pull-down arrow.
2. Choose color from the **Standard Colors** panel.

Or

3. Click the **Advanced** button. The **Advanced Colors** panel expands.
4. Define a color.

Font Color

There are nine **Standard Colors**: White, Gray, Black, Red, Green, Blue, Yellow, Orange, and Purple.

To Set a Font Color:

1. Click on the **Font Color** pull-down arrow.
2. Choose a color from the **Standard Colors** panel.

Or

3. Click **Advanced**. The **Advanced Colors** panel expands.
4. Define a color.

Media


A **Media** item can be either an image or a video. An image file may have the following formats: jpg, jpeg, png and gif. A video file can be of the following formats: **WebM**, **Avi**, **MP4** and **flv**.

WebM is a new open standard for compressing Video content. It is based on both the **VP8** and **Vorbis**, respectively for the Video and for the Audio. **Avi** (Audio Video Interleave) is a multimedia container format from Microsoft. **MP4**, an abbreviated term for **MPEG-4 Part 14** or **MPEG-4 AVC**, is also a multimedia container format. **FLV** known as **Flash Video**, is a video file format used to deliver video over the **Internet** using **Adobe Flash Player**.

Note: There is no restriction to the number of **Media** item you can add.

Including a Media File

To Include a Media File:

1. Click **Open** . The **Open** dialog opens.
2. Navigate to the drive/folder where the **Media** file is located.
3. Click on the file to select it. Its name appears in the **File Name** field (in the **Open** dialog).
4. Click **Open**. The **Open** dialog closes.
 - The selected **Media**'s name is displayed in the **Title** field.
 - Its Path appears in the **File Path** field.


Note: First click on the **Media** tab.

Tips:

- You can manually enter the path of a **Media** to include when publishing.
- You can also manually edit a **Media**'s name in the **Title** field.

Including Another Media Files

To Include Another Media File:

1. Click **Add** . Another **Media** line is added to the previous one.
2. Click **Open**. The **Open** dialog opens.
3. Navigate to the drive/folder where the **Media** file is located.
4. Click on the file to select it. Its name appears in the **File Name** field.
5. Click **Open**. The **Open** dialog closes.


Notes:

- First click on the **Media** tab, if not done.
- You cannot add another **Media** file unless the first **Media** has been added.

Links

What is a **Link**? A **Link** can be either a **Path** or **URL** pointing to a document, a Web page, etc.

To Add a Link:

1. Click **Open** . The **Open** dialog opens.
2. Navigate to the drive/folder where the **Link** file is located.
3. Click on the file to select it. Its name appears in the **File Name** field.
4. Click **Open**. The **Open** dialog closes.

Note: First click on the **Links** tab.

Tips:

- You can manually enter the path of a **Link** to include when publishing.
- You can also manually edit a **Link's** name in the **Title** field.

Caution: If you add an application (.exe); it could be not worked if all dependencies (of the application) are not present.

Note: When a **Link** points to a document, the document is copied into the final published project.

Data Size Reduction

There are three filters (called **Large**, **Medium** and **Small**) for reducing the data size. For each **TZF** format file, **Spatial Sampling** (with a step of 6 mm) is applied to the set of points and a size reduction to the file itself. The **File Size** in the **Publish** dialog is the size of the project once the reduction is applied. The **Estimated Time** is the time required to process the whole project.


To Choose a Data Size Reduction Method:

- Drag to the slider up or down to choose a reduction level.

Enable Extract

The **Enable Extract** option is by default unchecked. It activates removal of the **Extract Points** feature from the **Trimble Scan Explorer Web Viewer**. This means that the user cannot extract any points from the published data.

Caution: You will be warned that your data will be published with extraction capabilities if the **Enable Extract** option is checked.

Note: If the **Enable Extract** option is kept unchecked, a **TZF** format file, once published in the **Data/RWI** folder, is locked. Its icon is grayed-out  if you try to load it in **RealWorks**.

Published Projects

Once you start publishing your project (by clicking **Publish**), two progress bars appear, one for all the **TZF** format files and one for a **TZF** format file. Once the publication is completed, you may see the following texts: "Publish Succeeded" and "Nb. of files processed in minutes and seconds".

A main folder, named according to the name given in the **Title** field, is created under the default (or defined) **Output Folder**. The day and time information are added to the main folder (respectively in the **Day-Month-Year** and **Hour-Minute-Second** format). Two sub-folders (**Bin** and **Data**) and an **Html** format file (named **Index**) are created under the main folder.

Bin Folder

The **Bin Folder** is the folder used to embed **Trimble Scan Explorer** application files.

Data Folder

The **Data** folder is the folder where a published project is stored. By default, a published project shares the same name as the original project (project to publish) or the name defined by the user in **Title** field. A published project in the **Data** folder is a replica of the project that has been published (from **RealWorks**). This means that there is a **RWP** file and a **RWI** folder within (the **Data** folder). Each **TZF** format file from the project to publish is replicated in the published project in the **RWI** folder in the **Data** folder but with a size other than the original ones (due to the size reduction).

Caution: Only **TZF Scans** and images (**RWV**) are published.

Note: **Point Clouds** of published data are not accessible. Only **TZF Scans** and images are accessible. An error message appears if you try to open a published project. The message warns you that scan point file(s) is (or are) missing, points cannot be loaded and the project cannot be open.

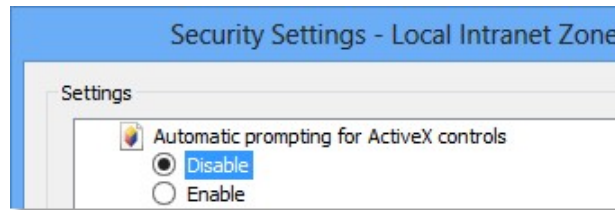
Viewing

To view a published project within the embedded version of **Trimble Scan Explorer**, just double-click on the **Index** file. Below is a sample of what the user should have.



- | | |
|---|--------------|
| 1 - Logo | 4 - Link(s) |
| 2 - Title | 5 - Video(s) |
| 3 - Embedded version of Trimble Scan Explorer | 6 - Image(s) |

Note: **Scan Explorer** is displayed as a webpage which requires running a script or **ActiveX** control. Please, allow the control to run by clicking the **Allow Blocked Content** button. If you do have permission, you could turn off the prompt for **Internet Explorer** by following **Tools, Internet Options, Security** (tab), **Custom Level** (button) and disable the **Automatic prompting for ActiveX controls** option.



Note: Windows may consider Trimble as an untrusted publisher and prevent Scan Explorer from running on your computer by opening the Application Run - Security Warning dialog. Please, do not take the warning into account and run Scan Explorer by clicking the Run button in the dialog.


Tip: A Media once included is displayed as a thumbnail. Click on an embedded Media to enlarge it.

Note: You need to have .Net Framework 4 and Visual C++ Redistributable 2010 X86 installed on your computer.

Opening Scan Explorer

Scan Explorer is a plug-in hosted in **RealWorks**. It is a navigator dedicated to handling and navigating large databases from which the user can extract and send data to **RealWorks** or to a specific file format.

To Open Scan Explorer:

1. Select either a project* or a station* (or set of stations*).
2. From the **File** menu, select **Open Scan Explorer** .

Note: (*) If the **TZF** format file(s) has (have) not yet been processed, the **Processing TZF Scans** dialog opens and prompts you to proceed to do so.

Tip: You can also select **Open Scan Explorer** from the **Main** toolbar.

Warning: A message appears if one (or more) TZF format file(s) is (or are) missing in the project (loaded through **RealWorks** or if the project is not compatible with **Scan Explorer**.

TZF Scans

Data coming from a Trimble FX instrument is stored in a C3D format file. Such data (in C3D format) once processed (in the Trimble FX Controller software) is saved as a Trimble Scan File (with T3S file extension). Files with such extension should now be converted to the TZF format, otherwise you cannot open them.

The table below lists the type of tool the user can use according to the license (of RealWorks) he has.

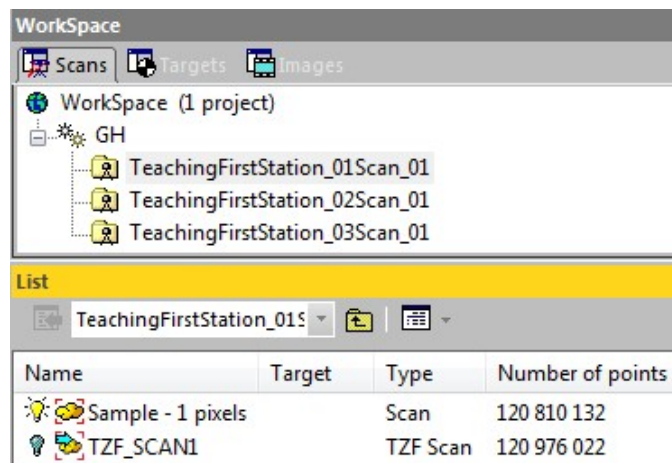
Tools (in)	Viewer	Base, Advanced, Advanced - Modeler, Advanced - Plant
Create Sampled Scans		✓
Modify Path for Input TZF Scan Files	✓	✓
Create Thumbnails	✓	✓
Generate Point Color-Coding by Height	✓	✓

Creating Sampled Scans

The **Create Sampled Scans** feature allows the user to create a series of **Sampled Scans** in batch processing mode without having to interact. But if required, the user can also create a series of **Sampled Scans** one by one.

To Create Sampled Scans:

1. Select either a project or a station (or set of stations).
2. From the **File** menu, select **TZF Scans / Create Sampled Scans**. The **Sampled Scan Creation** dialog appears.
3. Sample the scan data (or apply a distance filter).
4. Click **Ok**. The **Sampled Scan Creation** dialog closes.
 - Each new **Sampled Scan** is named as follows: **Sample - A px** where **A** is the **Step** value set in the **Sampling By Step** or **Sample - A`mm`** where **A`** is the **Distance** value set in the **Spatial Sampling**. (** If **Millimeter** has been set in **Preferences**).
 - You can add as many **Sampled Scans** as needed under a given station. If a project has been selected, **Sampled Scans** (one per station) are created in batch processing mode. If a station has been selected, only a **Sampled Scan** is created and under the selected station.



Warning: All **Sampled Scans** cannot overall four billions points.

Tip: When you create from several **TZF Scans** within a station, all **Sampled Scans** (in that station) do not have the same color. Each has its own color.

Notes:

- If the selected project has not been yet saved in the database, you are then prompted to do so. We advise you to save the project under the same folder as the **TZF** format files.
- Inside the RWI folder, a scan file (with RWCX extension) is created per sampled scan named as follows ProjectName_StationX_ScanY.

Sampling



There are two sampling methods (**Sampling by Step** and **Spatial Sampling**) that you can use in order to reduce the number of points in the scan data.

The sampling will be applied to the entirety of the scan data. The number of estimated points will be updated according to the defined parameter except when using the **Spatial Sampling**. It is in that case "**Undefined**". Note that the use of the **Filter by Range** has no effect on the number of estimated points.

Sampling by Step

In the **Sampling by Step** method, one point will be taken into account at each defined **Step** vertically and horizontally in the 2D image data.



To Sample by Step:

1. Click on the **Sampling** pull-down arrow.
2. Choose **Sampling by Step** from the drop-down list.
3. Enter a value in the **Step (In Pixels)** field.
4. Or use the **Up**  (or **Down** ) button to choose a value.

Sampling Spatially

The **Spatial Sampling** method allows you to obtain a point cloud with a homogeneous spatial density that you have to define.

To Sample Spatially:

1. Click on the **Sampling** pull-down arrow.
2. Choose **Spatial Sampling** from the drop-down list.
3. Enter a value in the **Resolution** field.
4. Or use the **Up**  (or **Down** ) button to choose a value.



Filtering (Optional)

There are two filters (**Filter by Range** and **Filter by Zone**) that you can use in order to reduce the number of points in the scan data. Note that the use of the **Filter by Range** has no effect on the number of estimated points.

Filtering by Range

The **By Range** allows you to define a distance (from the center of the FX instrument) beyond which no point will be taken into account. This filter is only applied to the scan data.

To Filter by Range:

1. Check the **Filter by Range** option. The **Max Distance** field becomes editable.
2. Enter a value in the **Max Distance** field.
3. Or use the **Up**  (or **Down** ) button to choose a value.

Filtering by Zone

The **By Zone** option allows filtering by defining a bounding box. The **Min Point** and **Max Point** are the two extremities of a bounding box diagonal.

To Filter by Zone:

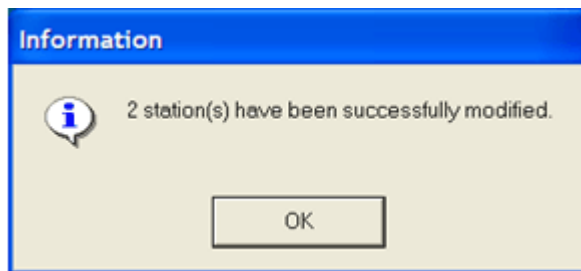
1. Check the **By Zone** option. The **Min Point** and **Max Point** fields become editable.
2. Enter a 3D coordinates in the **Min Point** field.
3. Enter a 3D coordinates in the **Max Point** field.

Modifying the Path for Input TZF Scan Files

Displacing a lone (or a set of) **TZF** format file(s) from its folder will break the correspondence with the station(s) created within **RealWorks**. You can no longer add new scans to the project*. The **Modify Path For TZF Input File(s)** command allows you to restore that correspondence by changing the path to the **TZF** format files.

To Modify the Path for the Input TZF Scan Files:

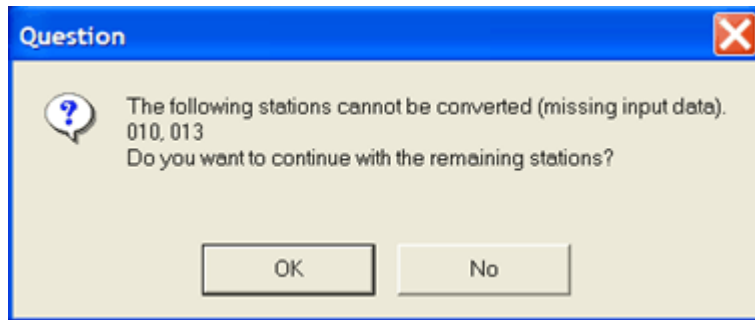
1. First create new stations (in batch processing mode or in interactive mode)**.
2. Then select either the **Project**, or the "New Group" or a **station**.
3. From the **File** menu, select **TZF Scans / Modify Path For Input TZF Scan Files**. The **Select New File Folder** dialog opens. The default folder is the one storing the opened project.
4. Navigate to the new location of **TZF** files stored in the **In** field.
5. Select the folder and click **OK**. The **Select New File Folder** dialog closes and the Information dialog below appears.



6. Click **OK** to close the information.

Notes:

- You may see the path to a **TZF** format file by displaying its related station's properties (see **Input Data Path Name** line).
- (*) Trying to add new scans without changing the path to the **TZF** format files will open a dialog which prompts you to continue with the remaining stations or not.



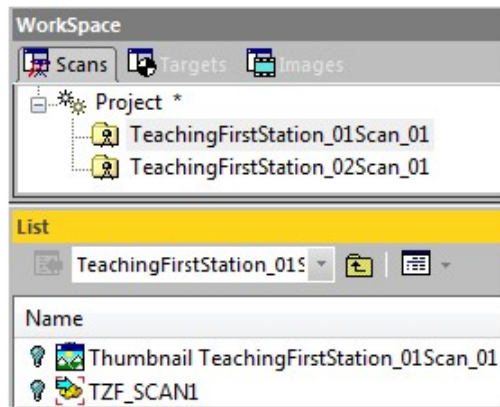
Note: When opening a project with **TZF** station(s) created out of **RealWorks**, the correspondence(s) (from the **TZF** station(s) to the **TZF** format file(s)) will be automatically updated only if the **TZF** format file(s) is (or are) in the **RWI** folder (of the **Project**).

Creating Thumbnails

The **Create Thumbnails** feature allows you to create **Thumbnails** in batch processing mode or not. In the batch processing mode, the user interaction is not required and a set of **Thumbnails** is created (one per **TZF** file). Out of the batch processing mode, the user interaction is required and one **Thumbnail** is created for a given **TZF** file. A **Thumbnail** is a preview of a **TZF** file within **RealWorks**.

To Create Thumbnails:

1. Select a project or a station (or set of stations).
2. From the **File** menu, select **TZF Scans / Create Thumbnails**.
 - For each **TZF** format file, a **Thumbnail** is created.
 - The created **Thumbnail** is put under its related station in the **Scans Tree** and as a list in the **Images Tree**.
 - Each thumbnail has the following name: **Thumbnail_ its related TZF file Name**.



Note: If the selected project has not been yet saved in the database, you are then prompted to do so. We advise you to save the project under the same folder as the **TZF** format files.

By default, a **Thumbnail** is not shown in the **3D View**. You need to toggle the **On/Off** icon to **On** to display it as a thumbnail in the **3D View**. You can then drag and drop the thumbnail to any location within the **3D View**. To hide the thumbnail; you have choice between toggling **Off** the **On/Off** icon and clicking on the **Close** button at the top right corner.

Generating Point Color-Coding by Height

This feature will apply a rendering to point clouds according to the height of each point along the **Z** (or **EI**) axis. This rendering will be applied to both loaded and unloaded points.

To Generate Point Color-Coding by Height:

1. Select a project (or a set of projects).
2. From the **File** menu, select **TZF Scans / Generate Point Color-Coding by Height**. A dialog opens warning you that the operation is definitive and may take a long time.
3. Click **Yes** to continue.
4. Click **No** to abort.

Note: The **Rendering** option will automatically swap to **True Color**.

Importing Images

Image files in JPEG, BMP (only of 24-bit depth) and TIF formats can be imported into an existing project. An imported image is rooted under the **Images Tree**. If you attempt to import an image of a format other than those mentioned above, an error message appears

To Import an Image:

1. Select a project from the **Project Tree**.
2. In the **File** menu, select **Import Image**. The **Import Image** dialog opens.
3. Select the right image type from the **File of Type** field.
4. Navigate to the drive/folder where the image file is located.
5. Click on the file to select it.
6. Click **Import**.

Selection Mechanism

Selection is a very important concept. All commands and tools in **RealWorks** will be applied only to the selected projects/objects. It is thus clear that before evoking commands and tools, you should first of all select the objects that you want to operate on. It is also important to note that according to the nature of the selected objects; only the applicable commands and tools will be available (whether from the menus, the toolbars or the pop-up menu). In other words, the applicability of tools and commands is context-sensitive.

WorkSpace Window

A selection in the **WorkSpace** window is done by picking an object by left clicking it. It is important to note that you can only make a single selection here and this selection will not be put in the **Selection List** window.

Models Tree

Selecting a project node will display its contents (**Project Cloud**, objects, or group of objects) in the **List** window. Selecting a group node will display its contents (objects or groups of objects) in the **List** window.

Scans Tree

Selecting a project node will display its contents (stations, group of stations, scans, or images) in the **List** window. Selecting a station node will display its contents (scans, targets or images) in the **List** window.

Images Tree

Selecting a project node will display its contents (images or group of images) in the **List** window. Selecting an image group node will display its contents (images or group(s) of images) in the **List** window.

Targets Tree

Selecting an unmatched group node will display all unmatched targets in the **List** window. Selecting a matched group node will display all matched targets in the **List** window.

3D View Window

A selection in the **3D View** consists of picking an object by left-clicking it. The selected object will be highlighted by its bounding box in this window and also in the **List** window and will be put in the **Selection List** window (if opened). If you want to make a multiple-selection, you can only do so by clicking on the objects in the **3D View** one by one (combine the **Ctrl** key with left-clicking). To select all displayed objects in this window, you should use the following combination **Ctrl + A**.

Note: You can change the bounding box's color in the **Preferences** dialog.

List Window

A selection in the **List** window consists of picking an object by left-clicking it. The selected object will be highlighted in this window and also in the **3D View** (if it is displayed) and will be put in the **Selection List** window (if opened). To make a multiple-selection, you can either combine the **Shift** key with left-clicking or **Ctrl** with left-clicking. To select all objects in this window, you should use the following combination **Ctrl + A**.


Clearing Selections

All selections done in the **List** window and in the **3D View** will reside in the **Selection List** window until you decide to clear them. To do so, use the **Clear Selection** button in the **Selection List** window.

Undoing Operations

You can undo the last operation when the **Undo Operation** command is available. You can execute multiple-level undo, but its behavior varies depending on whether you use a command or a tool. When you use a command, the undo will delete its effects. When you are inside a tool, multiple undo will be applied to all intermediate steps of the tool, including their parameter settings and database operations. Once you exit the tool, multiple undo will take effect only on the database operations carried out by all operations of the tool.

To Undo an Operation:

1. In the **Edit** menu, select **Undo Operation** .
2. Continue clicking **Undo Operation** to remove as many previous operations as necessary.


Notes:

- You can also use the **Ctrl + Z** shortcut keys or click the corresponding icon in the **Main** toolbar.
- The **Undo** stack is limited to the value defined in **Preferences**.

Redoing Operations

If you decide to restore the last operation (or action) you carried out in **RealWorks**, you can easily do so by using the **Redo Operation** command. When the **Redo Operation** command is unavailable (dimmed), it means that you cannot redo the last operation (or action).

To Redo an Operation:

1. In the **Edit** menu, select **Redo Operation** .
2. Continue clicking **Redo Operation** to redo as many previous actions as necessary.

Note: You can also use the **Ctrl + Y** shortcut keys or click the corresponding icon in the **Main** toolbar.

Closing Projects

You can either close a selected project (or all projects). When the project(s) is (or are) not saved, a warning will be issued. You will be prompted to save (or not to save it (or them)). Note that there is one warning per open project and **Close All** does not require selection.

Close

To Close the Selected Project:

1. Select a project from the **Project Tree**.
2. From the **File** menu, select **Close**.

Close All

To Close All Projects:

- From the **File** menu, select **Close All**.

Capturing the Screen

This command allows you to dump the screen of the **3D View** into a BMP image file.

To Capture the Screen:

1. In the **File** menu, select **Capture Screen**. The **Save File** dialog opens.
2. Type a name for the image file. **RealWorks** assigns automatically the .BMP extension to this file.
3. Specify the location you want to store this file by navigating through the drive/folder.
4. Click **Save**. The **Save File** dialog closes.

Setting Preferences

Preferences allow you to customize the behavior and aspects of **RealWorks**. These preferences are grouped under four tabs. Preferences are persistent in **RealWorks**, i.e. the setting changes will be memorized and used for the following **RealWorks** sessions.

To Set Preferences:

1. In the **Edit** menu, select **Preferences**. The **Preferences** dialog opens.
2. Or right-click in the **3D View** (except on displayed objects and not within a tool in use) and select **Preferences** from the pop-up menu.

General

Preferences in the **General** tab are described in the table below.

This Option	Enables
Stack Size For Undo/Redo	To specify the number of levels for undo/redo operations. You can choose between 1 and 50.
Location	RealWorks creates a temporary backup file for each opened project. This option allows you to specify a location to which this backup file will be stored. By default, the backup folder is Windows/Temp.
Capacity	This field indicates the capacity of the selected folder.
Coordinate System	To choice between Cartesian Coordinate System and Geographic Coordinate System.
Orientation Measure System	To choose between two systems for orientation measurements.
Language Settings*	To select a language. The setting will only take effect next time the application is launched.

To Set the General Preferences:

1. Click on the **General** tab. The **General** dialog appears.
2. Set a parameter for a preference item.
3. Click **Apply**. The setting will be applied and the **Preferences** dialog remains open.
4. Click **OK**. The setting will be applied and the **Preferences** dialog closes.

Note: (*) A warning dialog opens and warns that you need to restart **RealWorks** when changing the language setting. Otherwise the new language setting will not be taken effect.

Viewer

Preferences in the **Viewer** tab are described in the table below and allow you to control the behavior and aspect of the **3D View**.

This Option	Enables
Display Coordinate Frame	To specify the choice between displaying and hiding the coordinate frame in the 3D View.
Auto-Spin	A scene to turn around itself with a speed defined by the last mouse movement.
Display Scale In Orthographic Mode	To specify the choice between displaying and hiding the scale in the 3D View.
Head Always Up (Z Axis)	When you move a 3D scene (rotate, zoom or pan) in the 3D View, you may lose its orientation in relation to the coordinate frame. This option allows you to keep the 3D scene with its Z direction always up (relative to the active coordinate frame).
Reverse Mouse for Zoom	To invert the motion for zoom in (or out) in all viewers (3D or 2D).
Background Color	To change the background color of the 3D View window. The default color is dark gray when you first start RealWorks. You can change it to the color you prefer. In addition to this, you can apply a gradient effect to this background color.
Highlight Color	To change the color of the bounding box of selected objects. The default color is yellow.
Info Box Opacity	To define the transparency of the information box using a slider.
Has Video Memory	To use the full capability of your computer graphics card to accelerate the display when handling a large amount of points.

To Set a Viewer Preference:

1. Click on the **Viewer** tab (if required). The **Viewer** dialog appears.
2. Set a preference.
3. Click **Apply**. The setting(s) will be applied and the **Preferences** dialog remains open.
4. Click **OK**. The setting(s) will be applied and the **Preferences** dialog closes.

Units

Preferences in the **Units** tab are described in the table below and allow you to set the unit system to use in your project.

This Option	Enables
Decimal Places	To specify the number of digits after the decimal point.
Display Value With Unit Tag(s)	To display digital values with unit tag(s).
Unit System	To select the unit system between the Metric System and the US/British System for Length, Diameter, Angle, Area and Volume. Use the drop-down to choose one from the list.

To Set a Units Preference:

1. Click on the **Units** tab. The **Units** dialog appears.
2. Set a preference.
3. Click **Apply**. The setting will be taken into account and the **Preferences** dialog still opens.
4. Click **OK**. The setting will be taken into account and the **Preferences** dialog closes.

Print

To Set the Print Preferences:

1. Click on the **Print** tab. The **Print** dialog appears.
2. Click **Open** below the **User Defined Logo** option. The **Import User Defined Logo** dialog opens.
3. In the **Look In** field, navigate to the folder where the file is.
4. Click on the file to select it.
5. Click **Apply**. The setting will be applied and the **Preferences** dialog remains open.
6. Click **OK**. The setting will be applied and the **Preferences** dialog closes.

CHAPTER 5

Exploring Data

Exploration in **RealWorks** involves the comprehension of a complex scene from a loaded project by using all available information, whether it is a point cloud, a model, a set of images or a combination thereof. Exploration can be done via the **Project Tree** in order to understand how a scene is structured. It can also be done by examining (**Examiner** mode) or walking through (**Walkthrough** mode) a scene displayed in the **3D View** or viewing a scene from the instrument positions (**Station-Based** mode).

Expanding and Shrinking the Project Tree

You can explore the **Project Tree** in the **WorkSpace** window in order to determine how the data is structured. You can click on the **Expand** (or **Shrink**) icon located at the left side of each group of nodes to expand (or reduce) the **Project Tree** until you reach the level that contains the information you want. You can also use the scroll bar to go up (or down) the **Project Tree**. Exploration in this tree is similar to that of Microsoft Windows Explorer.

Locating an Item in the Project Tree

Locating an item in the **Project Tree** is done by using the **Locate** command. You can locate any item in the **Project Tree** by clicking on it (or double-clicking) in the **3D View**. Like the **Find** command, the name of the found item will be highlighted in the **List** window and its properties will appear in the **Property** window if it is opened. The father of the found item becomes the **Active Group** in the **Project Tree**.

To Locate an Item in the Project Tree:

1. Select an item from the **3D View**.
2. In the **Edit** menu, select **Locate**.

Finding an Item in the Project Tree

Finding an item in the **Project Tree** is done with the **Find** command. You can find any item by its complete or partial name. When an item is found, the path from the **Project Tree** root to this object will be expanded (if it is not already the case); its direct father group will become the **Active Group**. The name of this found item will be highlighted in the **List** window, and its representation, if displayed in the **3D View**, will also be highlighted. If the **Property** window is opened, the properties of this found item will be shown.

To Find an Item in the Project Tree:

1. From the **Edit**, select **Find**. The **Find** dialog box opens.
 - **Match Case**: This option allows you to find an object with its name case-matched to what was entered in the **Find** field.
 - **Direction**: This option is to specify the searching direction in the **Project Tree**. **Down** direction is the default setting.
2. Enter a name/or a partial name in the **Find What** field.
3. Select an option that corresponds to your needs.
4. Click **Find Next**. The next found object becomes the selected one.
5. Click **Find Next** again if the found objects are more than one.

Note: You can also right-click anywhere in the **3D View** to display the pop-up menu and select **Find** or use the short-cut key **Ctrl + F**.

Exploring in the 3D View

Data exploration in the **3D View** consists mainly of using the navigation commands such as Rotate, Pan, Zoom, etc. to examine the contents of displayed scene/objects.

Exploring in the Images Tree

As explained in the data organization chapter, the **Images Tree** contains only images which can be taken by an on-board camera in a 3D scanner or come from any 2D cameras. You can browse a set of images, compare each of them with a 3D scene in order to have a better perception and understanding, etc.

Visualizing Data

The visualization of objects in the **3D View** is like taking a photograph of a virtual scene with a camera. The steps may typically be the ones given below and each will be discussed separately.

- You specify the rendering parameters for displaying objects.
- You choose camera characteristics; for example, the projection modes.
- You then select objects to display.
- Finally, you choose the point of view from which you want to shoot your photo. This is equivalent to choosing the viewing camera's position and its aiming orientation.


Point Cloud


A point cloud is a set of 3D points. Each 3D point can contain not only its 3D coordinates, but also other attributes such as **Intensity** and **Surface Normal**.

Displaying

To display objects, you should first select them either from the **WorkSpace** window or the **List** window. Once objects are selected, you have the following options to display them. If the selected objects are group nodes, you can either use the command **Display Cloud** from the **Display** menu or its corresponding icon in the **Display** toolbar. You can also use the drag-and-drop function to drop the selected objects in the **3D View**. If the selected objects are object nodes (or scan nodes), you can either use the same methods as above or directly toggle the **On** or **Off** icon beside the name of these selected objects in the **List** window. It is important to note that if the selected objects have both the cloud and the geometry representations, only the later one will be displayed when you use this box.

To Display a Point Cloud:

1. Select an object with cloud property from the **Project Tree**.
2. In the **Display** menu, select **Display Cloud** .

You have two ways to check if a cloud is displayed (or not) in the **3D View**. The first way is to use the **Display Cloud** icon in the **Display** toolbar. It becomes active when you select a cloud which is not displayed in the **3D View**. It becomes inactive when the selected cloud is already displayed. The second way is to use the **List** window. A displayed cloud is turned-on .


Notes:

- You can also right-click in the **WorkSpace** (or **List**) window to get the pop-up menu from which you can choose the **Display Cloud** command.
- If you select a group node and display it, all leaf nodes under this group node will be displayed. This avoids having to display them one by one.

Hiding

Hiding an object with cloud property consists of removing its representation from the **3D View**. For hiding an object, you have to select it either from the **WorkSpace** window or the **List** window or directly from the **3D View**. If the selected object is a group node, you can either use the command **Hide Cloud** from the **Display** menu or its corresponding icon in the **Display** toolbar. If the selected object is a scan node, you can either use the same methods as above or directly toggle the **On** or **Off** icon beside the name of these selected objects in the **List** window.

To Hide a Point Cloud:

1. Select an object with cloud property from the **Project Tree**.
2. In the **Display** menu, select **Hide Cloud** .

Note: You can also right-click on a cloud in the **Project Tree** (or in the **3D View**) so as to display the pop-up menu and select **Hide Cloud**.

Geometry


A point cloud is a set of 3D points. Each 3D point can contain not only its 3D coordinates, but also other attributes such as **Intensity** and **Surface Normal**.

Displaying

To display objects, you should first select them either from the **WorkSpace** window or the **List** window. Once objects are selected, you have the following options to display them. If the selected objects are group nodes, you can either use the command **Display Geometry** from the **Display** menu or its corresponding icon in the **Display** toolbar. You can also use the drag-and-drop function to drop the selected objects in the **3D View**. If the selected objects are object nodes (or scan nodes), you can either use the same methods as above or directly toggle the **On** or **Off** icon beside the name of these selected objects in the **List** window. It is important to note that if the selected objects have both the cloud and the geometry representations, only the later one will be displayed when you use this box.

To Display a Geometry:

1. Select an object with geometric property from the **Project Tree**.
2. In the **Display** menu, select **Display Geometry** .

You have two ways to check if a geometry is displayed or not in the **3D View**. The first way is to use the **Display Geometry** icon in the **Display** toolbar. It becomes active when you select a geometry which is not displayed in the **3D View** and inactive when the geometry is displayed in the **3D view**. The second way is to use the **List** window. A displayed geometry is turned-on .


Notes:

- You can also right-click in the **WorkSpace** (or **List**) window to get the pop-up menu from which you can choose the **Display Geometry** command.
- If you select a group node and display it, all leaf nodes under this group node will be displayed. This avoids having to display them one by one.

Hiding

Hiding an object with geometric property consists of removing its representation from the **3D View**. For hiding an object, you have to select it either from the **WorkSpace** window or the **List** window or directly from the **3D View**. If the selected object is a group node, you can either use the command **Hide Geometry** from the **Display** menu or its corresponding icon in the **Display** toolbar. If the selected object is an object node, you can either use the same methods as above or directly toggle the **On** or **Off** icon beside the name of these selected objects in the **List** window.

To Hide a Geometry:


1. Select an object with geometric property from the **Project Tree**.
2. In the **Display** menu, select **Hide Geometry** .

Note: You can also right-click on an object with geometric property in the **Project Tree** (or in the **3D View**) to display the pop-up menu and select **Hide Geometry**.

Hiding all

This command enables to hide all displayed objects at once in the **3D View**, no matter the objects could be.

To Hide all:

1. In the **Display** menu, select **Hide All** .
2. Or click **Hide All** in the **Display** toolbar.

Note: You can also right-click anywhere in the **3D View** in order to display the pop-up menu and select **Hide All**.

Hiding all Except those Selected

You can hide other objects than the one you have selected. To do this, you can use the command **Hide Other** from the **Display** menu or the corresponding icon in the **Display** toolbar. You can also evoke this command from the right-click pop-up menu.

To Hide all Except those Selected:

1. Select an object from the **Project Tree**.
2. In the **Display** menu, select **Hide Other**.

Note: This command is not available when selecting a project.

Image

You can display an image in two ways either as a thumbnail in the **3D View** or in a separated 2D window beside the **3D View**. In the first case, you can only display an image once at a time. You can use a tip to check if an image is displayed or not; it consists of using the **List** window. An image when displayed has an **On**-bulb icon at its left side. In the second case, you can open as many images as required and no tip is available.

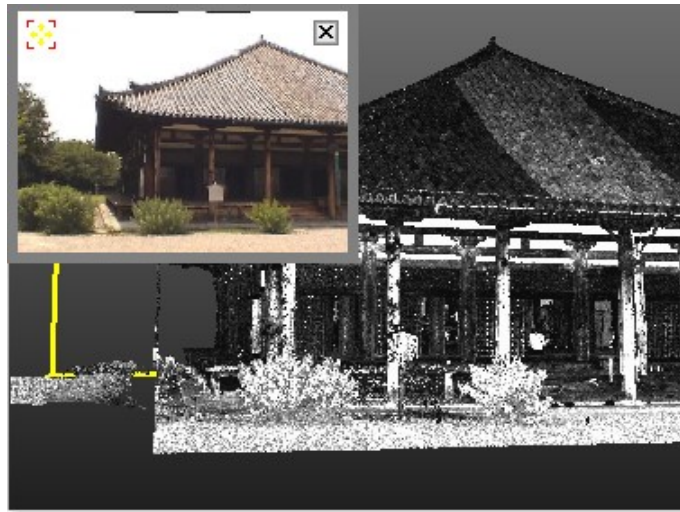
Note: An image (selected from the **Images Tree**) has its thumbnail displayed in the **Property** window (if open).


Displaying

Displaying an image consists of opening it as a thumbnail within the **3D View**. You can only display one image at once

To Display an Image:

1. Select an image from the **Images Tree**.
2. Do one of the following:
 - In the **List** window, toggle the **On/Off** icon beside the image name to **On**.
 - In the **Display** menu, select **Display Image**.
 - Right-click on an image in the **Images Tree** and select **Display Image** from the pop-up menu.



- **Resize:** Place the mouse cursor anywhere on the thumbnail image frame. Stretch or shrink the frame by dragging.
- **Change location:** Place the mouse cursor over the **Drag & Drop**  icon on the thumbnail image. Drag and drop the thumbnail image to a suitable location in the **3D view**.
- **Zoom:** Click in the thumbnail image and zoom it **In** or **Out** using the mouse wheel (if existed).


Hiding

The command enables to close an image that is open as a thumbnail in the **3D View**. You can only hide one image at once.

To Hide an Image:

1. Select an image opened as a thumbnail.
2. In the **Display** menu, select **Hide Image**.

Tips:

- You can also right-click on an image in the **List** window and select **Hide Image** from the pop-up menu.
- Select an image from the **List** window and toggle the **On/Off** icon to **Off**.
- Move your cursor over the thumbnail image and click the **Close**  icon.


Opening

Once an image is open in a separated window, you can zoom an area of this image **In** or **Out** using the **Zoom In** and **Zoom Out** commands, zoom the whole image **In** or **Out** using the mouse wheel or by defining a zoom factor. If the image is zoomed **In** more than the 2D window can display, you can pan it in any direction in order to view the hidden areas.

To Open an Image:

1. Select an image from the **Images Tree**.
2. Do one of the following:
 - In the **Display** menu, select **Open Image**.
 - In the **List** window, double-click on its name.
 - In the **List** window, right-click on its name and select **Open Image** from the pop-up menu.



3. To close the image window, click on the **Close**  icon on the border of its frame.

Note: You cannot perform a distance measurement on an image that is open in a separate window; the **Measurement Tool** icon is dimmed.

Inspection Map

An inspection map results from the comparison between two surfaces (cloud/cloud, cloud/mesh, mesh/mesh and cloud (or mesh)/primitive, etc.). An inspection map may have three shapes: **Plane**, **Cylinder** and **Tunnel**; this depends on the surfaces selected for comparing and the projection type applied to that comparison. Each inspection map has two directions (**Vertical** and **Horizontal**) shown by its own red-and-green-axis frame and a color bar associated to it.

Note: To view which shape has an inspection map, display its properties in the **Property** window and check for the **Projection Type** (**Planar** for a plane, **Tunnel** for a tunnel and **Cylinder** for a cylinder).

Opening

If the project you load contains an existing inspection map, you can open it in an independent window.



To Open an Inspection Map:

1. Select an inspection map from the **Project Tree**.
2. In the **Display** menu, select **Open Inspection Map**.

The inspection map is shown in a specific window (beside the **3D View**) called by the inspection map's name. Note that this window is called **Map Preview** inside comparison/inspections tools.

Inside this window, you can zoom the inspection map **In** or **Out**. If it is bigger than the window can show, you can pan it on left-click in any direction.

Tips:

- You cannot toggle the **On**  (or **Off** ) icon beside an inspection map in the **List** window to display (or hide) it.
- You can also right-click on an inspection map in the **Models Tree** to display the pop-up menu and select **Open Inspection Map**.

Closing

To Open an Inspection Map:

- Click on the **Cross** button at the top right corner of the window.

Station



You can display the position, name and properties of a station (or of all stations) within a project.

Positions

The position of a station can now be displayed in the **3D View** regardless of the navigation mode (**Examiner**, **Walkthrough** or **Station-Based**) you are in.

Displaying

To Display Positions Within the Project:

1. From the **3D View** menu, select **Rendering**. A sub-menu drops down.
2. Select **Display Stations** . The **Display Station Labels** icon becomes enabled. Each station's position within the project is marked as follows .





Tips:

- You can also select **Display Stations** from the **3D View** toolbar.
- You can jump from one station to another by double-clicking on the station position icon in the **3D View**. The navigation mode will be automatically switched to the **Station Based** navigation mode.

Hiding

To Hide Positions from the Project:

1. From the **3D View** menu, select **Rendering**. A sub-menu drops down.
2. Select **Display Stations** . The **Display Station Labels** icon becomes dimmed. Each position marked as follows  is removed from the project.

Labels

For a given station, a label is its name in text displayed in the **3D View**, next to its position.

Displaying

To Display Labels:

1. In the **3D View** menu, select **Rendering**. A sub-menu drops down.
2. Select **Display Station Labels** **1**.



Tip: You can also select **Display Station Labels** from the **3D View** toolbar.

Hiding

To Display Labels:

1. In the **3D View** menu, select **Rendering**. A sub-menu drops down.
2. Select **Display Station Labels** **1**.

Network Visuals of a Single Station

The **Network Visuals** in the **3D View** is similar to a set of vectors, each vector connecting the 3D position of the station (or a point on the ground (if that station has a height)) to a registration target (or to its point on the ground (if the target has a height)).

Note: The user should be in the **Registration** processing mode.

Displaying

To Display the Network Visuals of a Station:

1. First display the 3D position of the stations.
2. In the **3D View**, right-click on a 3D position.
3. Select **Display/Hide Network Visuals (Selected Station)** from the pop-menu.

Or

4. In the **Registration** processing mode, select a single (or a set of) station(s) from the **List** window.
5. Right-click to display the pop-menu.
6. Select **Display/Hide Network Visuals (Selected Station)**.

Or

7. In the **Registration** processing mode, select a single (or a set of) station(s) from the **List** window.
8. From the **Display** menu, select **Display/Hide Network Visuals (Selected Station)**.



Note: For the steps from 4 to 6 and 7 to 8, you cannot view the network visuals if the 3D position(s) of the station(s) is (or are) not displayed.

Hiding

To Hide the Network Visuals of a Station:

1. In the **3D View**, right-click on the same 3D position.
2. Select **Display/Hide Network Visuals (Selected Station)** from the pop-menu.

Or

3. In the **Registration** processing mode, select the same lonely (or set of) station(s) from the **List** window.
4. Right-click to display the pop-menu.
5. Select **Display/Hide Network Visuals (Selected Station)**.

Or

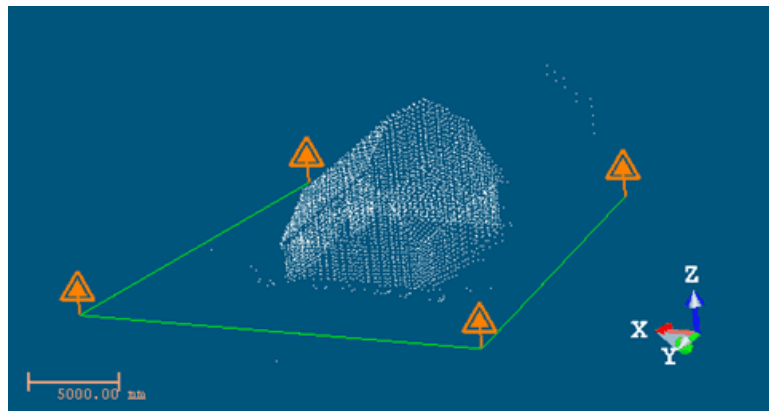
6. In the **Registration** processing mode, select the same lonely (or a set of) station(s) from the **List** window.
7. From the **Display** menu, select **Display/Hide Network Visuals (Selected Station)**.

Note: (*) You cannot get the pop-up menu when selecting a set of stations.

Network Visuals of all Stations

This feature is similar to the [Display/Hide Network Visuals \(Selected Station\)](#) feature but applied to all stations in the project. The user can be in any processing mode: [Registration](#), [OfficeSurvey](#) or [Modeling](#).

In the [OfficeSurvey](#) (or [Modeling](#)) processing mode, both the 3D positions and the [Network Visuals](#) are shown.



In the [Registration](#) processing mode, each station is shown by its 3D position, its height (if existed), the registration targets that are inside as well as their height (if existed).



Note: No station selection is required except in the [3D View](#).

Displaying

To Display the Network Visuals of all Stations:

1. In the **Registration** processing mode, first display the 3D position of the stations.
2. In the **3D View**, right-click on a 3D position.
3. Select **Display/Hide Network Visuals (All Stations)** from the pop-menu.

Or

4. In the **Registration** processing mode, select a single (or a set of) station(s) from the **List** window.
5. Right-click to display the pop-menu.
6. Select **Display/Hide Network Visuals (All Stations)**.

Or

7. In the **Registration** (or **OfficeSurvey** or **Modeling**) processing mode, select **Display/Hide Network Visuals (All Stations)** from the **Display** menu.

Note: For the steps from 4 to 6 and 7, you cannot view the network visuals if the 3D positions of the stations are not displayed.

Hiding

To Hide the Network Visuals of all Stations:

1. In the **3D View**, right-click on any 3D position.
2. Select **Display/Hide Network Visuals (All Stations)** from the pop-menu.

Or

3. In the **Registration** processing mode, select any single (or set of) station(s) from the **List** window.
4. Right-click to display the pop-menu.
5. Select **Display/Hide Network Visuals (All Stations)**.

Or

6. In the **Registration** (or **OfficeSurvey** or **Modeling**) processing mode, select **Display/Hide Network Visuals (All Stations)** from the **Display** menu.

TZF Scan

A **TZF Scan** is a **2D Map Data**, linked to a **TZF** format file which can come from a **TZS**, **Faro** and etc. format file conversion. There is only one **TZF Scan** per station.

Displaying

Displaying a **TZF Scan** consists of opening it as a thumbnail within the **3D View**. You can only display one **TZF Scan** at a time.

To Display a TZF Scan:

1. Right-click on a **TZF** scan in the **Scans Tree**.
2. Select **Display Image** from the pop-up menu.

Tip: You can also select **Display Image** from the **Display** menu.

Hiding

Hiding a **TZF Scan** consists of closing its thumbnail displayed within the **3D View**.

To Display a TZF Scan:

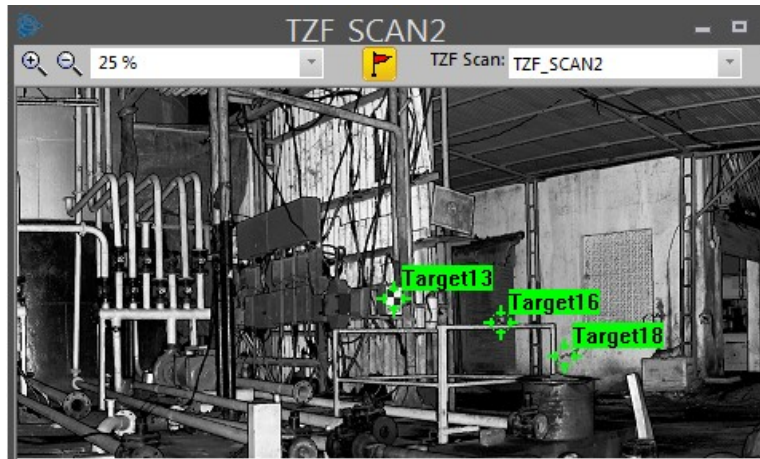
1. Right-click on a **TZF** scan in the **Scans Tree**.
2. Select **Hide Image** from the pop-up menu.

Tip: You can also select **Hide mage** from the **Display** menu.

2D Preview Image

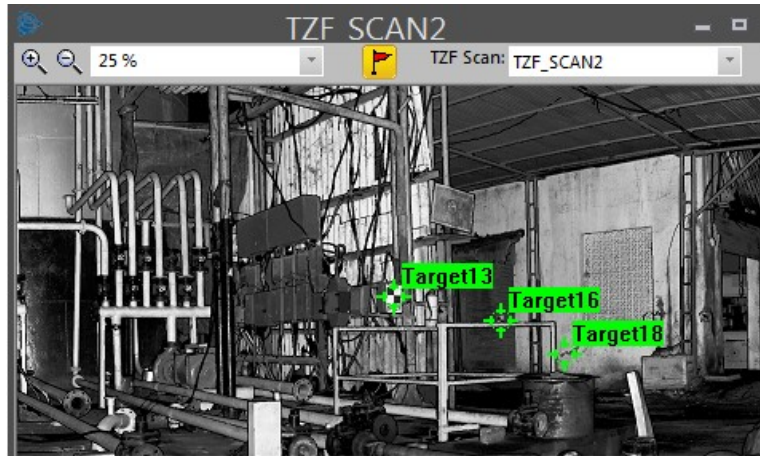
You can zoom an area of this **2D Preview Image In** or **Out** using the **Zoom In** and **Zoom Out** commands, zoom the whole image **In** or **Out** using the mouse wheel or by defining a zoom factor. If the image is zoomed **In** more than the **2D Viewer** can display, you can pan it in any direction in order to view the hidden areas.

Spherical Targets, **Black and White Flat Targets** or **Point Targets** extracted from a **TZF Scan** by using e.g. the **Auto-Extract Targets and Register** feature, once created, are displayed within the **TZF Scan**.



Extracted Targets

Spherical Targets, Black and White Flat Targets or Point Targets extracted from a TZF Scan by using e.g. the Auto-Extract Targets and Register feature, once created, are displayed within the TZF Scan.











A Target selected from the List window is highlighted in the TZF Scan (once open) as shown below.



Rendering Data

Renderings define how an object is displayed. For example, a geometric object can be displayed in wire-frame or shaded surface. We introduce the different options available for different object representations. Renderings will be applied to all objects of the same type displayed in the **3D View**. This means that you cannot specify different renderings for different displayed objects of the same type. For example, you cannot display **Geometry Object A** in **Wire-Frame**, and the **Geometry Object B** in **Surface**.

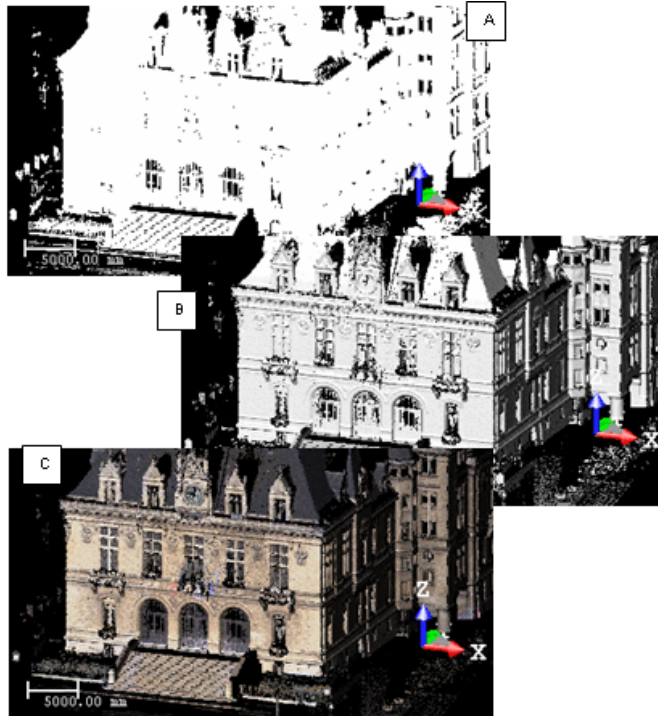
Point Cloud

There are several renderings that you can apply to displayed point clouds. **White Color**  enables to render all displayed points with white color. **Cloud Color**  enables to render displayed points with the color of the clouds they belong to. **Station Color**  enables to render displayed points with the color of the stations they belong to. **Scan Color**  enables to render displayed points with the color of the scans they belong to. **Grey Scaled Intensity**  enables to render displayed points using the gray scale defined by their intensity. **Color Coded Intensity**  enables to render displayed points using the color encoded intensity. **True Color**  enables to render displayed points using their color. You can choose these options from either the menu bar or the corresponding icons in the **3D View** toolbar. There are three special modification options for displaying points. **Normal Shading**  shades each displayed point by using its normal information. **Point Size** changes the point size when displaying the point clouds. **Discontinuity Display*** displays the discontinuity of a point cloud (where available).

To Render a Point Cloud:

1. Select an object with cloud property from the **Project Tree** and display it.
2. In the **3D View** menu, select **Rendering**. A sub-menu drops down.
3. Select a rendering from the drop-down sub-menu.

For point clouds, you can combine the **Normal Shading** rendering with any of the renderings named above (except for **Point Size** and **Discontinuity Display**) in order to have a relief (or depth) display. In (A), the **White Color** rendering is applied to the point cloud in selection. The **Normal Shading** information is added to the **White Color** rendering in (B) and to the **True Color** rendering in (C). When such a combination is applied, you can use the **Lighting Direction** tool to modify the light source position. Note that such a combination reduces by half the time required to display points.

**Tips:**

- You can right-click anywhere in the **3D View** (except on displayed objects) and select **Rendering** from the pop-up menu. A sub-menu drops down. Select then a rendering option.
- You can click on the **Change Cloud Color** pull-down arrow on the **3D View** toolbar and choose a rendering option.

Note: The **Discontinuity Display** rendering, removed from the **3D View** toolbar, is still present in the **3D View / Rendering** menu.

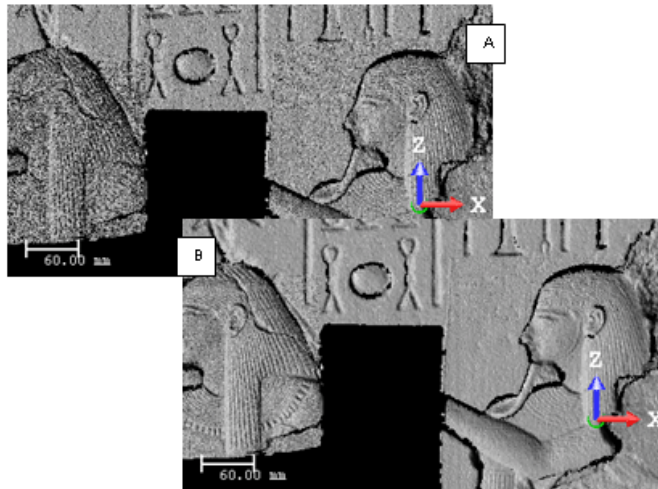
Geometry

There are four renderings that you can apply to displayed geometries. **Wireframe** enables to render the selected geometries in wire-frame. **Hidden Lines** enables to render the selected geometries in wire-frame with hidden lines removed. **Surface** is to render the selected geometries as smooth shaded surfaces. **Textured** renders the selected geometries as a texture mapped surface if such a mapping exists.

To Render a Geometry

1. Select a geometry from the **Project Tree** and display it.
2. In the **3D View** menu, select **Rendering**. A sub-menu drops down.
3. Select a rendering from the drop-down sub-menu.

The **Normal Shading** rendering - when applied to a mesh displayed in **Wireframe** or **Hide Lines** - produces no effect. The same rendering when applied to a mesh displayed in **Surface** adds a smooth rendering (one color per triangle with gradient effect) and when not applied gives a flat rendering (one color per triangle with no gradient effect). The **Normal Shading** rendering - when applied to a textured mesh - adds a relief (depth) effect. In (A), the **Surface** rendering is applied to the mesh in display. The **Normal Shading** information is added to the **Surface** rendering in (B). When such a combination is applied, you can use the **Lighting Direction** tool - see next topic - to modify the light source position. Note that such a combination reduces by half the time required for displaying the mesh.




Tips:

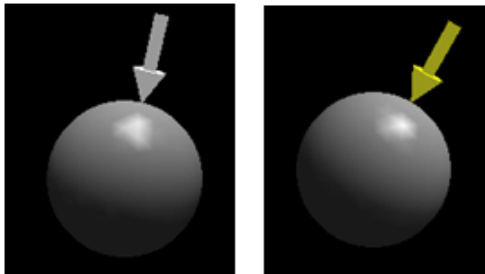
- You can right-click anywhere in the **3D View** (except on displayed objects) and select **Rendering** from the pop-up menu. A sub-menu drops down. Select then a rendering option.
- You can click on the **Change Geometry Color** pull-down arrow on the **3D View** toolbar and choose a rendering option.






Lighting Directions

The **Lighting Direction** feature lets you produce infinity of lighting effects on complex objects. You can change the spotlight's direction or use pre-defined light effects - five are available. Note that lighting effect modifications will be applied to the current (active) **3D View**.

To Add Lighting Directions:

1. If required, display objects in the **3D View**.
2. In the **3D View** menu, select **Lighting Direction** . The **Lighting Direction** dialog opens.
3. Do one of the following:
 - Change the spotlight's direction at will.
 - a) Place the mouse cursor over the handle.
 - b) Drag the handle. The spotlight direction changes and the handle color switches from grey to yellow.
 - c) Drop the handle when you have the desired light effect. The handle returns again to its previous color.



- Change the spotlight's direction using pre-defined light effects.
 - Click  to light from the top left position.
 - Click  to light from the down right position.
 - Click  to light from the top right position.
 - Click  to light from the down left position.
 - Click  to return to the default position.

Note: Instead of selecting **Lighting Direction** from the menu bar, you can also click on its icon in the **Display** toolbar.

Navigating Through Data


In the last session, we mentioned that the visualization of objects in the **3D View** is like taking a photograph with a camera. A photograph is a static view of the scene being visualized. If we can modify the camera position continuously, we can obtain the so-called object animation effects. We call this the navigation of the scene. Here, the modification of the camera position will be executed interactively by the user. In **RealWorks**, there are mainly three different ways for navigating through a 3D scene: **Examiner**, **Walkthrough** and **Station-Based**.

Projection Modes

There are two projection modes. In **Isometric**, the distance from the viewing camera origin to displayed objects has no impact on how large an object appears. In **Perspective**, the most unmistakable characteristic is foreshortening - the further an object from the viewing camera, the smaller it appears in the final screen image.

Perspective

To Set the Perspective Mode:

1. In the **3D View** menu, select **Mode**. A sub-menu drops down.
2. Select **Perspective**  from the drop-down submenu.


Tips:

- You can right-click anywhere in the **3D View** (except on displayed objects) and select **Mode / Perspective** from the pop-up menu.
- You can also click on the **Projection Mode** pull-down arrow in the **3D View** toolbar and choose **Perspective** from the list.

Note: In **Perspective**, you can use any of the displacement modes (Walkthrough, Examiner or Station-Based).

Isometric

To Set the Isometric Mode:

1. In the **3D View** menu, select **Mode**. A sub-menu drops down.
2. Select **Isometric**  from the drop-down submenu.

Note: The scale at the left down corner of the **3D View** is only available in **Isometric**.

Tips:

- You can right-click anywhere in the **3D View** (except on displayed objects) and select **Mode** / **Isometric** from the pop-up menu.
- You can also click on the **Projection Mode** pull-down arrow in the **3D View** toolbar and choose **Isometric** from the list.

Note: In **Isometric**, you can only use the Examiner mode. Choosing **Isometric** will automatically swap the displacement mode to **Examiner**.

Displacement Modes

To Set a Displacement Mode:


1. From the **3D View** menu, select **Mode**. A submenu drops down.
2. Choose among **Examiner**, **Walkthrough** and **Station-Based** from the submenu.

Tips:

- You can right-click anywhere in the **3D View** (except on displayed objects) and select **Mode** from the pop-up menu. A sub-menu drops down. Then select **Examiner**, **Walkthrough** or **Station-Based**.
- You can click on the **Displacement Mode** pull-down arrow on the **3D View** toolbar and choose among **Examiner**, **Walkthrough** or **Station-Based** from the drop-down menu.

Examiner

The **Examiner** mode is the base mode of **RealWorks**, that is, it is the default navigation mode when you start the software. In this mode, you turn the camera around an object. These operations are actually obtained by moving the scene with a mouse. Navigation is free inside the **3D View**; we call this mode the **Standard Navigation**.


Note: If the **Head Always Up (Z Axis)** option has been checked in **Preferences**,  is displayed at the bottom right corner of the **3D View**.


Without Constraints

Navigation in the **Examiner** mode can be free (without constraints).

Rotate


To Rotate Around the Center of the Screen:

1. Press the left button of the mouse. The cursor takes the following shape .
2. Drag the mouse in a **Direction** while holding the left button pressed to rotate in that **Direction** and around the center of the screen*.

Note: (*) You can change the center of rotation using .


Pan

To Pan in a Direction:

1. Press the middle button of the mouse. The mouse takes the following shape .
2. Drag the mouse in a **Direction** while holding the button pressed to pan in that **Direction**.

Zoom In (or Out)

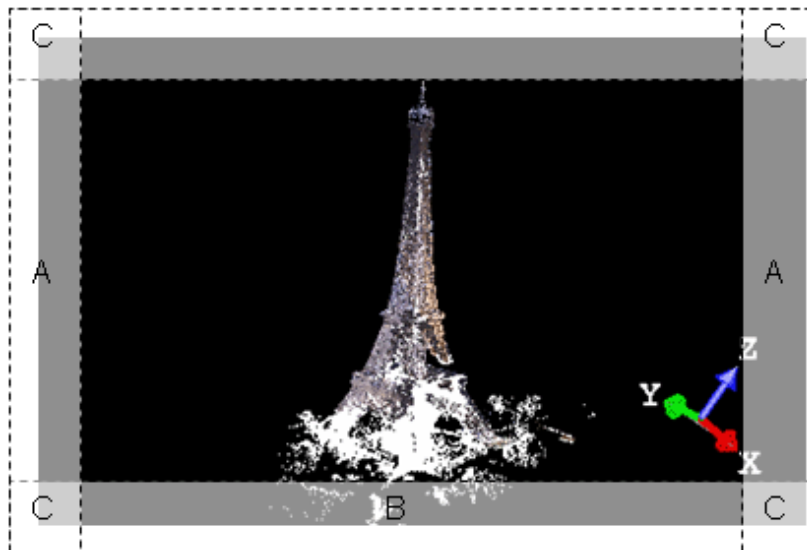
To Zoom In (or Out):

1. Press the left and middle buttons together. The mouse takes the following shape .
2. Drag the mouse **Forward*** while holding both pressed to **Zoom Out**.
3. Drag the mouse **Backward*** while holding both pressed to **Zoom In**.
4. Or use the mouse wheel.

Tip: (*) You can reverse the mouse for zoom in **Preferences**.

With Temporary Constraints

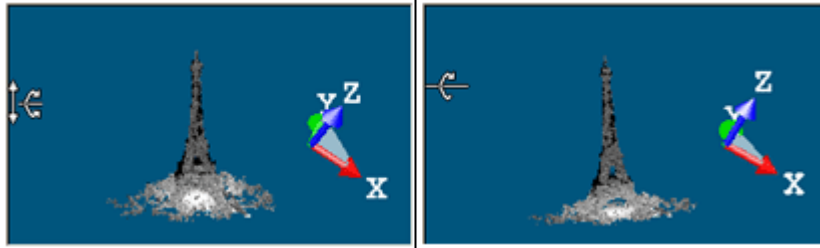
You can navigate under constraint according to three directions: **Horizontal**, **Vertical** and **Perpendicular-to-the-Screen**. Constraint can be temporary; this enables to switch between free navigation and constrained navigation. This illustration shows where to place the cursor in the **3D View** to activate the temporary constraint - four strips (left, right, top and bottom) along each side of the **3D View**. For each constraint, you may see two shapes that the cursor takes when you position it upon these strips. The first shape corresponds to the cursor shape before activating the temporary constraint mode. The second shape corresponds to the cursor shape when you activate the temporary constraint mode.



Horizontal Rotation

To Rotate a Scene Constrained Under a Vertical Axis:

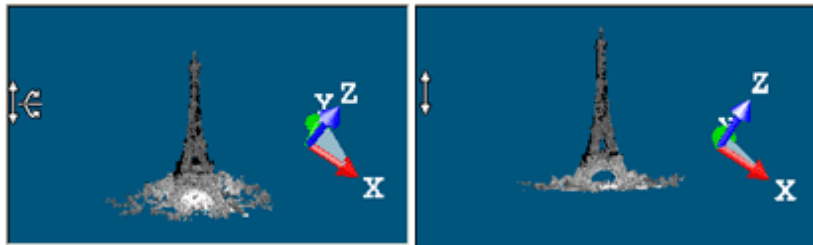
1. Place the cursor anywhere along the right (or left) side of the **3D View (A)**.
2. Click on the left button of your mouse.
3. Move your mouse from up to down and in reverse to rotate a scene constrained under the horizontal direction.



Vertical Pan

To Pan a Scene Constrained Under a Vertical Axis:

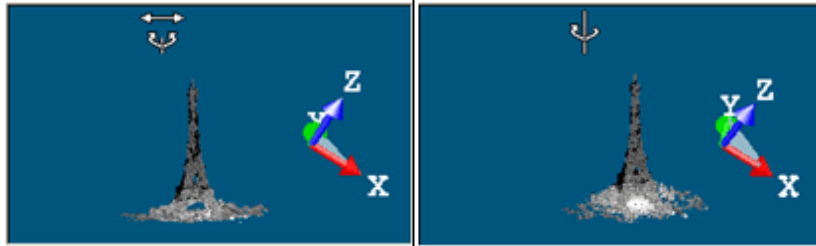
1. Place the cursor anywhere along the right (or left) side of the **3D View (A)**.
2. Click on the middle button of your mouse.
3. Move your mouse from top to bottom and in reverse to pan a scene constrained under the vertical direction.



Vertical Rotation

To Rotate a Scene Constrained Under a Vertical Axis:

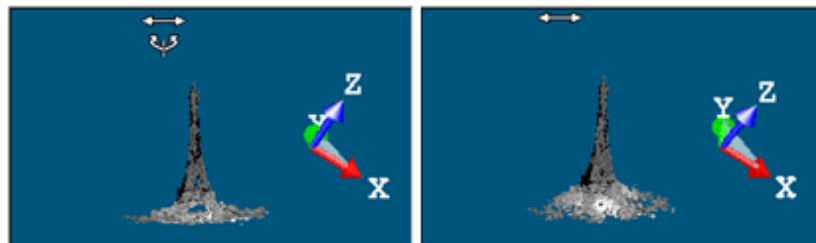
1. Place the cursor anywhere along the top of the **3D View** (B).
2. Click on the left button of your mouse.
3. Move your mouse from right to left and in reverse to rotate a scene constrained under the vertical direction.



Horizontal Pan

To Pan a Scene Constrained Under a Horizontal Axis:

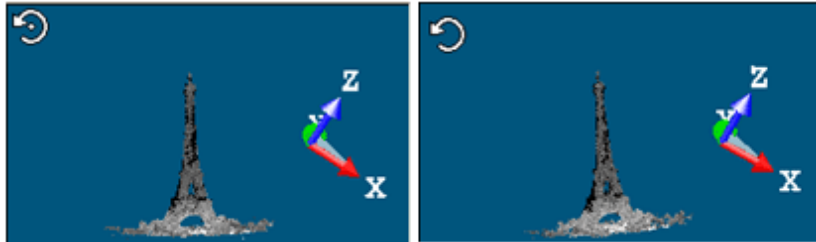
1. Place the cursor anywhere along the top of the **3D View** (B).
2. Click on the middle button of your mouse.
3. Move your mouse from right to left and in reverse to pan a scene constrained under the horizontal direction.



Screen Rotation

To Rotate a Scene Constrained Under an Axis Perpendicular to the Screen:

1. Place the cursor anywhere at one of the four corners of the **3D View (C)**.
2. Click on the left button of your mouse and move it clockwise and anticlockwise and in reverse to rotate a scene constrained under an axis perpendicular to the screen.



Note: You cannot rotate a scene constrained under an axis perpendicular to the screen if the **Head Always Up (Z Axis)** option is checked in the **Preferences** dialog.


With Permanent Constraints

Constraints can also be permanent (for all navigation purposes) by evoking commands.

Tips:

- You can also right-click on the constraint mode icon at the right side of the status bar to display the pop-up menu.
- For all navigations with permanent constraint, you can right-click anywhere in the **3D View** (except on displayed objects) to display the pop-up menu and select **Mode**. A sub-menu drops down. Then select the constraint mode you want.

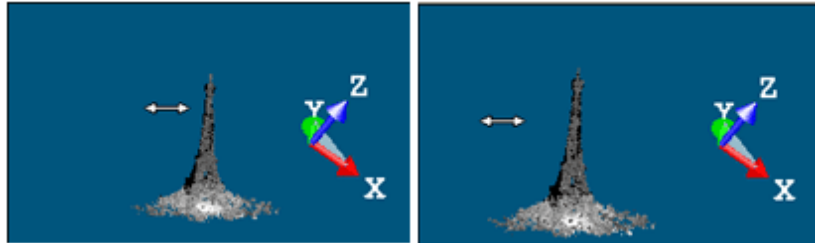
Notes:

- You can use the **Alt** key to slow down all navigations in the **3D View**, free or permanent constraint.
- To get the **Standard Navigation**  mode back, you can double-click on the constraint mode icon at the right side of the status bar.

Horizontal Pan

To Pan Constrained Under a Horizontal Axis:

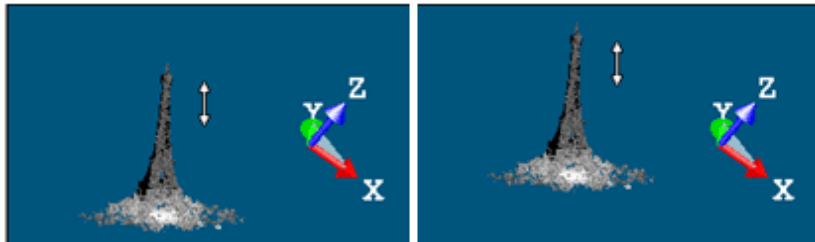
1. From the **3D View** menu, select **Mode**. A submenu drops down.
2. Select **Horizontal Pan** from the submenu.
3. Press on the middle button of your mouse and move it from left to right and in reverse to pan the displayed constrained under the horizontal axis.



Vertical Pan


To Pan Constrained Under a Vertical Axis:

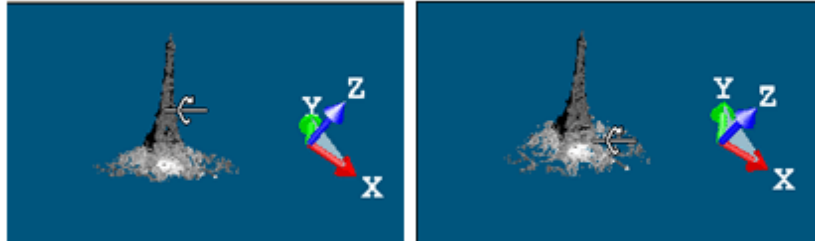
1. From the **3D View** menu, select **Mode**. A submenu drops down.
2. Select **Vertical Pan** from the submenu.
3. Press on the middle button of your mouse and move it from top to bottom and in reverse to translate the displayed constrained under the vertical axis.



Horizontal Rotation


To Rotate Constrained Under a Horizontal Axis:

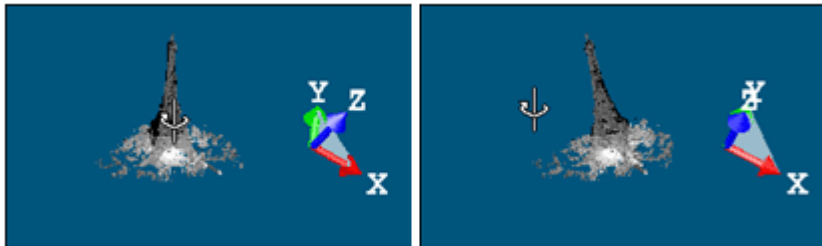
1. From the **3D View** menu, select **Mode**. A submenu drops down.
2. Select **Horizontal Rotation**  from the submenu.
3. Press on the left button of your mouse and move it from top to bottom and in reverse to rotate the displayed constrained under the horizontal axis.



Vertical Rotation


To Rotate Constrained Under a Vertical Axis:

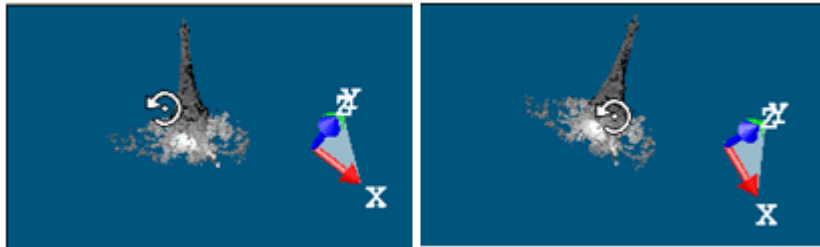
1. From the **3D View** menu, select **Mode**. A submenu drops down.
2. Select **Vertical Rotation**  from the submenu.
3. Press on the left button of your mouse and move it from left to right and in reverse to rotate the displayed constrained under the vertical axis.



Screen Rotation


To Rotate Constrained Under an Axis Perpendicular to the Screen.

1. From the **3D View** menu, select **Mode**. A submenu drops down.
2. Select **Screen Rotation**  from the submenu.
3. Press on the left button of your mouse and move it clockwise and anticlockwise to rotate the displayed scene constrained under an axis perpendicular to the screen.



Walkthrough


In the **Walkthrough** mode, you use the mouse movement to simulate a walking through of the displayed scene. You cannot apply constraints in this navigation mode.

Note: If the **Head Always Up (Z axis)** option has been checked in the **Preferences** dialog,  is displayed in the bottom right corner of the **3D View**.

Tip: If **Isometric** is the current projection mode, choosing **Walkthrough** will automatically swap the projection mode to **Perspective**.


Tilt (or Rotate)

To Tilt (or Rotate) "Look at" a Direction:

1. Press the left button of the mouse. The cursor takes the following shape .
2. Drag the mouse while holding the button pressed.


Pan

To Pan in a Direction:

1. Press the middle button of the mouse. The cursor takes the following shape .
2. Drag the mouse in a **Direction** while holding the button pressed to pan in the opposite **Direction**.

Walkthrough

To Walkthrough a Scene (or Objects):

1. Press the left and middle buttons together. The cursor takes the following shape .
2. Drag the mouse **Forward*** while holding both pressed to **Walk Inside** a scene (objects).
3. Drag the mouse **Backward*** while holding both pressed to **Walk Out Of** a scene (or objects).
4. Or use the mouse wheel.

Tip: (*) You can reverse the mouse for zoom in **Preferences**.

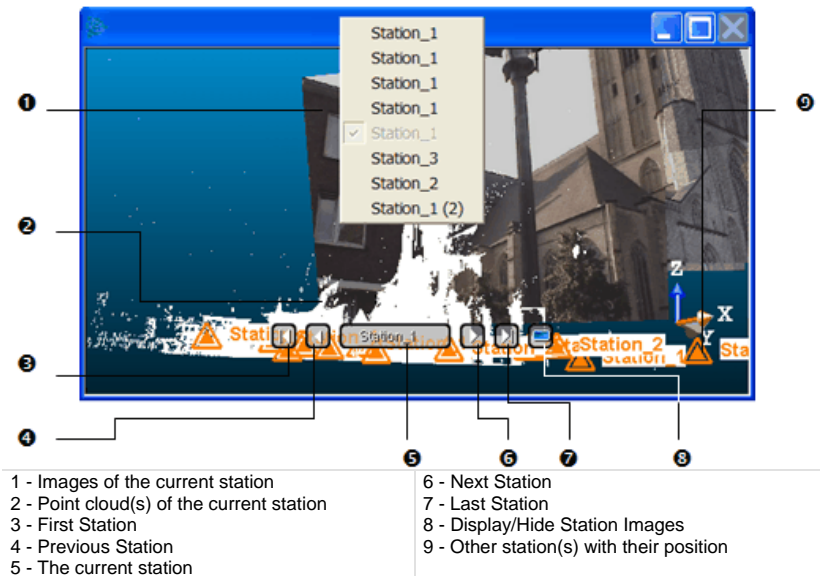
Station-Based

In **Station-Based**, a scene is viewed from the viewpoint of one of the stations, i.e., the instrument location for this station, and you can jump from one station to another (if there is more than one).

Tip: If **Isometric** is the current projection mode, choosing **Station-Based** will automatically swap the projection mode to **Perspective**.

Browsing Through Stations

You can view a scene from the instrument viewpoint. In this particular viewing mode, the **Head Always Up (Z Axis)** preference and the **Perspective** projection mode are both set. Note that navigating through a scene in such mode is restricted to rotating and zooming (in or out). You cannot apply temporary (or permanent) constraints while navigating through a scene. Below are an illustration and a description of items of that mode.




To Browse Through Stations:

1. Display objects in the **3D View** (if not present).
2. From the **3D View** menu, select **Mode**. A submenu drops down.
3. Select **Station-Based** from the submenu.



The scene is viewed from the first station viewpoint (the first in the **Project Tree**) with overlapped images in the background (see the upper illustration). You can use the **AutoSpin** feature in **Preferences** to endlessly turn a scene around the station position.

4. Do one of the following:
 - To go to the first (or last) station, click the **First Station** (or **Last Station**) button.
 - To go to the next (previous) station, click the **Next** (or **Previous Station**) button.

- Click the current station button and choose another station from the drop-down list. The current station is grayed out and has a check mark at its side.
 - Double-click on a station position .
5. If required, hide the images by clicking the **Display/Hide Station Images** button.


Note: To leave the **Station-Based** mode, choose between **Examiner** and **Walkthrough** from either the **3D View > Mode** menu or the pop-up menu.

Tips:


- In the **Scans Tree**, if you select a station from the **Project Tree**, right-click and select **Station-Based Mode** from the pop-up menu; the selected scene will be viewed from the selected station viewpoint. If no station has been selected, the scene will be viewed from the first station viewpoint.
- The **Display/Hide Station Images** button takes the following shape  (or  when images are hidden (or displayed)).
- You can also select a station from the **Station Navigation > Edit** menu.

Navigating Within a Station

To Rotate Around the Viewpoint of the Current Station:

1. Press the left button of the mouse. The cursor takes the following shape .
2. Drag the mouse in a **Direction** while holding the button pressed to rotate in that **Direction** and around the viewpoint of the current station.

To Zoom In (or Out):

1. Press the left and middle buttons together. The mouse takes the following shape .
2. Drag the mouse **Forward*** while holding both pressed to **Zoom Out**.
3. Drag the mouse **Backward*** while holding both pressed to **Zoom In**.
4. Or use the mouse wheel.

Note: (*) You can reverse the mouse for zoom in **Preferences**.

View Alignment

At any time during the navigation in all the navigation modes (**Examiner**, **Walkthrough** and **Station-Based**), you can re-align your view frustum by using the following functions: **Zoom In**, **Zoom Out**, **Zoom on Selection**, **Zoom All**, **Center on Point** and **Go to Shooting Position**.

Zooming In (or Out)

The **Zoom In** feature enables to fit a fenced zone into the whole **3D View** while the **Zoom Out** feature enables to fit the whole the **3D View** into a fenced zone.

To Zoom In (or Out):

1. In the **3D View** menu, select **View Alignment**. A submenu drops down.
2. Do one of the following:
 - Select **Zoom In**.
 - Select **Zoom Out**.
3. Draw a fence in the **3D View** window.

Note: Before drawing a fence, pressing **Esc** will leave the **Zoom In** or **Zoom Out** tool.

Tip: You can use the mouse buttons **Left + Right** to zoom in and out.

Zooming All

The **Zoom All** feature enables to fit the whole displayed scene into the **3D View** (except in **Station-Based**, where the field of view is limited).

To Zoom All:

1. In the **3D View** menu, select **View Alignment**. A submenu drops down.
2. Select **Zoom All** from the sub-menu.

Tips:

- You can right-click anywhere in the **3D View** (except on displayed) and select **View Alignment** from the pop-up menu. A sub-menu drops down. Select then **Zoom All**.
- You can use the short-cut key **Home** instead of selecting the **Zoom All** command.

Zooming on Selection

The **Zoom on Selection** feature enables to fit an object (or a set of objects) in selection into the **3D View**.

To Zoom on Selection:


1. Select an object either in the **3D View** or from the **Project Tree**.
2. In the **3D View** menu, select **View Alignment**. A submenu drops down.
3. Select **Zoom on selection** from the sub-menu.

Tip: You can right-click on an object (cloud or geometry) in the **3D View** and select **Zoom On Selection** from the pop-up menu and

Centering on Point

The **Center on Point** feature enables to locate a center of rotation onto a selected point (in **Examiner**) or to merely view towards this point (in **Station-Based** and **Walkthrough**).

To Center on a Point:

1. In the **3D View** menu, select **View Alignment**. A submenu drops down.
2. Select **Center on Point** from the sub-menu. The **Picking Parameters** toolbar appears and the cursor becomes as follows .
3. Pick a point on displayed objects.


Note: Before picking a point, press **Esc** will leave the **Center on Point** tool.

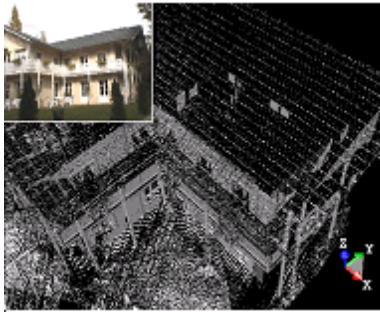
Tip: You can use the **X** key on your keyboard as a shortcut.

Going to a Shooting Position

This feature allows you to align your viewing frustum from the shooting position of an image as registered to the 3D data. It is not accessible if the selected image has no 3D data correspondence.

To Go to a Shooting Position:

1. Select an image from the **Images Tree**.
2. Display (or open) the image if required.
3. In the **3D View** menu, select **View Alignment**. A submenu drops down.
4. Select **Go to Shooting Position**  from the drop-down menu.



The first screen capture shows an image thumbnail with a displayed cloud



The second image shows the view alignment after the Go to Shooting Position command is executed

Tips:

- You can right-click on an image and select **Go to Shooting Position** from the drop-down menu.
- You can click on the **Go to Shooting Position** icon on the **View Alignment** toolbar.

Note: The **Go to Shooting Position** feature is only available in **Examiner** (or **Walkthrough**) mode.

Standard Views

RealWorks provides the user with twelve pre-programmed standard viewing positions. Six of them (**Top**, **Bottom**, **Right**, **Left**, **Front** and **Back**) are defined as below where **X**, **Y**, **Z** represent the three axes of the **Active Frame**.

View	View Direction
Top	Looking parallel to - Z-axis, + Y-axis bottom to top, + X-axis left to right
Bottom	Looking parallel to + Z-axis, + Y-axis top to bottom, + X-axis left to right
Front	Looking parallel to + Y-axis, + Z-axis bottom to top, + X-axis left to right
Back	Looking parallel to - Y-axis, + Z-axis bottom to top, + X-axis right to left
Left	Looking parallel to + X-axis, + Z-axis bottom to top, + Y-axis right to left
Right	Looking parallel to - X-axis, + Z-axis bottom to top, + Y-axis left to right

The six others (**Top of the Selected Object**, **Bottom of the Selected Object**, **Right of the Selected Object**, **Left of the Selected Object**, **Front of the Selected Object** and **Back of the Selected Object**) correspond to the top, bottom, right, left, front and back face of an entity which should be selected and of geometry property.

Global Views

To Align to a Global View:

1. In the **3D View** menu, select **Standard Views**. A submenu drops down.
2. Select one of the six options from the submenu.

Note: No selection is required to apply a standard view.

Tips:

- You can use the corresponding icon in the **View Alignment** toolbar.
- You can right-click anywhere in the **3D View** (except on displayed objects) and select **Standard Views** from the pop-up menu. A sub-menu drops down from which you can select a view.

Local Views

To Align to a Local View:

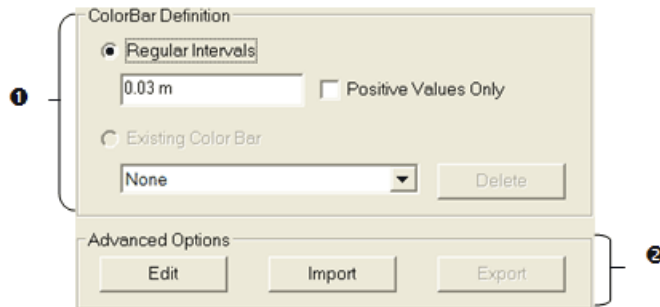
1. Select and display an object with geometry property in the **3D View**.
2. In the **3D View** menu, select **Standard Views**. A submenu drops down.
3. Select one of the six options from the submenu.

Tips:

- You can use the corresponding icon in the **View Alignment** toolbar.
- You can right-click anywhere in the **3D View** (except on displayed objects) and select **Standard Views** from the pop-up menu. A sub-menu drops down from which you can select a view.

ColorBars

A **ColorBar** is a scale of elevation values and each color corresponds to a range of elevation values; it is always linked to an inspection map (or 3D inspection cloud). You can create, edit, delete, rearrange, import or export a color bar inside tools like **Twin Surface Inspection***, **Surface-to-Model Inspection***, **3D Inspection***, etc. after comparing (or inspecting) or outside tools after opening e.g. an inspection map*.



1 - Basic options of the color bar definition tool

2 - Advanced options of the color bar definition tool


Notes:

- * These tools are not present in **RealWorks (Base)** and **Advanced**.
- ** Outside comparison (or inspection) tools, only an inspection map can be opened in an independent window while a 3D inspection cloud cannot.

Creating

You can create as many **ColorBars** as required inside your project but only one can be linked to an inspection map (or 3D inspection cloud) at once. If you do not create a **ColorBar** after creating an inspection map (or 3D inspection cloud), a by-default one will be automatically attached to it. When you save your project, the created color bar(s) will be saved too and you cannot see it (or them) in the **Project Tree**. You should open the **ColorBar** box for this.

To Create a ColorBar:

1. Click **Edit ColorBar** . The **ColorBar** dialog opens.
2. Click **Edit**. The **ColorBar Editing** dialog opens.
3. Enter a name in **Name** field or keep the default one.
4. Define the intervals of the color bar you wish to create.
5. If required check **Positive Values Only**.
6. Choose the color of each interval.
7. Click **Create**. The **ColorBar Editing** dialog closes.
8. Click **OK**. The **ColorBar** dialog closes.


Notes:

- To create a color bar based on an existing one; click on the **Existing ColorBar** pull-down arrow in the **ColorBar** dialog and select one from the drop-down list.
- If two **ColorBars** share the name; the second (in creation order) has the number two between brackets at the end of its name.

Importing


You can use any TXT editor (**WordPad** for example) to create a **ColorBar** file and import it in your current project. This file should contain a series of RGB (Red Green Blue) and interval value pairs. The **ColorBar** once imported will be automatically applied to the opened (selected) inspection map (or 3D inspection cloud).

To Import a ColorBar:

1. Click **Edit ColorBar** . The **ColorBar** dialog opens.
2. Click **Import**. The **Import ColorBar** "Name of the Project" dialog opens.
3. Find a location in your disk where the color bar file is stored in the **Look In** field.
4. Click on the **ColorBar** file name to select it.
5. Click **Open**. The **Import ColorBar** "Name of the Project" dialog closes.
6. Click **OK**. The **ColorBar** dialog closes.

Deleting

To Delete a ColorBar:

1. Click **Edit ColorBar** . The **ColorBar** dialog opens.
2. Check the **Existing ColorBar** option.
3. Drop-down the **Existing ColorBar** list.
4. Select a **ColorBar**.
5. Click **Delete**. If there is more than one **ColorBar** in your project, the one that comes after will be applied to the selected inspection map (or 3D inspection cloud).
6. Click **OK**. The **ColorBar** dialog closes.


Notes:

- It is not necessary to check the **Existing ColorBar** option; clicking **Delete** will by-default select the option and delete the current **ColorBar**.
- It is not necessary to check the **Existing ColorBar** option; drop-down the **Existing ColorBar** list will by-default select the option.

Exporting

A **ColorBar** when exported is a TXT format file. This file, when opened, contains a series of RGB (Red Green Blue) and interval value pairs. You can use any TXT editor (**WordPad** for example) to modify the exported **ColorBar** file or to create a new one. A newly created **ColorBar** file can then be imported into your project.

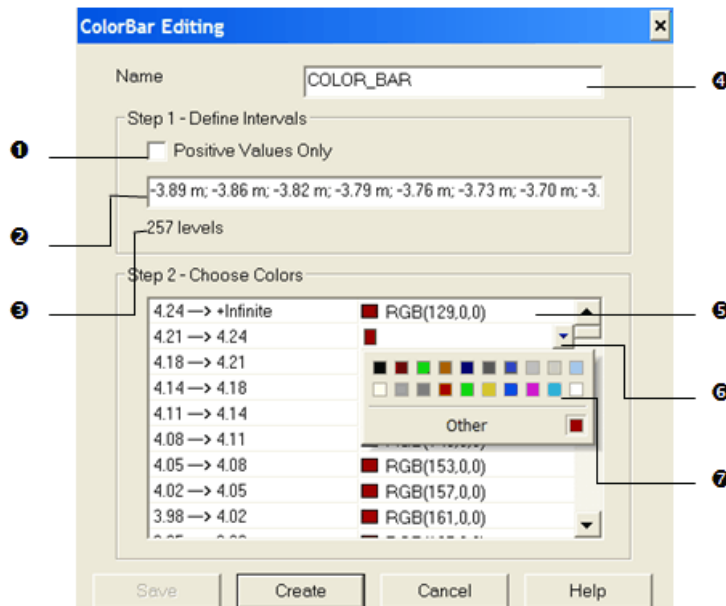
To Export a ColorBar:

1. Click **Edit ColorBar** . The **ColorBar** dialog opens.
2. Check the **Existing ColorBar** option.
3. Or click on the pull down arrow below the **Existing ColorBar** option.
4. Select a **ColorBar** from the drop-down list.
5. Click **Export**. The **Export ColorBar** dialog box opens.
6. Enter a name in the **File Name** field.
7. Find a location in your disk in the **Look In** field.
8. Click **Save**. The **Export ColorBar** dialog box closes.
9. Click **OK**. The **ColorBar** dialog closes.

Editing

To Edit a ColorBar:

1. Click **Create ColorBar** . The **ColorBar** dialog opens.
2. Click **Edit**. The **ColorBar Editing** dialog opens.




- | | |
|--|--|
| <p>1 - Option for setting only positive values of intervals</p> <p>2 - Field for defining intervals by setting values</p> <p>3 - Number of levels in the current ColorBar or in a ColorBar to come</p> | <p>4 - The name of the current ColorBar or of a ColorBar to come</p> <p>5 - Field for defining an interval color</p> <p>6 - Pull down arrow</p> <p>7 - Palette of colors</p> |
|--|--|

3. Click in the interval value field and key in new values. The number of levels is displayed below.
4. Check the **Positive Values Only** option if required.
5. Click in an interval line. A pull-down arrow appears
6. Click on the pull-down arrow. A color palette appears.
7. Choose an existing color or click **Other** to define yours.
8. Click **Save**. The **ColorBar Editing** dialog closes.
9. Click **OK**. The **ColorBar** dialog closes.

Hiding/Opening

You can hide the current **ColorBar** and display it again. Doing this will remove the red-and-green-axis frame and the scale from the "**Inspection Map's Name**" window.

To Hide/Open a ColorBar:

1. First open an inspection map if you are outside the comparison/inspection tools.
2. Click **Hide/Show ColorBar**  to hide the current color bar.
3. Click again **Hide/Show ColorBar** to display the current color bar.

CHAPTER 6

Editing Data

We have introduced a point cloud loading tool for supporting huge amount of points. The user is able to precisely control which points are loaded into memory and thus available for all the regular tools.

Dragging and Dropping an Item

The **Drag & Drop** functionality provides shortcut methods for performing common tasks. You can use the drag and drop functionality to modify the organization of the **Project Tree** according to conditions).

In the Scans Tree:

From:	To:	
	Project	Group
Project	No	No
Group	No	Yes (1)
Station	No	Yes (1)
Scan	No	No
Targets	No	No

In the Models Tree:

From:	To:	
	Project	Group
Project	No	No
Project Cloud	No	No
Group	No	Yes (1)
Object	No	Yes (1)


In the Images Tree:

From:	To:	
	Project	Group
Project	No	No
Group	No	Yes (1)
Images	No	Yes (1)

(1) In a group with the same level or in a sub-level group

To Drag and Drop an Item:

1. Select a group (or object node(s)) that you want to drag and drop.
2. Press and hold the left mouse button while you drag the object to its destination.
3. Release the mouse button to drop the object.

Note: The cursor changes to  when you try to drag and drop an item for which the operation cannot be performed.

Cutting and Pasting an Item

You can use the **Cut/Paste** functionality to delete or re-organize the **Project Tree** according to conditions. It is important to note that you cannot apply these operations to a scan or to an image; and these operations should be used inside a project. Copying a group node will duplicate its contents.

In the Scans Tree:

From:	To:	
	Project	Group
Project	No	No
Group	No	Yes (1)
Station	No	Yes (1)
Scan	No	No
Targets	No	No

In the Models Tree:

From:	To:	
	Project	Group
Project	No	No
Project Cloud	No	No
Group	No	Yes (1)
Object	No	Yes (1)

In the Images Tree:

From:	To:	
	Project	Group

Project	No	No
Group	No	Yes (1)
Images	No	Yes (1)

(1) In a group with the same level or in a sub-level group

To Cut an Item:

1. Select an object (or a group of objects) from the **Models Tree**.
2. In the **Edit** menu, select **Cut** from the menu bar.
3. Navigate through the **Models Tree** to select a new location.
4. In the **Edit** menu, select **Paste**. The selected object (or group of objects) is moved.

Notes:

- You undo or redo the **Cut** operations you have previously performed.
- You can also pick an object directly in the **3D View** and select a command from the pop-up menu.

Tip: You can use the following shortcut-keys instead of selecting commands from the menu bar (**Ctrl + X** for **Cut** and **Ctrl + V** for **Paste**).

Note: The **Cut** command from the **Edit** menu remains grayed out when you select an item for which the cut cannot be performed.

Copying and Pasting an Item

You can use the **Copy/Paste** functionality to delete or re-organize the **Project Tree** according to conditions. It is important to note that you cannot apply this operation to a scan or to an image; and this operation should be used inside a project. Copying a group node will duplicate its contents.

In the Scans Tree:

From:	To:	
	Project	Group
Project	No	No
Group	No	No
Station	No	No
Scan	No	No
Targets	No	No

In the Models Tree:

From:	To:	
	Project	Group
Project	No	No
Project Cloud	No	No
Group	No	Yes (1)
Object	No	Yes (1)

In the Images Tree:

From:	To:	
	Project	Group

Project	No	No
Group	No	No
Images	No	No

(1) In a group with the same level or in a sub-level group

To Copy an Item:

1. Select an object (or a group of objects) from the **Models Tree**.
2. In the **Edit** menu, select **Copy**.
3. Navigate through the **Models Tree** to select a new location.
4. In the **Edit** menu, select **Paste**. The selected object (or group of objects) is duplicated.

A copied object has as name Copy of "Name_Of_The_Object_To_Copy".

Notes:

- You undo or redo the **Copy** operations you have previously performed.
- You can also pick an object directly in the **3D View** and select a command from the pop-up menu.

Tip: You can use the following shortcut-keys instead of selecting commands from the menu bar (**Ctrl + C** for **Copy** and **Ctrl + V** for **Paste**).

Note: The **Copy** command from the **Edit** menu remains grayed out when you select an item for which the copy cannot be performed.

Deleting an Item

This command can be used to delete objects from the **RealWorks** database. It is important to note that you cannot delete a station, a scan or an image in the **Scans Tree**, the unmatched target group in the **Targets Tree** and the **Project Cloud** in the **Models Tree**.

To Delete an Item:

1. Select an object (or a group of objects) from the **Models Tree**.
2. In the **Edit** menu, select **Delete**.
3. Click **OK**.

Note: You can also use the **Del** key instead of selecting the **Delete** command from the menu bar.

Creating a New Group Node

This command allows you to create a new group. You can do this in all four sub-trees.

To Create a New Group Node:

1. Select a project/group object from the **Project Tree**.
2. In the **Edit** menu, select **New Group**. The new group will be created under the selected project/group.

Tip: You can also choose the **New Group** command from the pop-up menu in the **WorkSpace** window.

Changing Color

You can change the color of a selected item in two different ways.

From the Property Window

To Change an Item's Color in the Property Window:

1. Select an item from the **Project Tree**.
2. Right-click to display the pop-up menu.
3. Select **Properties** from the pop-up menu. The **Property** window opens.
4. If the item has only a point cloud representation, click in the **Color of Cloud** field. Its color becomes editable and a pull-down arrow button appears.
5. If the item has only a geometry representation, click in the **Color of Geometry** field. Its color becomes editable and a pull-down arrow button appears.
6. If the item has both representations*, click in either the **Color of Cloud** or **Color of Geometry** field.
7. Click on the pull-down arrow button. A color palette appears.
8. Do one of the following:
 - Choose a color from the color palette.
 - Or define a color:
 - a) Click **Other**. The **Colors** dialog opens.
 - b) Define a color.
 - c) Click **OK**.

Note: (*) Fitted item

From the Menu Bar

To Change an Item's Color from the Menu Bar:

1. Select an object from the **Project Tree**.
2. If the item has only a point cloud representation, click on the **Change Cloud Color** pull-down arrow in the **3D View** toolbar. A color palette appears.
3. If the item has only a geometry representation, click on the **Change Geometry Color** pull-down arrow in the **3D View** toolbar. A color palette appears.
4. If the item has both representations*, click on either the **Change Cloud Color** or **Change Geometry Color** pull-down arrow. A color palette appears.
5. Do one of the following:
 - Choose a color in the color palette.
 - Define a color:
 - a) Click **Other**. The **Colors** dialog opens.
 - b) Define a color.
 - c) Click **OK**.

Note: (*) Fitted item

Changing Name

You can change the name of an object in two places: either in the **WorkSpace/List** window, or in the **Property** window. You can rename all objects except the project node itself, the **Project Cloud** and unmatched targets.

To Rename in the Property Window:

1. Select an object from the **Project Tree** and right-click to display the pop-up menu.
2. Select **Properties** from the pop-up menu. The **Property** window opens.
3. Click in the **Name** field. The selected object name becomes editable.
4. Enter a new name.
5. Press **Enter**.

To Rename in the WorkSpace Window:

1. Select an object from the **Project Tree**.
2. Left-click twice on the name of the found object.
3. Enter a new name.
4. Press **Enter**.

Note: You can also select an object and use the **F2** key to rename it.

Merging Point Clouds

Merging cloud objects consists of creating a new cloud from the selected clouds and deleting them at the same time. It is important to note that you can only merge objects containing only the cloud representation. If one of the selected objects contains a geometry representation, a warning will be issued to user. If user decides to continue, the geometric shape of the selected object will be lost. You cannot merge the scans. The selected clouds for merging must belong to the same project. When you select clouds from two different groups, the merged cloud will be put under the group which contains the last selected cloud.

To Merge Point Clouds:

1. Select cloud objects from the **Project Tree**.
2. From the **Edit** menu, select **Merge Clouds**.

Note: You can also use the following short-cut key **Ctrl + M** instead of selecting **Merge Clouds** from the menu bar.

Picking Items

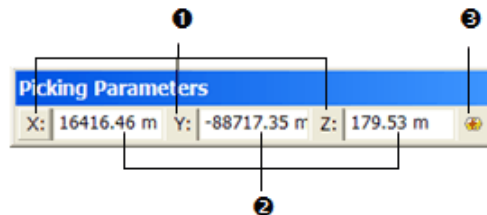
There is no command for opening the **Picking Parameters** toolbar; it comes with tools where pickings are required - like **Polyline Drawing**, **Measurement**, **Geometry Creator***, etc. The **Picking Parameters** toolbar (when appeared) is composed of three fields (X, Y and Z coordinates)** and a button (**Lock on Primitive Center**) in 3D constraint mode and of two fields (**Angle** and **Distance** called **Polar coordinates** or **Distance** and **Distance** called **Cartesian coordinates**) in 2D constraint mode.

Notes:

- (*) This tool is not present in **RealWorks (Base)** and **Advanced**.
- (**) In the **X, Y Z Coordinate System** or **North, East and Elevation** in the **North, East, Elevation Coordinate System**.

Picking in the 3D Constraint Mode

In the 3D constraint mode, you can lock in the active coordinate frame a coordinate, a couple of coordinates, all coordinates at once or a primitive center for constraining pickings. When only one coordinate is locked, the picking is constrained to a plane. When two coordinates are locked, the picking is constrained to a line. And three locked coordinates define the position of a point. All fields are blank before picking.



1 - Clickable buttons 2 - Lock on Primitive Center 3 - Editable fields

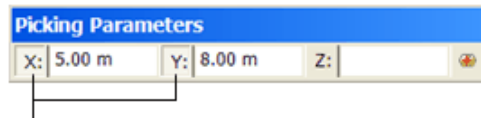
Notes:

- Instead of clicking on the **X/Y/Z** buttons*, you can also use the following short-cut keys **X/Y/Z***.
- The unit of measurement is set by default to Meters; you do not have to enter "m" and you can change it when necessary (see **Preferences**).
- (*) In the **X, Y, Z Coordinate System**.

Constraining Picking on a Plane

To Constrain Picking on a Plane:

1. Enter a coordinate in any of the three fields. Its related button is automatically pressed-on.



X* is locked in this example

2. Pick one point on the displayed object. Picking is locked in the X* coordinate.

Note: (*) In the X, Y, Z Coordinate System.

Constraining Picking on a Line

To Constrain Picking on a Line:

1. Enter a coordinate in any of the three fields. Its related button is automatically pressed-on.
2. Enter another coordinate in any of the two remaining fields. Its related button is automatically pressed-on.



X* and Y* are locked in this example

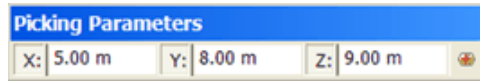
3. Pick one point on the displayed object. Picking is locked in the X and Y coordinates.

Note: (*) In the X, Y, Z Coordinate System.

Constraining Picking on a Point

To Constrain Picking on a Point:

1. Enter a coordinate in each of the three fields. Its related button is automatically pressed-on.



2. Go to the **3D View** and pick one point. Picking is locked in that position.

Locking on Primitive

To Lock on a Primitive Center:

1. Click on **Lock on Primitive Center**
2. Go to the **3D View** and pick on a primitive. Wherever you pick on the primitive, you are locked on its center and its 3D coordinates are displayed in the **X***, **Y*** and **Z*** fields.

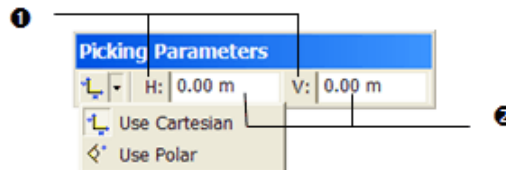
Note: (*) In the **X, Y, Z Coordinate System**.

Picking in the 2D Constraint Mode

In the 2D constraint mode, you can use **Cartesian** (**H** and **V** distances both in mm*) or **Polar** (**Angle** and **Distance** respectively in degrees** and in mm**). The **Cartesian** and **Polar** constraint picking modes come automatically when you have to 2D-pick. In each mode, you can constrain one or both items. To tilt from one constraint mode to the other, click on the **Switch to Polar or Cartesian** button or on the pull-down arrow and chose the constraint mode you need. Note that you can do this at any time before and while picking points.

Cartesian Constraint Mode:

Before you pick a first point, both fields (**H** and **V**) are grayed out. After you pick your first point; this point is assumed as the origin - with 0 and 0 as **H** and **V** coordinates - for the next point to come. This next point is itself assumed as the origin for the third point to come and so on.

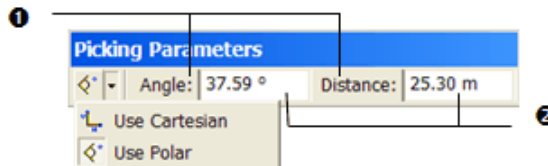


1 - Clickable buttons

2 - Editable fields

Polar Constraint Mode:

Before you pick a first point, both fields (**Angle** and **Distance**) are grayed out. After you pick your first point; the **Angle** and **Distance** fields are empty of value. When you pick the next point; the **Angle** field remains empty of value and the **Distance** field is filled with a value that corresponds to the distance from the first point to this second point. When you try to pick a third point, the **Angle** field is filled with the second point/first point and second point/third point angle value.



1 - Clickable buttons

2 - Editable fields

Notes:

- Instead of clicking on the **H** and **V** buttons for **Cartesian** (or **Angle** and **Distance** buttons for **Polar**); you can also use its related shortcuts key **H** and **V** (or **A** and **D**).
- (*) The unit of measurement for **H** (or **V**) in **Cartesian** is set by-default in Millimeter. You can change it in **Preferences \ Units**.
- (**) The unit of measurement for the **Angle** (or **Distance**) in **Polar** is set by-default in Degree (or Meter). You can change it in **Preferences \ Units**.

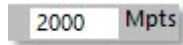
CHAPTER 7

Point Loading Manager

We have introduced a **Point Loading Manager** for supporting huge amount of points. The user is able to precisely control which points are loaded into memory and thus available for all the regular tools.

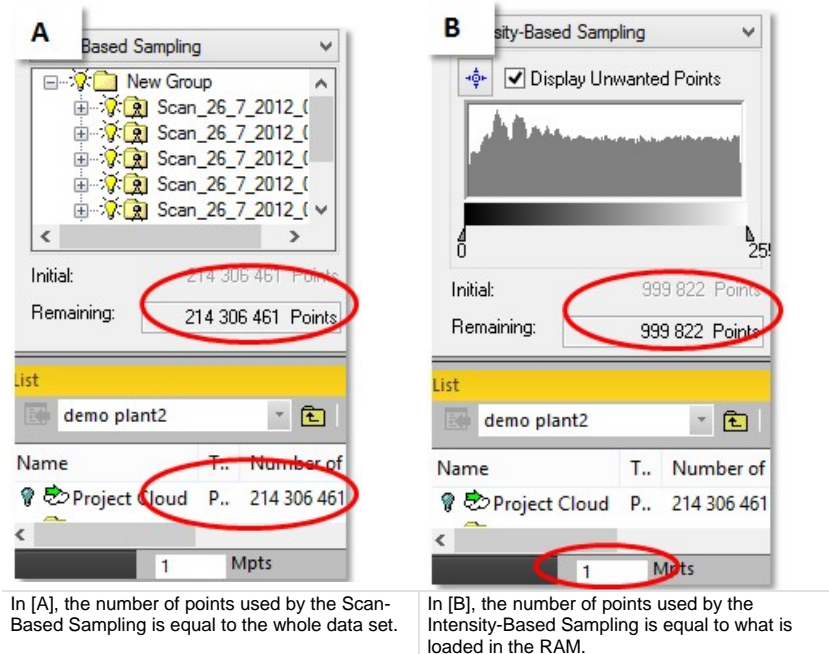
Loading Data

We distinguish two types of data: data loaded on disk and data loaded in RAM. The way the user is able to load (or unload) points in RAM can be done through a field in the status bar. At any time, the user can enter a value between 1 and 2000 (in Millions of Points) in the field and press **Enter**.

A screenshot of a status bar input field. It consists of a rectangular box with a light gray border. Inside the box, the number '2000' is displayed in a dark gray font. To the right of the number, the text 'Mpts' is displayed in a smaller, dark gray font. The entire input field is set against a light gray background.

Processing Data

Some tools can work "On Disk", i.e., on the full data, independently of what is loaded in the RAM (see [A]). Others work directly on the data loaded in the RAM (see [B]).



Here is a list of tools for which the need is to work on the full data, i.e., on disk.

- **Segmentation Tool:** As a stand-alone tool, as a sub-tool in other tools and all tools working in a similar way (**Cloud-based Modeler**, **SteelWorks** and **EasyPipe**).
- **Scan-Based Sampling** and **Random Sampling** methods from the **Sampling Tool** (as a stand-alone tool and as a sub-tool in other tools).
- Exports of point cloud data,
- Generate Point Color-Coding by Height,
- Color Points Using Station Images,
- Coloring in Image Matching Tool.

In all the other tools, the deliverable will be produced with what is loaded in RAM.

Basic Tools

Tools in **RealWorks** can be classified into two categories: base tools and high-level tools. The base tools can be used alone or be open inside the high-level tools to perform basic operations in the three following processing modes - **Registration***, **OfficeSurvey** and **Modeling**** -, i.e., preparing data for high-level tools. In such cases, you cannot save the result. In general, the base tools are represented by toolbars containing the operations arranged as icons or dialogs. Currently, the base tool set includes the **Measurement Tool**, the **Segmentation Tool**, the **Sampling Tool**, the **Frame Creation Tool**, the **Convert to Mesh** tool, etc.

Notes:

- (*) In the **Registration** processing mode, only the **Measurement Tool** is available.
- (**) The **Modeling** processing mode is not present in **RealWorks (Base)** and **Advanced**.

The **Tools** menu is present in all **RealWorks** products but the contents (of tools) differ from a license to another. The table below listed the contents the user can have according to the license (of **RealWorks**) he has.

Tools in the Tools Menu	Viewer	Base	Advanced, Advanced - Modeler, Advanced - Plant
Segmentation	✓	✓	✓
Sampling	✓	✓	✓
Measurement	✓	✓	✓
Frame Creation	✓	✓	✓
Shift Project	✓	✓	✓
Convert to Mesh		✓	✓
Create Merged Mesh		✓	✓
Generate Key Plan From TZF Scans			✓
Generate Key Plan From Current View			✓


Measurement Tool

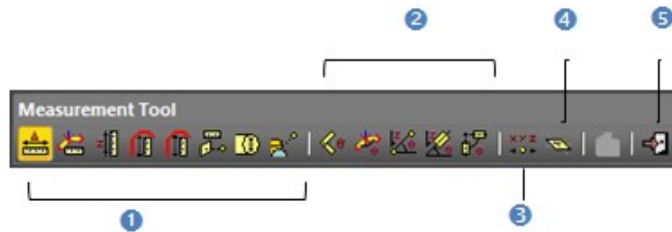
This tool allows you to make point-to-point distance measurements, angular measurements, point-to-scanning position measurements, orientation measurements, etc. You can try as many measurements as you wish and for those you need later on, you can create them as persistent objects in the database. The created measurement objects will be put under the current active group. Measurements are based on pickings which can be free (or constrained).

Opening the Tool

A measurement determines the distance two between picked points, calculates the angle from three picked points, shows the **XYZ** coordinates of a picked point and gives the orientation of a picked point on a sloping surface, etc. It is important to note that picking for the measurement will always function on objects, that is, either on points or on geometric shapes. You can still navigate in the **3D View** while performing a measurement but you cannot select an object.

To Open the Tool:

1. Display an object (point cloud, mesh or geometry) in the **3D View**.
2. From the **Tools** menu, select **Measurement Tool** . The **Measurement Tool** and **Picking Parameters** toolbars appear.

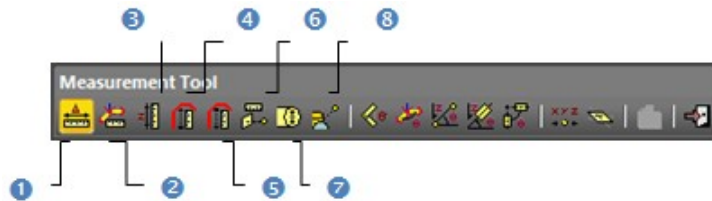


The toolbar is composed of seventeen icons. They are grouped by category: 1 - **Distance Measurement**, 2 - **Angular Measurement**, 3 - **Point Measurement**, 4 - **Slope Measurement**, 5 - **Create** (dimmed) and 6 - **Close**. The measurement type which comes first is the one selected during the last use of that tool. While you are in this tool, an information box will appear on the top right corner of the **3D View**, and the mouse's cursor will change its shape to that of a ruler. When you are on a 3D point, a circle surrounding this point appears at the end of the ruler.

Note: Each type of measurement can be activated via its corresponding icon in the **Measurement Tool** toolbar or by selecting its related command from the pop-up menu.

Distance Measurements


To perform a distance measurement, choose the appropriate type of measurement by clicking on the associated icon. In each case, you should pick two points except for the **Vertical Clearance Measurement (Upward)** or the **Vertical Clearance Measurement (Downward)** where just one point is required.



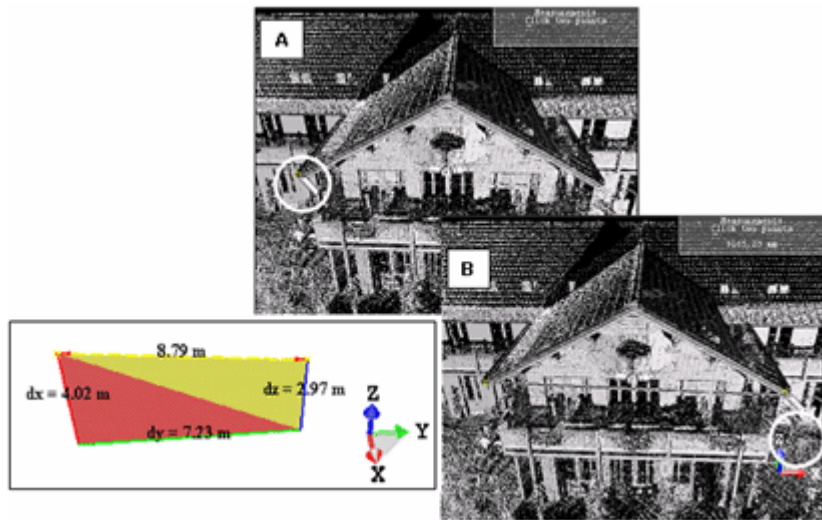
There are eight icons dedicated to distance measurements: 1 - **Distance Measurement**, 2 - **Distance Measurement In Horizontal Plane**, 3 - **Distance Measurement Along Vertical Axis**, 4 - **Vertical Clearance Measurement (Upward)**, 5 - **Vertical Clearance Measurement (Downward)**, 6 - **'Point to Fitted Plane' Distance Measurement**, 7 - **Fitted Cylinder Diameter Measurement**, 8 - **'Point to Geometry' Distance Measurement**.

Measuring a Distance

To Measure a Distance:

1. Click the **Distance Measurement**  icon.
2. Pick one point on the displayed object. This point is the first measurement point (A).
3. Navigate in the **3D View** and pick another point on the displayed object to assign the second measurement point (B).

Once the second point is picked, the distance measurement and its projections along the X, Y, Z axes are displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement, this information box updates automatically the information inside.



Length: Distance from the two picked points

Delta X: Delta distance between the two points along the X axis


Delta Y: Delta distance between the two points along the Y axis

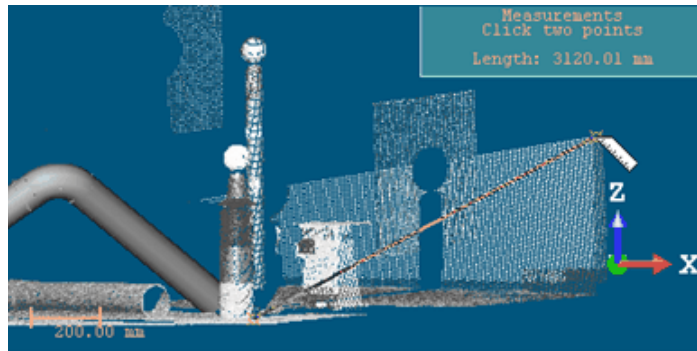
Delta Z: Delta distance between the two points along the Z axis

Note: Press **Esc** (or select another measurement type) to undo the point-to-distance measurement.

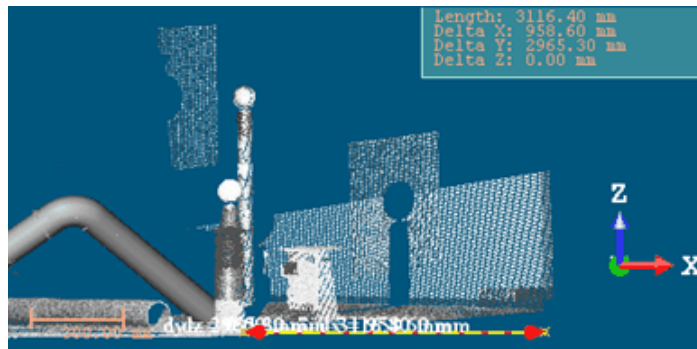
Measuring a Distance in a Horizontal Plane

To Measure a Distance in a Horizontal Plane:

1. Click the **Distance Measurement in Horizontal Plane**  icon.
2. Pick a point on the selected object. This point is the first measurement point (A).



3. Navigate in the **3D View** and pick another point on the selected object to assign the second measurement point (B).



The measurement is performed between point (A) and the projection of point (B) in the **XY** plane. The result and its projections along the X, Y, Z axes are displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement, this information box updates the information inside automatically.

Length: Distance from point (A) to the projection of point (B) in the XY plane

Delta X: Delta distance between the two points along the X axis


Delta Y: Delta distance between the two points along the Y axis

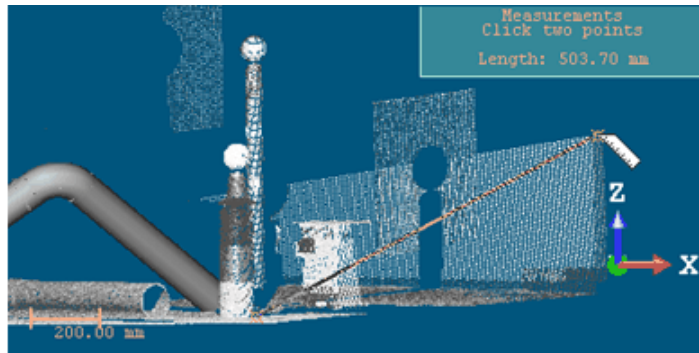
Delta Z: Delta distance between the two points along the Z axis

Note: Press **Esc** (or select another measurement type) to undo the point-to-distance measurement.

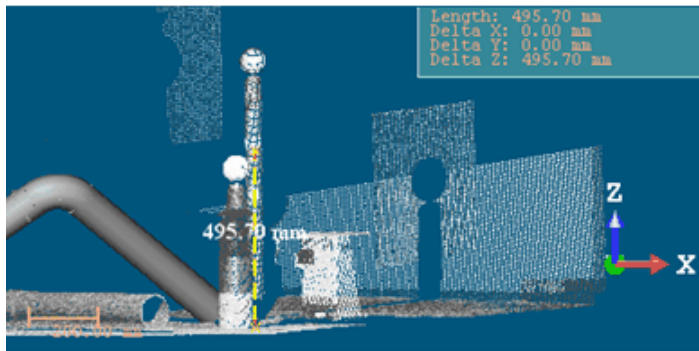
Measuring a Distance Along a Vertical Axis

To Measure a Distance Along a Vertical Axis:

1. Click the **Distance Measurement Along Vertical Axis**  icon.
2. Pick a point on the selected object. This point is the first measurement point (A).



3. Navigate in the **3D View** and pick another point on the selected object to assign the second measurement point (B).



The measurement is performed between point (A) and the projection of point (B) along the Z axis. The result is displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement, this information box updates the information inside automatically.

Length: Distance from point (A) to the projection of point (B) along the Z axis
Delta X: Delta distance between the two points along the X axis

Delta Y: Delta distance between the two points along the Y axis

Delta Z: Delta distance between the two points along the Z axis

Note: Press **Esc** (or select another measurement type) to undo the point-to-distance measurement.

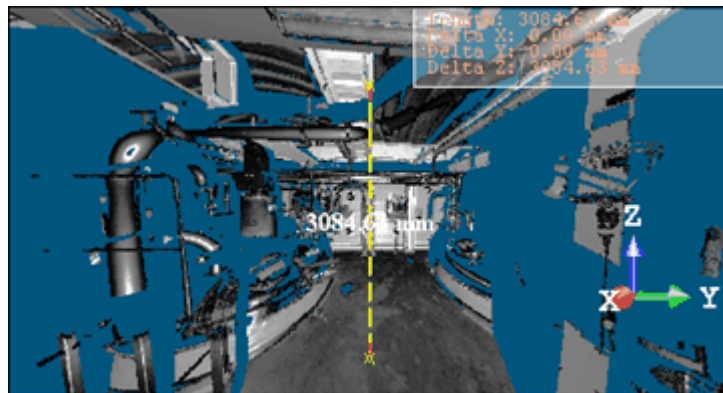
Measuring a Vertical Clearance (Upward)

A **Vertical Clearance** is the minimum unobstructed vertical space between two points along the **Z-Axis**. The **Vertical Clearance Measurement (Upward)** is dedicated to indoor (or outdoor) measurements where the user needs to know the unobstructed distance between two points (from e.g. the ground to the ceiling).

To Measure a Vertical Clearance Distance (Upward):

1. Click the **Vertical Clearance Measurement (Upward)**  icon.
2. Pick a point on the selected object.

The distance measurement is displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement, this information box updates the information inside automatically.



Length: Vertical clearance distance

Delta X: Delta distance between the two points along the X axis

Delta Y: Delta distance between the two points along the Y axis

Delta Z: Delta distance between the two points along the Z axis

Note: Press **Esc** (or select another measurement type) to undo the point-to-distance measurement.

Measuring a Vertical Clearance (Downward)

A **Vertical Clearance** is the minimum unobstructed vertical space between two points along the **Z-Axis**. The **Vertical Clearance Measurement (Downward)** is dedicated to indoor (or outdoor) measurements where the user needs to know the unobstructed distance between two points (from e.g. the ceiling to the ground).

To Measure a Vertical Clearance Distance (Downward):

1. Click the **Vertical Clearance Measurement (Downward)**  icon.
2. Pick a point on the selected object.

The distance measurement is displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement, this information box updates the information inside automatically.

Length: Vertical clearance distance

Delta X: Delta distance between the two points along the X axis

Delta Y: Delta distance between the two points along the Y axis


Delta Z: Delta distance between the two points along the Z axis

Note: Press **Esc** (or select another measurement type) to undo the point-to-distance measurement.

Measuring a Fitted Cylinder Diameter

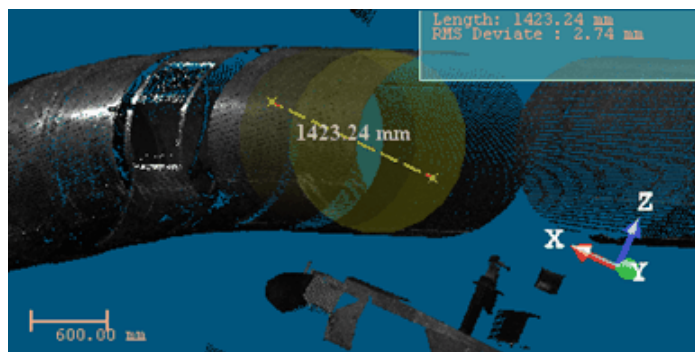
This method enables from a point picked on a set of points to first fit this set of points with a cylinder and then to measure its diameter. This method of measurement cannot be applied to sagging pipes.

To Measure a Fitted Cylinder Diameter:

1. Click on the **Fitted Cylinder Diameter Measurement**  icon.
2. Pick a point on the displayed object.
3. Move your mouse. A sphere whose diameter is formed by the first picked point and the cursor position appears. This sphere is used as bounds for the cylinder fitting.



4. Pick a new point not necessary on the displayed object.




The set of points in the neighborhood of the first picked point is fitted with a cylinder and its diameter is measured and displayed.

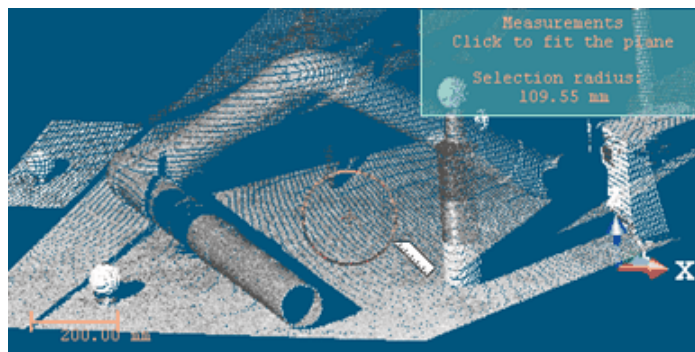
Note: The fitted cylinder will not be created in the **RealWorks** database once the measurement has been validated.

Measuring a Distance to a Fitted Plane

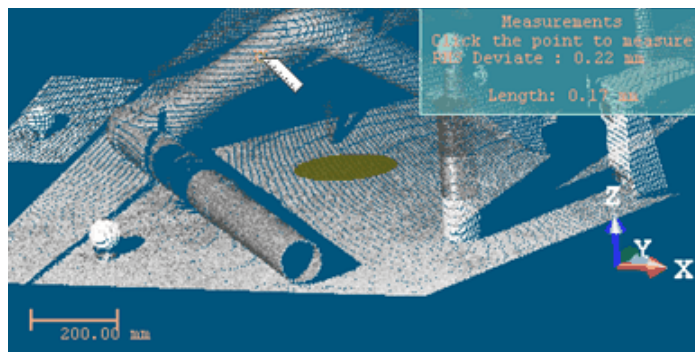
This method enables to measure a distance from three picked points. The two first points are used to fit a set of points with a (circular) plane. The first point which should be picked on a set of points defines its center. The second point (with the first one) defines its diameter. The distance from the third picked point to its projection on the fitted plane is then measured.

To Measure a Distance to a Fitted Plane:

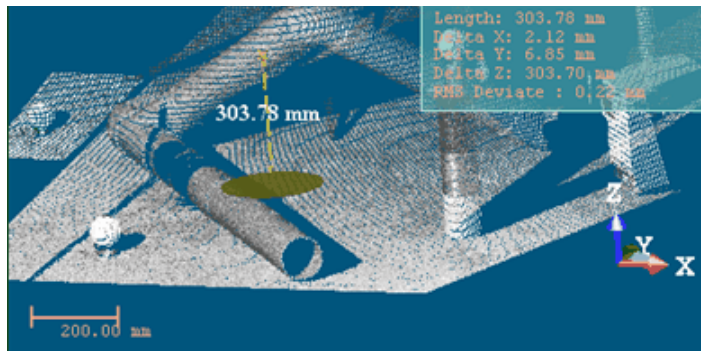
1. Click the '**Point to Fitted Plane**' Distance Measurement  icon.
2. Pick a point on the displayed object.
3. Move your mouse. A sphere whose diameter is formed by the first picked point and the cursor position appears. This sphere is used as bounds for fitting a circular plane.



4. Pick a new point not necessary on the displayed object. A fitted circular plane appears.



5. Pick another new point, now on the displayed object.


**Notes:**

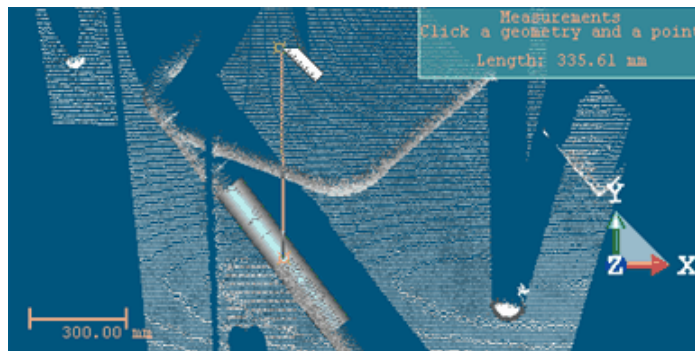
- The fitted (circular) plane will not be created in the RealWorks database once the measurement has been validated.
- The 'Point-to-Fitted Plane' Distance Measurement feature is not present in the toolbar when using the Measurement Tool as a sub-tool in the Cloud-Based Registration Tool.

Measuring a Point-to-Geometry Distance

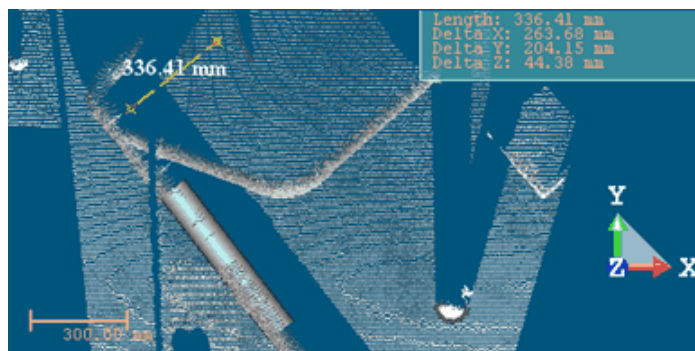
This method measures the shortest distance between a 3D point and a geometry.

To Measure a Point-to-Geometry Distance:

1. Click the 'Point to Geometry' Distance Measurement  icon.
2. Pick a geometry. The cursor takes the shape shown below.
3. Move your cursor. The shortest distance from the picked geometry to your cursor position is displayed in the information box at the top right corner of the 3D View.



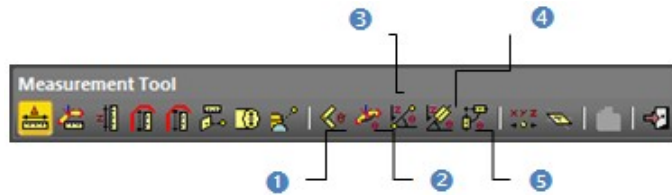
4. Pick a 3D point on the displayed object.



Note: The 'Point to Geometry' Distance Measurement feature is not present in the toolbar when using the Measurement Tool as a sub-tool in the Cloud-Based Registration Tool.

Angular Measurements


To perform an angular measurement, choose the appropriate type of measurement by clicking on the associated icon.

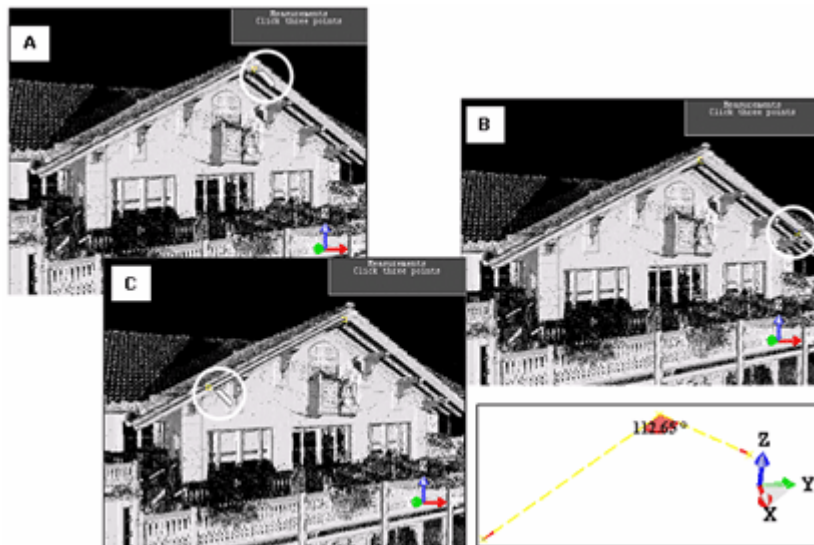


There are five icons dedicated to angular measurements: 1 - Angular Measurement, 2 - Horizontal Angular Measurement, 3 - Slope Angle Measurement, 4 - Geometry Slope Angle Measurement, 5 - 'Angle Between Geometries' Measurement.

Measuring an Angle

To Measure an Angle:

1. Click on the **Angular Measurement**  icon.
2. Pick a point. This point will be the vertex of the angle to measure (A).
3. Navigate through the scene and pick a new point. This point will form with the first point the first segment of the angle to measure (B).
4. Navigate through the scene and pick a new point. This point will form with the first point the second segment of the angle to measure (C).




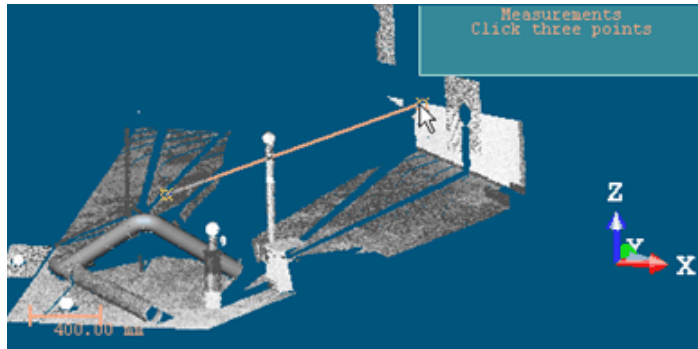
Once the third point is picked, the angular measurement will be displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement this information box will update automatically the information inside.

Note: The three picked points should be on displayed object.

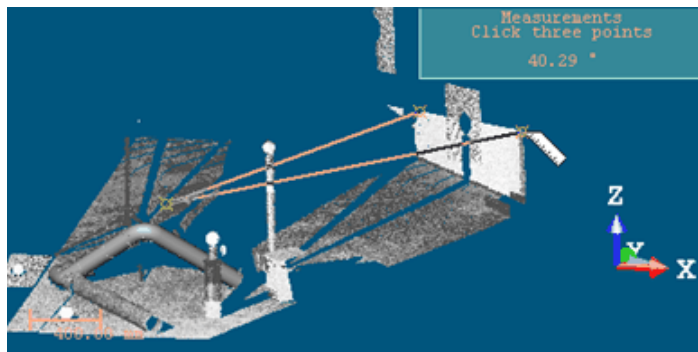
Measuring a Horizontal Angle

To Measure a Horizontal Angle:

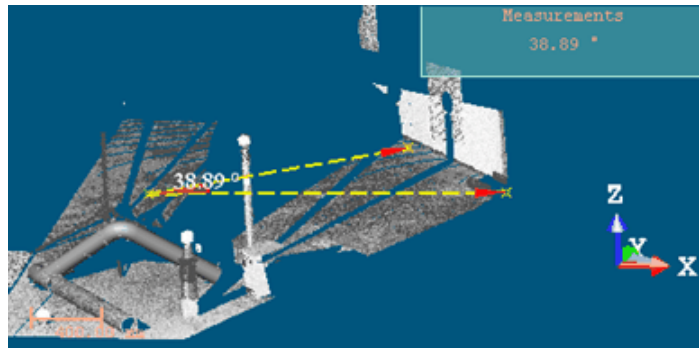
1. Click the **Horizontal Angular Measurement**  icon.
2. Pick a point [A]. This point will be set as the vertex of an angle to measure.



3. Navigate through the scene and pick a new point [B]. This point will form with the first point the first segment of the angle.



4. Navigate through the scene and pick a new point [C]. This point will form with the first point the second segment of the angle.




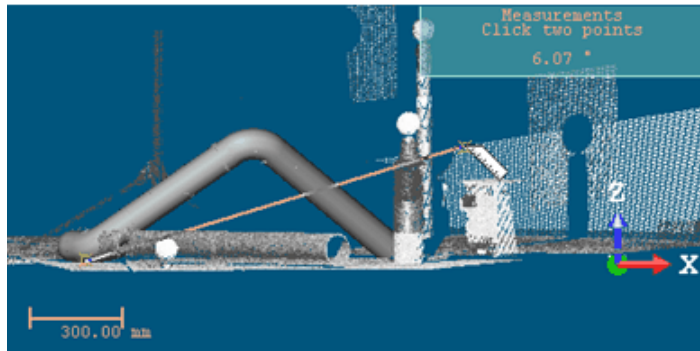
The angular measurement will not be performed between the vertex [A] and points [B] and [C] but between the vertex [A] and the projections of point [B] and point [C] in the XY plane. The result is displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement, this information box updates the information inside automatically.

Note: The three picked points should be on displayed object.

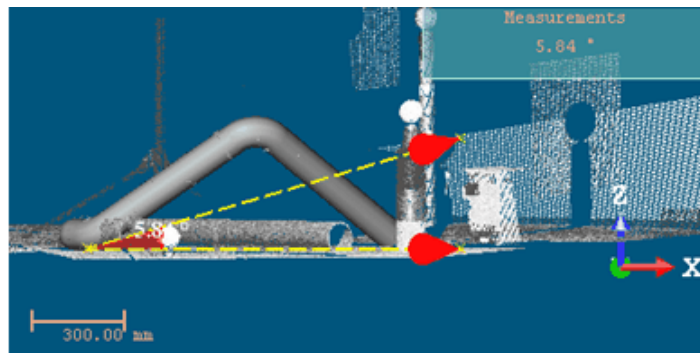
Measuring a Slope Angle

To Measure a Horizontal Angle:

1. Click the **Slope Angular Measurement**  icon.
2. Pick a point [A]. This point will be set as the vertex of the angle to measure.



3. Navigate through the scene and pick a new point [B]. This point will form with the first point the first segment of the angle to measure.

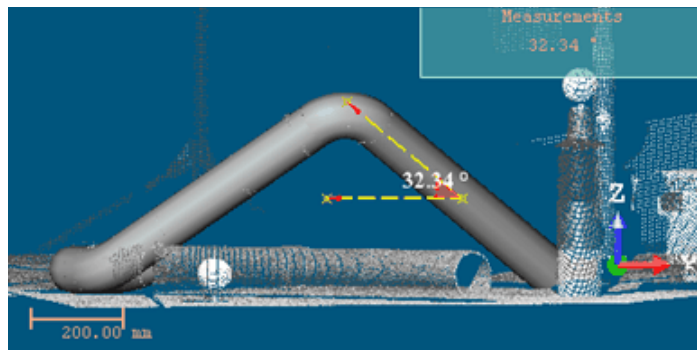
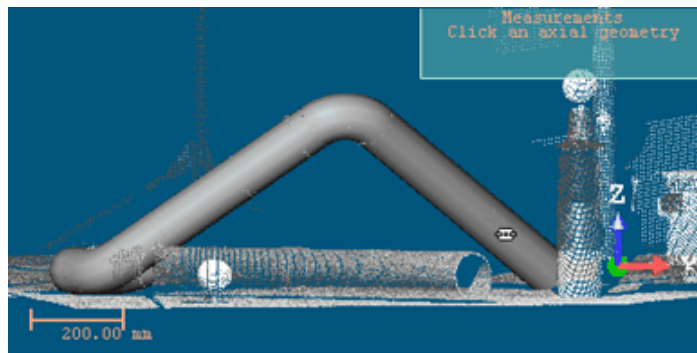


The angular measurement will be performed between the vertex [A], the point [B] and the projection of point [B] in the XY plane. The result is displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement, this information box updates the information inside automatically.

Measuring a Geometry Slope Angle

To Measure a Geometry Slope Angle:

1. Click the **Geometry Slope Angle Measurement**  icon.
2. Pick an axial geometry. Its center will be set as the vertex of the angle to measure. Its axis will be the first segment of the angle to measure.

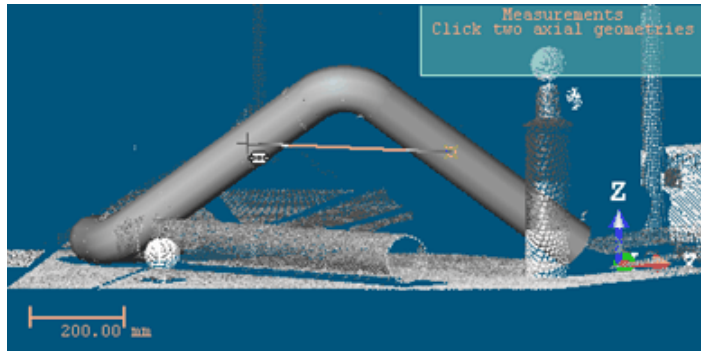


The angular measurement will be performed between the vertex, the geometry's axis and the projection of the geometry's axis in the XY plane. The result is displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement, this information box updates automatically the information inside.

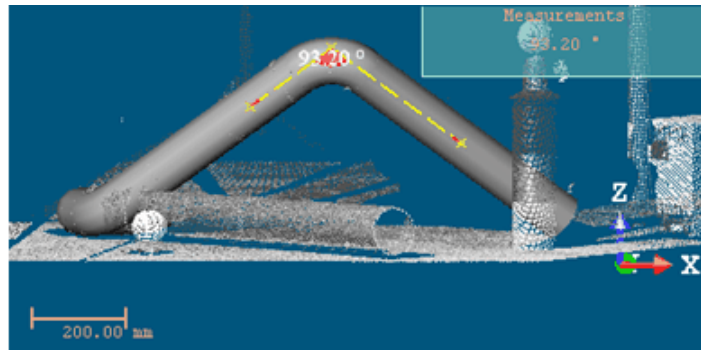
Measuring a Between-Geometry Angle

To Measure a Between-Geometry Angle:

1. Click the '**Angle Between Geometries**' Measurement  icon.
2. Pick an axial geometry.



3. Pick another axial geometry.




The angular measurement will be performed between the two axes of the picked geometries. The result is displayed in the **3D View**. At the same time, the information box will display the measurement result in text. Each time you start a new measurement, this information box updates automatically the information inside.

Note: A warning message appears if the axes (of the geometries) are not secant.

Measuring a Position

For a point-to-scanning position measurement, you need just one point.

To Measure a Position:

1. Click on the **Point Measurement**  icon.
2. Pick one point on the displayed object to measure its 3D position.

A measured point is displayed with a label showing its coordinates. At the same time, the information box will display the measurement in text. Each time you start a new measurement, this information box will update automatically the information inside.


Note: Press **Esc** (or select another measurement type) to undo the point measurement.

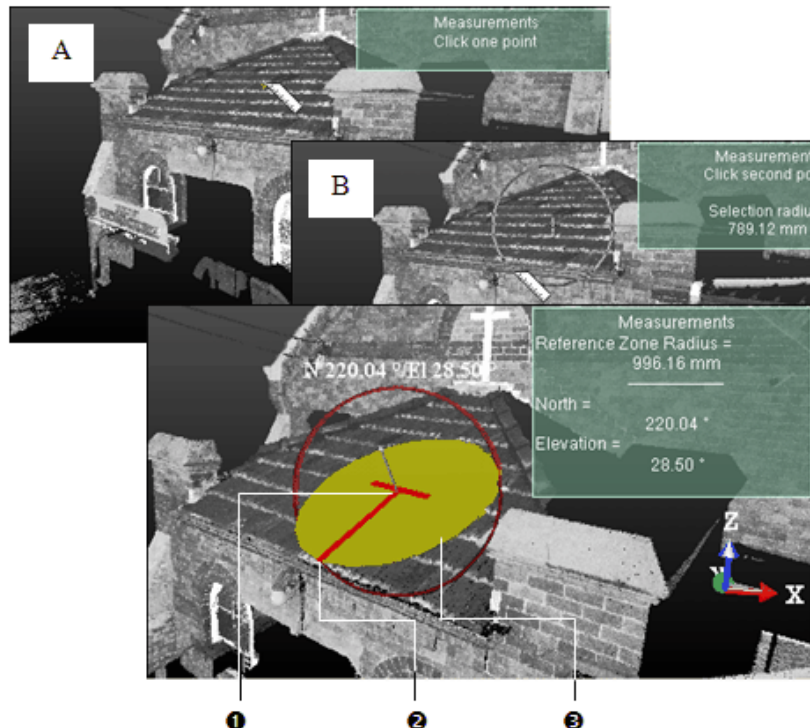
Tip: You can remove a measured point's label by first selecting **Rendering**, then **Display 3D Labels** from the **3D View** menu.

Measuring an Orientation

The **Orientation Measurement** lets you know the orientation of a given point on a sloping surface. An orientation is expressed in the form of two angles. The measurement method is as follows. A plane (of circular shape) is extracted from the point. Two angles are then calculated. The first angle called **Elevation** is formed by the extracted plane and the YX plane of the active coordinate frame. The second angle called **North** is formed by the extracted plane and the ZX plane of the active coordinate frame.

To Measure an Orientation:

1. Click on the **Orientation Measurement**  icon.
2. Pick a point on the displayed object (A).
3. Navigate through the 3D scene. A sphere whose diameter is formed by the first picked point and the cursor position appears. This sphere is used as bounds for the plane extraction.
4. Pick a new point on the displayed object (B).



1 - The first picked point
2 - The second picked point

3 - A plane of circular shape

Once the second point is picked, the orientation measurement result will be shown in the **3D View**. At the same time, the information box will display the measurement results in text. Each time you start a new measurement this information box will update automatically the information inside.

Tip: Press **Esc** (or select another measurement type) to undo the point measurement.

Notes:

- Because measurements are based on point picking, you cannot perform an orientation measurement on an object of geometry type.
- You can reverse the orientation of a measurement. To do this, right-click anywhere in the **3D View** to display the pop-up menu and select **Reverse Orientation Measurement**.

Tip: You can switch the orientation measurement's notation from **North/Elevation** to **Elevation/North** and vice versa in the **Preferences** dialog.

Note: The **Orientation Measurement** feature is not present in the toolbar when using the **Measurement Tool** as a sub-tool in the **Cloud-Based Registration Tool**.

Refining a Measurement

You can refine the measurement you have just performed by modifying the picked points except for the point-to-scanning position measurement. For the orientation measurement, you can enlarge/reduce the sphere diameter (or move its center). For the point-to-point distance measurement, you can move each end in order to extend (or shorten) its length. For the angular measurement, you can move each of the ends to change its angle, etc.

To Refine a Measurement:


1. Place the mouse cursor upon an already picked point.
2. Drag and drop it to a new location on the displayed object.

Tip: Before starting a measurement, press **Esc** (or click **Close Tool** in the toolbar) to close the **Measurement Tool**. When a measurement is in progress, press **Esc** to cancel it and start a new one.

Saving a Measurement

You can save the measurement you have just performed as a persistent object in the **RealWorks** database. For each saved measurement, a geometric object is created and put under the active group in the **Models Tree**. You can save as many measurements as you need without leaving this tool. You can also export a measurement result as a report in **Excel** format (*.CSV files).

To Save a Measurement:

1. Click **Create** . The measurement is saved in the database.
2. Start a new measurement (if required).
3. Click **Close Tool**.

Tips:

- Press **Esc** (or select **Close Tool** from the pop-up menu) to leave the tool.
- Press **Enter** (or select **Create** from the pop-up menu) to save the result.

Note: You should close first the **Measurement Tool** before exporting a result in **Excel** file format. Otherwise, the **Exportation Measurements** command is deactivated.


Segmentation Tool

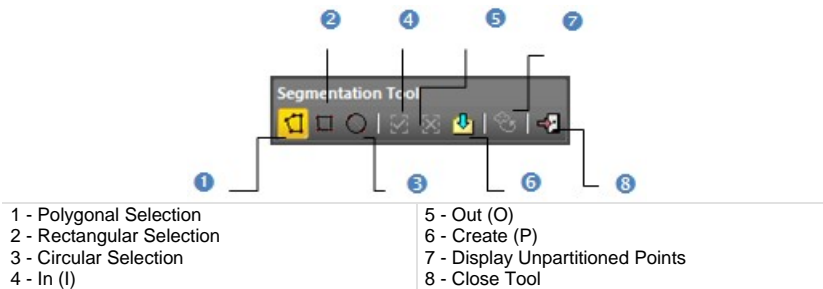
This tool allows you to segment a point cloud object into several point sub-clouds. By using this tool, you can structure a complex scene into its logical component parts, and work subsequently on each part. It is important to note that an object containing both the point cloud and geometry representations cannot be segmented. In order to do this, you have to use the **Sampling Tool** to create a new point cloud without geometry and then perform the segmentation on the newly created point cloud. In order to enable this tool, you should select one or several point cloud objects.

Note: The **Segmentation Tool**, as a standalone version, is only available in **OfficeSurvey** and **Modeling**. When using it as a sub-tool, it is available anywhere.

Opening the Tool

To Open the Tool:

1. Select one point cloud (or more*) from the **Project Tree**.
2. In the **Tools** menu, select **Segmentation Tool** . The **Segmentation Tool** toolbar appears.



An information box appears at the top right corner of the **3D View** and displays the sum total of points included in the selected object. The mouse cursor shape changes; the arrow becomes a pointer. You can still navigate (**Zoom**, **Pan** and **Rotation**) through the 3D scene, select or hide objects (or groups) or change the active group before you start drawing a fence. Once you start drawing, these commands become unavailable except **Pan**. Note that all displayed objects are hidden except those selected. After opening the tool, the default selection tool is **Polygonal Selection**.

Caution:

- (*) You can select several point clouds as input of the tool but one of them should not be the **Project Cloud**.
- You can select the **Project Cloud** alone as input of the tool.

Drawing a Fence

A fence may have three shapes (**Polygonal**, **Rectangular** and **Circular**) and is composed of segments/circular arcs and vertices. A fence is used as segmentation boundaries and drawing one is done by picking the **3D View**. Once the first vertex of a fence is picked, you can no longer move the scene. After drawing a fence, you can rotate the scene to do the selection from another point of view. Such combination allows you to do a 3D point selection (like to segment the cloud with a 3D polyhedron which is the intersection of the extrusion of these 2D fenced polygons).


Tip: You can use the **Esc** key to cancel the fence that you are drawing.

Notes:

- If you select **Close Tool** (or **New Fence** from the pop-up menu) while you have not finished drawing, your fence will be cancelled.
- To start a new fence, you need to cancel the current one by selecting **New Fence** from the pop-up menu.
- Even if the pop-up menu is displayed, the fence still snaps to the mouse cursor. You have to terminate the fence to free it. When selecting **In** (or **Out**), **RealWorks** will close automatically your fence and points inside (or outside) this fence will be kept.

Polygonal Selection

To Draw a Polygonal Fence:


1. Click the **Polygonal Selection**  icon.
2. Click anywhere to draw the first vertex of a **Polygonal Fence**.
3. Click anywhere to draw the second vertex. The two vertices are linked by a segment.
4. Continue to define other vertices. The **Polygonal Fence** is always closed in such a way that the start vertex is always linked to the last one.
5. Right-click in the **3D View** to display the pop-up menu.
6. Select **End Fence** to terminate the **Polygonal Fence**.

Note: To end a fence, you can double-click (or press on the **Space Bar**).

Tip: You can select **Polygonal Selection** from the pop-up menu.

Rectangular Selection


To Draw a Rectangular Fence:

1. Click the **Rectangular Selection**  icon.
2. Click anywhere to draw the first corner of a **Rectangular Fence**.
3. Click anywhere to draw the second and opposite corner. The **Rectangular Fence** is drawn.

Tip: You can select **Rectangular Selection** from the pop-up menu.

Circular Selection

To Draw a Circular Fence:




1. Click the **Circular Selection**  icon.
2. Pick a point to start the first point of a **Circular Fence**'s diameter.
3. Pick another point to set the second point of the diameter.

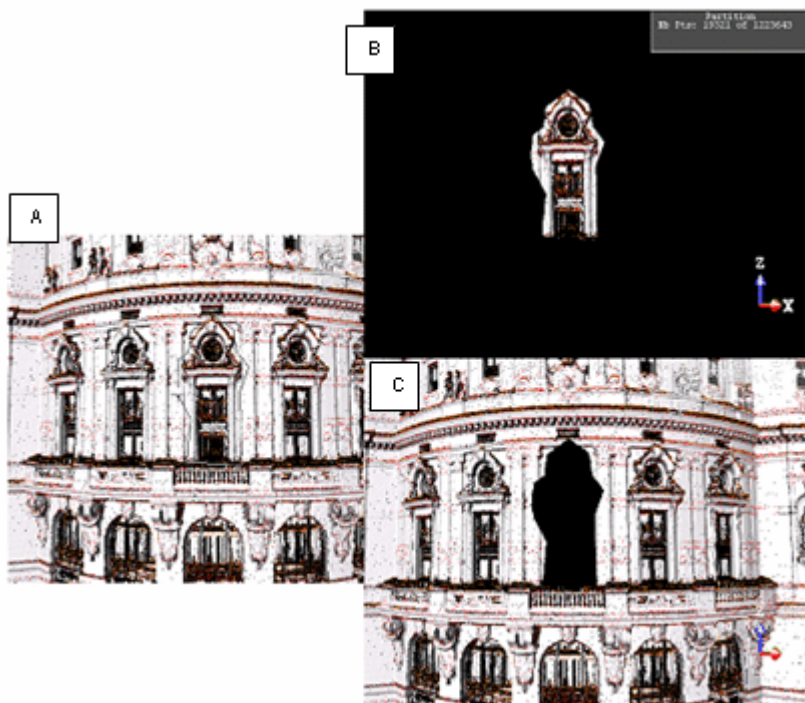
Tip: You can select **Circular Selection** from the pop-up menu.

Keeping In (or Out)

Once you have finished defining a fence, you can now segment the selected point cloud(s) by keeping either the points inside or outside of the fence. Note that any segmented cloud is not permanently created in the database. You should use the **Create** command to perform this operation. This also means that you can turn around the so-segmented cloud, and continue to perform fencing and segmentation.

To Keep In (or Out):

1. Click the **In**  icon. Points inside the fence are kept. Points outside the fence are unkept. This doesn't mean that they are not deleted from the initial cloud but just hidden in the **3D View**. The number of points inside the fence is shown in the information box. You can also use the short-cut key **I** to do this. If the fence does not contain any points, selecting **In** will not take any points into account.
2. Or click the **Out**  icon. Points outside the fence are kept while those that are inside are unkept. The number of remaining points is shown in the information box. You can also use the short-cut key **O** to do this. If the fence does not contain any points, selecting **Out** will not take any points into account.
3. Click **Display Un-partitioned Points**  if you want to work with the same cloud.



A - Polygonal fence

B - Points inside the polygonal fence are kept

C - Points outside the polygonal fence are kept

Tip: You can also select **In** (or **Out**) from the pop-up menu.


Creating the Segmentation Results

Create enables to create a new cloud and puts it in the current group behind other objects. **Create In** creates a new cloud and put it under the model group you have to choose from the drop-down pop-up submenu. The **Create In** feature can only be selected from the pop-up menu. It only appears if there is at least a group in the **Models Tree**.

To Create the Segmentation Result:

1. Right-click anywhere in the **3D View** to display the pop-up menu.
2. Select one of the two available creation modes (**Create** or **Create In**).
3. Select **Close Tool**.

- A cloud named **OBJECTX** is created in the **Models Tree**.

Tip: Instead of selecting **Create**  from the pop-up menu, you can also press the **P** key on your keyboard.

Notes:

- Selecting **Create** (or **Create In**) without closing the fence will close it automatically and a new cloud object will be created from all points inside (or outside) this fence.
- The **Create** feature is always enabled after entering the tool even if there is no drawn fence. This way, you can create a new cloud. It is based on all points of the selected cloud.

Caution: You cannot save your result(s) when using the **Segmentation Tool** as a sub-tool. The **Create** button is dimmed.


Note: A dialog appears and asks if you really want to close the tool without saving the result in the database (after a fencing only).

Sampling Tool

The **Sampling Tool** enables to create a sub-point cloud from a selected point cloud. There are at all six methods: **Spatial Sampling**, **Random Sampling**, **Scan-Based Sampling**, **Intensity-Based Sampling**, **Discontinuity-Based Sampling** and **Topography-Based Sampling**. The initial point cloud remains unchanged after sampling. You can combine these six different methods to sample the selected point cloud, that is; you can use the result from one method as the input to another method and continue until you are satisfied with the result. Or you can create several sub-point clouds within an opened session.

Opening the Tool

To Open the Tool:

1. Select an object (like a point cloud) from the **Project Tree**.
2. From the **Tools** menu, select **Sampling Tool** . The **Sampling** dialog opens.
 - The sampling method which appears first is the one that you have selected during the last opened session.
 - The **Spatial Sampling**, **Intensity-Based Sampling**, **Discontinuity-Based Sampling** and **Topography-based Sampling** methods work on what is loaded only in the RAM.
 - The **Scan-Based Sampling** and **Random Sampling** methods work on disk', i.e., on the full data, independently of what is loaded in the RAM.

Tip: To leave the **Sampling Tool**, you can select **Close** from the pop-up menu.

Caution: You are prevented from changing the number of loaded points inside the **Sampling Tool**. The **Point Loading Manager** is grayed out.

Sampling Methods

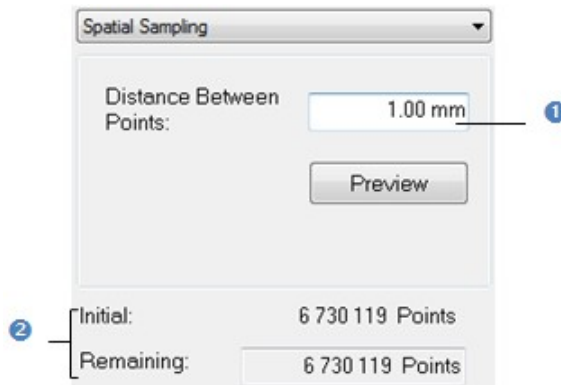
Inside each sampling method, you can use the **Segmentation Tool** to select a data subset for performing a sampling. When used in such condition, you cannot save the result. The **Create** command is deactivated.

Spatial Sampling

Point clouds obtained by scanning from different positions and at different distances are often not uniform in terms of point density. This method enables to obtain a point cloud with a homogeneous density (that the user has to define).

To Sample Spatially:

1. In the **Sampling** dialog, click the pull down arrow.
2. Choose **Spatial Sampling** from the drop-down list. The **Spatial Sampling** dialog appears.



1 - Density of points expressed in terms of a distance between points

2 - Non editable fields

This dialog displays two numbers: **Initial** and **Remaining**. The **Initial** number is the total number of points before sampling. The **Remaining** number corresponds to the number of points after sampling. The unit of measurement is set by default in millimeters; but you can change it when necessary in **Preferences**.

3. Enter a value in the **Distance Between Points** field.
4. Click **Preview** to view the result before saving it.
5. Click **Create** and **Close**.

A sub-point cloud whose name is "**Sample - "Distance Between Points" Value**" is created under the current project in the **Models Tree**.

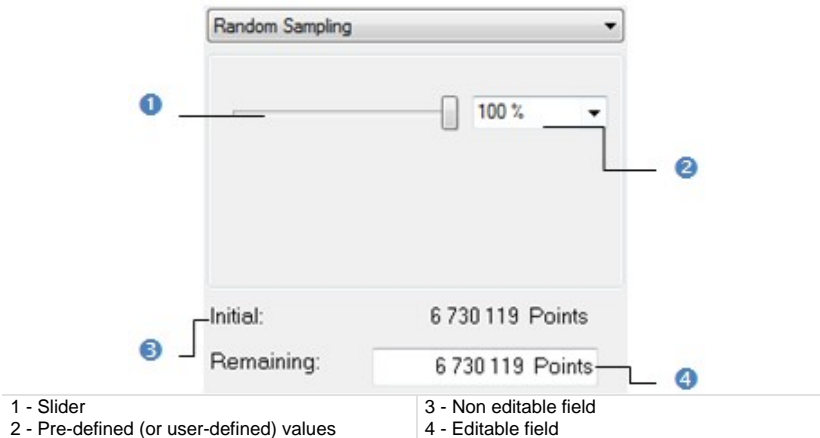
Note: A negative value input the **Distance Between Points** field is automatically converted to positive.

Random Sampling

This method consists in sampling a point cloud by using a percentage ratio defined by the user, which will determine the amount of points that will be kept in the initial point cloud. These points will be randomly selected from the original point cloud.

To Sample Randomly:

1. In the **Sampling** dialog box, click the pull down arrow.
2. Select **Random Sampling**. The **Random Sampling** dialog appears.



In this method, there is no **Preview**. There are four ways to define the percentage ratio. The first is to use a slider. The second is to select a predetermined value among those pre-defined (**25**, **50**, **75** and **100%**). The third is to enter a rate value manually and the fourth is to enter the number of points in the **Remaining** field. Each time you define a new ratio, the sampling will be performed dynamically, and the results (the number of points and the final cloud - respectively in the dialog and in the **3D View**) are displayed in real time.

3. Define a percentage ratio.
4. Click **Create** to save the result.
5. Click **Close**.

A sub-point cloud whose name is "**Sample - "Rate in Percent" Value**" is created under the current project in the **Models Tree**.

Note: When entering a value in the **Rate** or **Remaining** field, do not forget to press **Enter**.

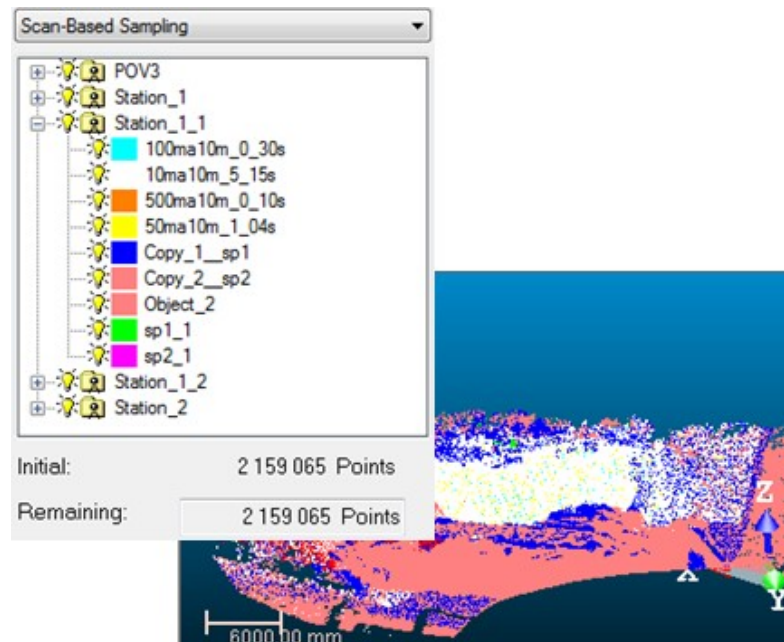
Scan-Based Sampling

On certain occasions, you need to create a sub-point cloud including all points belonging to certain stations or scans. You can use this method to achieve this. After selecting the method, the selected point cloud will be automatically rendered according to the **Scan Color**.

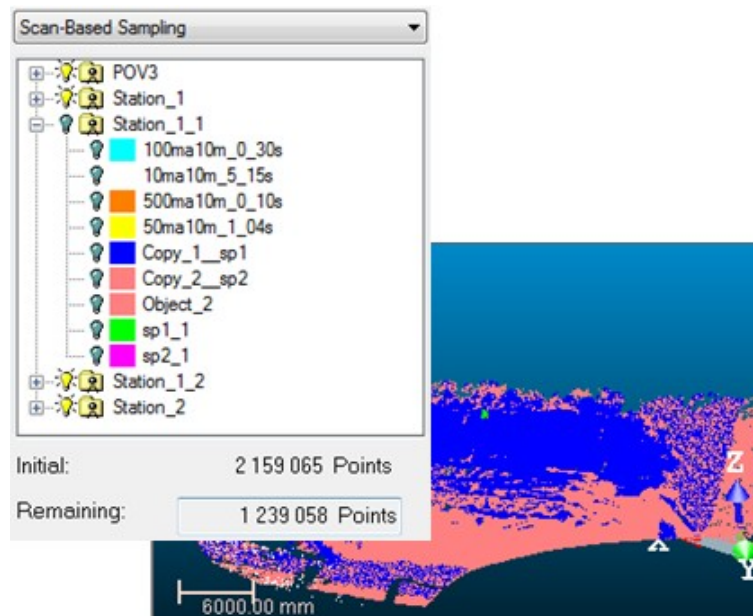
To Sample Based on Scan(s):

1. In the **Sampling** dialog, click the pull down arrow.
2. Select **Scan-Based Sampling**. The **Scan-Based Sampling** dialog appears.

A sub-window including the **Scans Tree** appears in the dialog. By default, all elements (stations and scans) in this tree are selected (**On**). You can un-select (**Off**) a station, a scan or a set of scans from this tree. The total number of points will be changed accordingly and the point cloud displayed in the **3D View** will be updated.



"On" items are shown in the **3D View** and will be considered in the final result



"Off" items (white points) are hidden in the **3D View** and won't be taken into account in the final result

3. Turn **Off** some of the stations (or scans).
4. Click **Create** to save the result.
5. Click **Close** to leave the tool.

A sub-point cloud whose name is "**By Scan (X)**" is created under the current project in the **Models Tree**. **X** is its order.

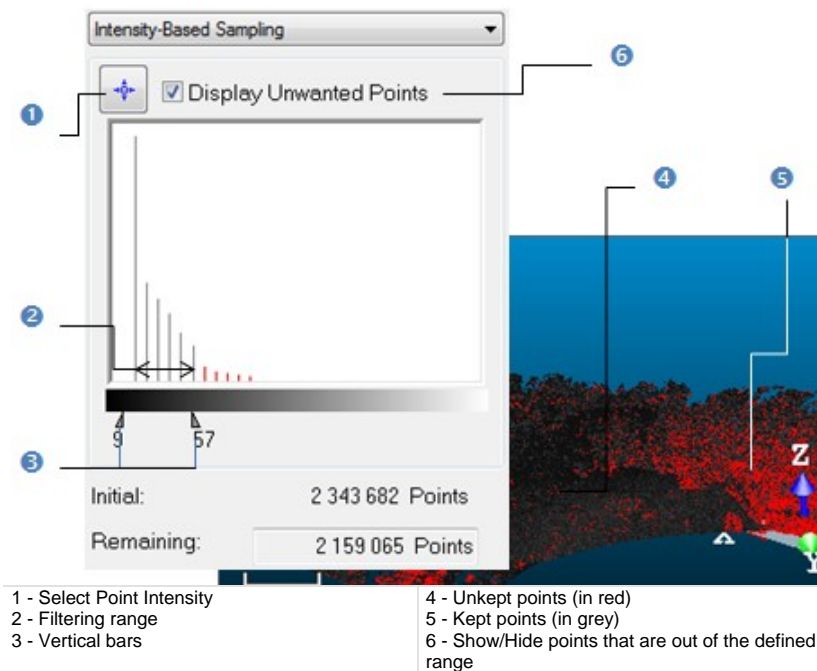
Tip: You can select several stations (or scans or a mix of them) by using the **Ctrl** (or **Shift**) key combined with the left clicking.

Intensity-Based Sampling

This method can be used for sampling the selected point cloud according to the intensity associated with each point. In **RealWorks**, the intensity value ranges from 0 to 255. After selecting the **Intensity Based Sampling** method, the selected point cloud will be rendered in **Gray Scale Intensity**.

To Sample Based on Intensity(ies):

1. In the **Sampling** dialog, click the pull down arrow.
2. Select **Intensity-Based Sampling**. The **Intensity-Based Sampling** dialog appears.



A histogram window appears in the dialog. You can then use the two vertical bars, which are shown as arrows at the bottom of the histogram in the figure above, to define an intensity range so that all points with intensity within this range will be kept. All points outside of this intensity range (called **Unwanted Points**) will be un-kept and shown in red in the histogram and in the **3D View** (only if the **Display Unwanted Points** option is checked). To manipulate the two vertical bars, you can either directly use the cursor to move them, or use the picking mechanism (button on top of the histogram) to select an intensity level from the displayed points.

3. Do one of the following:

- Sample according to point intensity.
 - a) Click **Select Point Intensity**.
 - b) Pick one point in the **3D View**.
- Sample according to a range of intensity.
 - a) Place the mouse cursor over a vertical bar.
 - b) Drag and drop the vertical bar when the intensity value you need is reached.
 - c) Do the same operations for the other vertical bar.

4. Click **Create** and **Close**.

A sub-point cloud whose name is "**By Intensity (X)**" is created under the current project in the **Models Tree**. **X** is its order.

Notes:

- If the selected cloud contains points with no intensity information, these points will not be taken into account.
- If you switch from **Gray Scaled Intensity** to **Color Code Intensity**; the selected **Point Cloud** is then rendered with a range of colors - from red to blue with intermediate colors like orange, yellow and green. Points with intensity of **0** are rendered in red; those with intensity of **128** are in yellow and those with **255** are in blue.

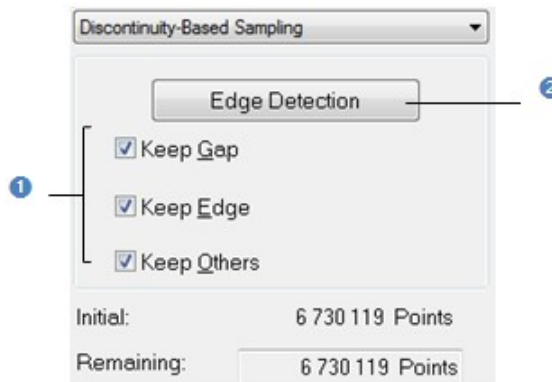
Discontinuity-Based Sampling

You may encounter discontinuities in a point cloud in three cases. First is when some points of the cloud have opposite or different normal direction than the rest. We call this discontinuity **Edge**. Second is when all the cloud points have the same normal direction but some of them are separated by irregular distance. We call this discontinuity **Gap**. The third case can be everything except **Edges** and **Gaps**.

When loading a file of **ASCII** format, **Edges** and **Gaps** are un-generated. You can use the **Edge Detection Tool** to generate them. After selecting the **Discontinuity-Based Sampling** method, the selected point cloud will be automatically rendered in **Discontinuity Display**.

To Sample Based on Discontinuities:

1. In the **Sampling** dialog box, click the pull down arrow.
2. Select **Discontinuity-Based Sampling**. The **Discontinuity-Based Sampling** dialog appears.



1 - Display/Hide discontinuity options

2 - Launch the Edge Detection Tool

When selecting this method, the three discontinuity options are all checked. The **Keep Edge** option (when unchecked) enables the removal of **Edge** discontinuities from the point cloud. The **Keep Gap** option (when unchecked) enables the removal of **Gap** discontinuities from the point cloud. The **Keep Others** option (when unchecked) enables removal of all discontinuities except **Edges** and **Gaps**.

3. If required, detect edges.
4. Click **Create** and **Close**. A sub-point cloud will be created in the database.

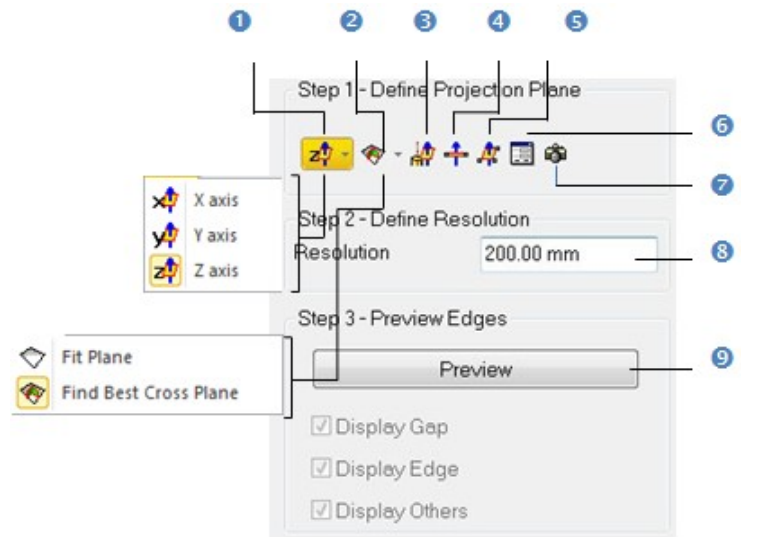
A sub-point cloud whose name is "By Discontinuity (X)" is created under the current project in the **Models Tree**. X is its order.

Detecting Edges

The **Edge Detection** is based on a grid method. You should first define a projection surface (mainly a plane) and then set its grid resolution. The grid resolution is square - the same in both of the projection plane directions (length and width).

To Detect Edges:

1. Click **Edge Detection**. The **Edge Detection** dialog opens.



- | | |
|-----------------------------------|-----------------------------------|
| 1 - Set From Frame | 6 - Edit Parameters |
| 2 - Fit | 7 - Plane Parallel to Screen View |
| 3 - Pick Axis From Object | 8 - Projection Plane Resolution |
| 4 - Plane Perpendicular to Screen | 9 - Discontinuity display options |
| 5 - Pick 3 Points on Plane | |

2. Do one of the following to define a projection plane:

- Select a frame's axis (1).
- Fit an extracted set of points with a plane (1).
- Find a perpendicular view plane from an extracted set of points (1).
- Pick an object's axis (1)(2).
- Pick a plane perpendicular to the screen (1)(2).
- Pick three points (1)(2).
- Edit the project plane's parameters.
 - a) Click **Edit Parameters**. The **3D Plane Editing** dialog opens.
 - b) Click on the pull down arrow.

- c) Choose between **Normal + Point** and **Point + Point**.
 - d) If **Normal + Point** has been chosen, enter a direction in the **Normal** field and give a position in the **Point** field.
 - e) If **Point + Point** has been chosen, enter a position in each of the **Point** fields.
 - f) Click **OK**. The **3D Plane Editing** dialog closes.
- Set the plane parallel to the screen view.
3. Enter a value in the **Resolution** field.
 4. Click **Preview**.
 5. Click **Apply**.

Applying the results after previewing them creates them in the database and closes the **Edge Detection** dialog. Note that clicking **Apply** without previewing the results cancels the generated discontinuities and clicking **Cancel** opens an information box which prompts you to cancel or confirm the action you attempt to do.

Notes:

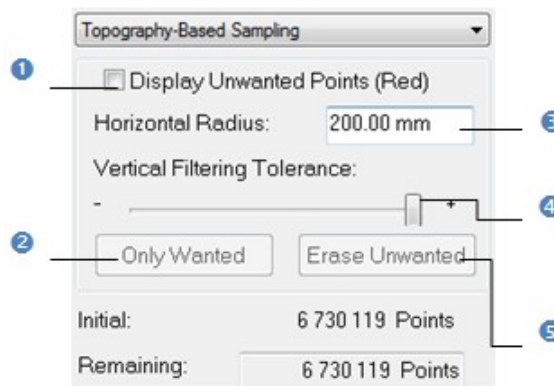
- For more information related to (1), see **Step 2** of the **Cutting Plane Tool**.
- When selecting (2), the **Picking Parameters** toolbar appears, it's up to you to do a free picking or a constrained picking.

Topography-Based Sampling

The idea behind this method is to separate valid points from invalid points inside a given point cloud. Because invalid points are more or less important according to where they are on the point cloud, you need to be able to work separately on them. This kind of situation occurs when the point cloud is a scene presentation with trees, bushes and the like. In such a case, invalid points are trees, bushes, etc. and valid points are the ground.

To Sample Based on Topography(ies):

1. In the **Sampling** dialog, click the pull down arrow.
2. Select **Topography-Based Sampling**. The **Topography-Based Sampling** dialog appears.




- | | |
|--|--|
| 1 - Show/hide invalid points | 4 - Point distance selection |
| 2 - Set valid points as reference cloud | 5 - Delete invalid points from the reference cloud |
| 3 - Resolution of the square-grid method | |

First, you should define a point cloud as reference. This means that this point cloud remains unchanged whatever the operations you apply to it except when deleting points or when replacing it by a new one. You can reload it as often as required. If the point cloud you select comes from another sampling method or results from a segmentation, the **Only Wanted** button is active. This means that you can set this point cloud as a reference.

The **Topography-Based Sampling** method is based on a grid method and the resolution is square by default - the same in both of the projection surface (XY plane of the active coordinate frame) directions. Points of the selected cloud outside the square-grid tolerance will be not taken into account. And those nearby of faraway from the square-grid boundary can be gradually ignored using a slider.

3. Fence an area on the reference cloud using the **Segmentation Tool**.
4. Enter a value in the **Horizontal Radius** field and press **Enter**.
5. Slide the cursor to a position between + and -.

The result from the square-grid projection and the distance selection is a set of invalid points (called **Unwanted Points**). These points will be un-kept and shown in red in the **3D View** - only if the **Display Unwanted Points (Red)** option is checked.

6. Click **Erase Unwanted**. Invalid points will be deleted from the reference cloud.
7. Click **Reload Reference Points** . The reference cloud is reloaded with invalid points less.
8. Repeat the steps from 3 to 7 on another area of the reference cloud.
9. Click **Create** to save the result and click **Close**.

A sub-point cloud whose name is "**By Topography**" is created under the current project in the **Models Tree**. **X** is its order.

Frame Creation Tool

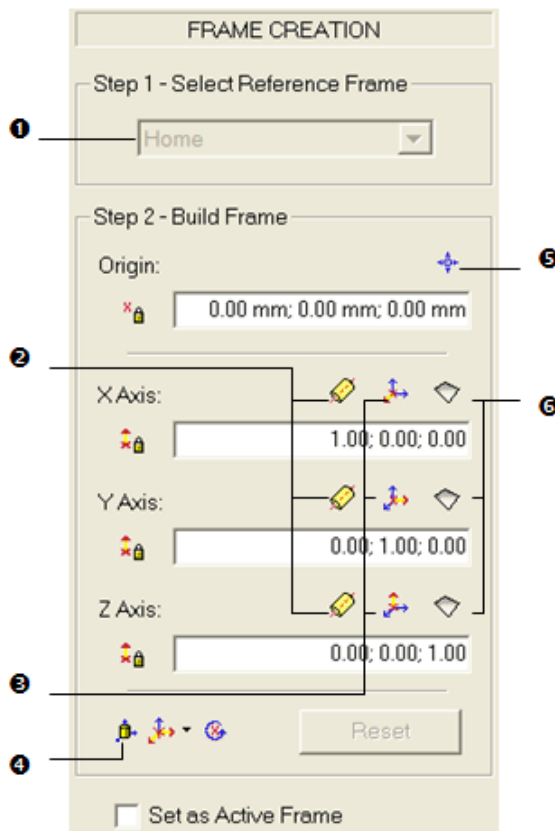
RealWorks enables to create any number of frames. Each of them may be selected and set as **Active Frame**. When a frame is designated as the **Active Frame**, all coordinates will be represented relative to this frame. This allows the user to perform its modeling or to take measurements in any arbitrary default frame and to represent them in a frame that better describes the data. The **Frame Creation Tool** provides you with several methods to create such frames and almost all of them are mainly based on pickings; which can be constrained or free.

Opening the Tool

To Open the Tool:

1. If needed, select an object (point cloud or geometry) from the **Project Tree**.
2. In the **Tools** menu, select **Frame Creation Tool**. The **Frame Creation** dialog opens as well as the **Picking Parameters** toolbar.

Inside the **Frame Creation Tool**, we distinguish two groups of methods to build a frame: "**Without Constraints**" and "**With Constraints**". Below is a detailed description of the **Frame Creation Tool**.



1 - Field for selecting a reference frame for parameters
 2 - Set Axial Geometry Axis as X/Y/Z Axis*
 3 - Pick X/Y/Z Axis*

4 - Pick Local Frame of Object
 5 - Pick Origin
 6 - Fit X/Y/Z Axis*
 7 - Reset the defined parameters

A temporary frame in yellow appears in the middle of the scene. If the default origin is too far from the displayed scene; **RealWorks** will prompt you to set it so that it matches the scene's center.

Tip: You can open the **Frame Creation Tool** by clicking its related icon in the **Active Frame** toolbar.

Note: (*) In the **X, Y, Z Coordinate System**.

Selecting a Reference Frame

If the project you load contains more than one frame, you can choose one as the **Reference Frame** in which you can specify the coordinates of a new frame to come. Otherwise, the **Reference Frame for Parameters** field is dimmed and the reference frame will be the default frame (**Home**).

To Select a Reference Frame:

1. Click the pull down arrow in **Step 1**.
2. Select a frame from the drop-down list.

Building a Frame without Constraints

To build a frame, you can define each of its items by specifying the coordinates, by picking points (seven are required if you wish to construct the whole frame using this method, two per axis and one for the origin), by fitting an axis with a plane, by picking points (origin and two directions) or by picking an object's local frame. For a given frame, you can mix these sub-methods (except for picking three points). Note that the coordinates for the three axes will be automatically normalized.

Specifying Coordinates

In the **By-Specifying-Coordinates** method, neither selection nor display is required. The default units of measurement for the **Origin** are set to millimeters and you do not have to enter “mm” after each value. Note that the coordinates for the three axes will be automatically normalized.

To Specify Coordinates:

1. Enter a 3D point position in the **Origin** field.
2. Enter a direction in the **X-Axis** field*.
3. Enter a direction in the **Y-Axis** field*.
4. Enter a direction in the **Z-Axis** field*.
5. Click **Reset** (if required).





Caution: If you create a frame which is not orthogonal, a warning message appears.

Note: In the **X, Y, Z Coordinate System**.

Picking Points

The **By-Picking-Points** method is based not on selection but on display. Display can be done before (or after) opening the **Frame Creation Tool** and should be either of point cloud type or of mesh type. **Pick Origin** requires one point to be picked whereas **Pick Axis** requires two points to be picked.


To Pick Points:

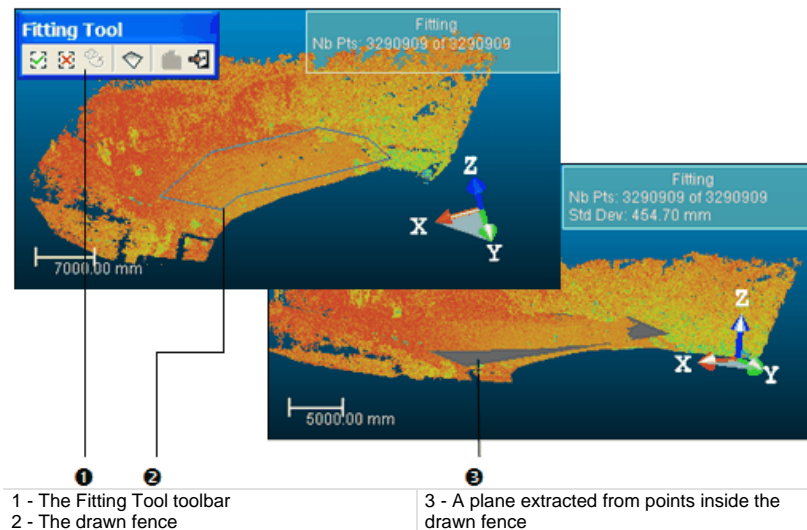
1. Click the **Pick Origin**  icon.
2. Pick one point (free or constrained).
3. Click the **Pick Axis** , , or  icon.
4. Pick two points (free or constrained). The two points give a direction for the axis.
5. Repeat the steps from 3 to 4 for the two other axes.
6. Click **Reset** to cancel the parameters (if required).

Fitting an Axis

The **By-Fitting-an-Axis** method is based not on selection but on display. Display can be done before (or after) opening the **Frame Creation Tool** and should be of point cloud type; otherwise the three **Fit Axis** icons are dimmed.

To Fit an Axis:

1. Click the **Fit Axis**  icon. The **Fitting Tool** toolbar appears.
2. Fence an area on the displayed point cloud.
3. Choose **In** to keep points inside the fence.
4. Or choose **Out** to keep points outside the fence.
5. Click **Plane**. A plane is extracted from kept points and its normal gives the direction of the axis.



6. Click **Reset** to cancel the parameters (if required).




Note: The dimmed **Create Fitted Geometry** icon in the **Fitting Tool** toolbar means that you cannot save the result in the database.

Tips:

- Instead of selecting **Close Fence** from the pop-up menu, you can also double-click or press on the **Space Bar** of your keyboard to close the fence.

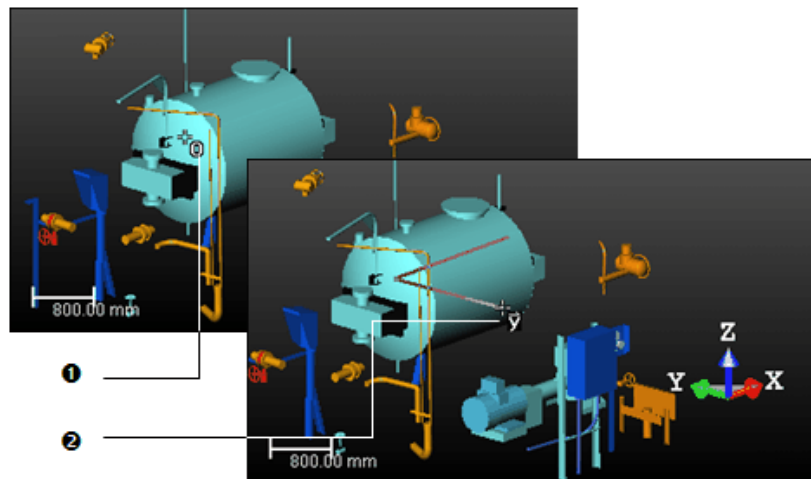
- Instead of clicking on an icon in the **Fitting Tool** toolbar, you can also select its related command from the pop-up menu or use its related short-cut key: **I** for **In**, **O** for **Out**.

Picking Three Points

To construct a frame by picking three points, you have the following options: Pick 3 Points (Origin, X direction*, Y direction*) , Pick 3 Points (Origin, Y direction*, Z direction*)  and Pick 3 Points (Origin, Z direction*, X direction*) . Once three points are picked, a right-angled frame will be created.

To Pick Three Points:

1. Click the **Change 3 Points Pick Mode** pull down arrow.
2. Choose a picking mode from the drop-down list.
3. Pick three points on the displayed object.



1 - The first picked point

2 - The second picked point

The first picked point will be the origin of the frame you wish to create.
The second picked point will form with the first one a first vector.
The third picked point will form with the first one the second vector.

4. Click **Reset** to cancel the parameters of the new frame (if required).


Notes:

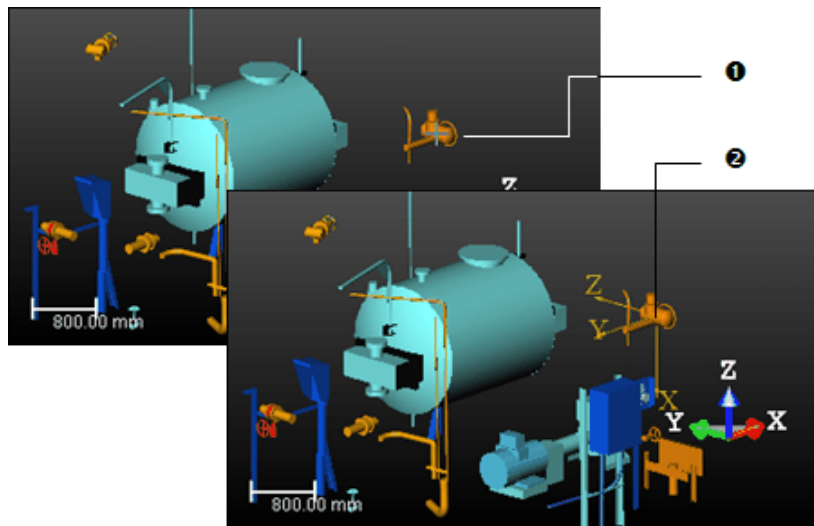
- You can select **Cancel Picking** from the pop-up menu or press **Esc** to cancel the frame in progress.
- In the **X, Y, Z Coordinate System**.

Picking an Object's Local Frame

You can construct a new frame such that it becomes the local frame of the selected object.

To Pick an Object's Local Frame:

1. Click the **Pick Object Local Frame**  icon.
2. In the **WorkSpace** window, click on the **Models** tab.
3. Right-click on the selected object.
4. Select **Display Geometry** from the pop-up menu.
5. Pick a point (free or constrained).



1 - The top picture shows a picked object


2 - Its associated local frame now becomes the constructed frame and is shown in yellow

6. Click **Reset** to cancel the parameters (if required).

Note: Picking another object will cancel the frame you have just constructed.

Setting an Axial Entity Axis as Axis

To Set an Axial Entity Axis as Axis:

1. Click e.g. the **Set Axial Geometry Axis as X-Axis**  icon.
2. In the **3D View**, pick an entity with an axial direction.

The values in the **X-Axis** field are updated with the values of the picked entity's axis direction.

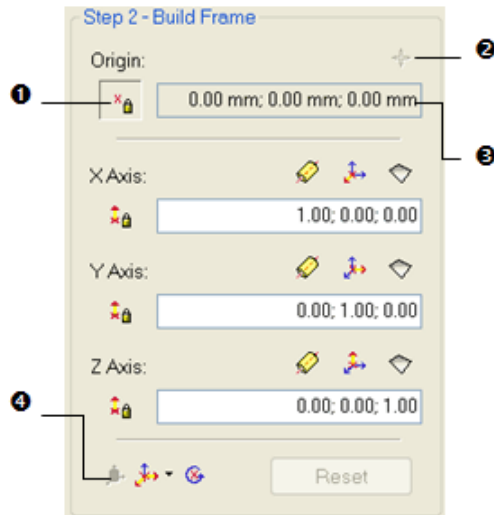
Building a Frame with Constraints

You can build a frame under constraint either by locking one of its components - for e.g. its origin or axis - or by rotating around an axis. No more than two components can be locked at once; it's always a pair formed by the origin and an axis. You cannot lock two axes together.

Locking the Origin

To Lock the Origin:

1. Define an origin and click **Lock Origin**.



1 - The Lock Origin is activated
2 - The Pick Origin is unavailable

3 - The Origin field is grayed out
4 - The Pick Local Frame of Object is unavailable

2. Use the three-point pick mode.
3. Or use the **Select Axial Entity Axis as Axis** mode.
4. Or use the **Pick Axis** mode.
5. Or use the **Fit Axis** mode.
6. Click **Reset** to cancel the parameters (if required).

Using the Three Point Pick Mode

To Use the Pick Three Points Mode

1. Click the **Change 3 Points Pick Mode** pull down arrow.
2. Choose a picking mode from the drop-down list.
3. Pick two points on the displayed object.

If no origin has been defined, the default origin will be the origin of the active frame. The first picked point will form a first vector with the default origin and the second picked point will form the second vector with the default origin. Once two points are picked, a right-angled frame will be created.

Using the Pick Axis Mode

To Use the Pick Axis Mode:

1. Click e.g. on the **Pick X-Axis*** button.
2. Pick two points on the displayed object.
3. Repeat the steps from 1 to 2 for the two other axes.

Note: (*) In the **X, Y, Z Coordinate System**.

Using the Fit Axis Mode

To Use the Fit Axis Mode:

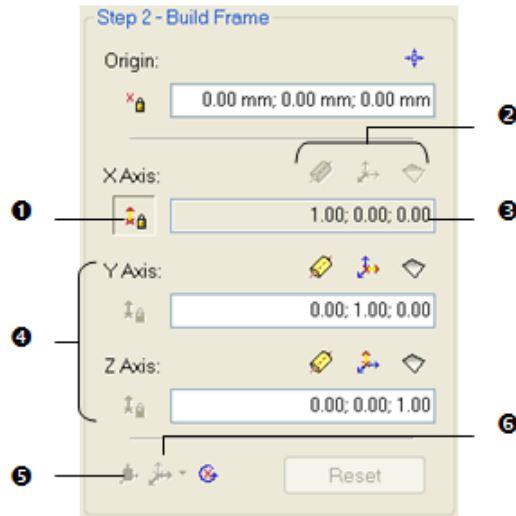
1. Click e.g. on the **Fit X-Axis*** button.
2. Define an axis by fitting a set of points.
3. Repeat the steps from 1 to 2 for the two other axes.

Note: (*) In the **X, Y, Z Coordinate System**.

Locking an Axis

To Lock an Axis:

1. Define e.g. the X-axis* and click **Lock X-Axis***.



- 1 - The Lock X-Axis* is selected
- 2 - The Take Axial Geometry Axis as X-Axis*, Pick X-Axis* and Fit X-Axis* are dimmed
- 3 - The X-Axis* field is dimmed

- 4 - The Lock Y-Axis* and Lock Z-Axis* are deactivated
- 5 - The Pick Local Frame of Object is unavailable
- 6 - The Change 3 Points Pick mode is unavailable

2. Define an origin.
3. Define two other axes.
4. Click **Reset** if required.

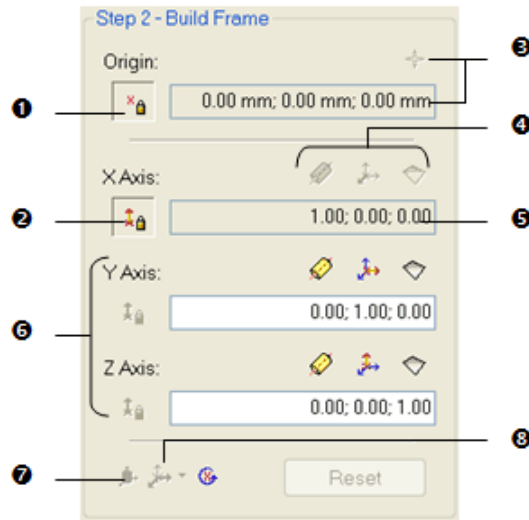
Notes:

- If you open the **Turn Around** dialog, you may see the unlocked axes dimmed (Y* and Z* in this example) and the locked axis (X)* checked by default.
- (*) In the **X, Y, Z Coordinate System**.

Locking the Origin and an Axis

To Lock the Origin and an Axis:

1. Define an origin and click **Lock Origin**.
2. Define e.g. the X-axis* and click **Lock X-Axis***.



1 - The Lock Origin is selected
 2 - The Lock X-Axis* is selected
 3 - The Pick Origin and the Origin field are unavailable
 4 - The Take Axial Geometry Axis as X-Axis*, Pick X-Axis* and Fit X-Axis* are dimmed

5 - The X-Axis* field is unavailable
 6 - The Lock Y-Axis* and Lock Z-Axis* are deactivated
 7 - The Pick Local Frame of Object is unavailable
 8 - The Change 3 Points Pick Mode is unavailable

3. Define two other axes.
4. Click **Reset** (if required).

Notes:

- If you open the **Turn Around** dialog, you may see the unlocked axes dimmed (Y* and Z* in the example) and the locked axis (X)* checked by default.
- (*) In the **X, Y, Z Coordinate System**.

Rotating Around an Axis

To Rotate Around an Axis:

1. Click **Turn Around Axis**. The **Turn Around** dialog opens.



1 - Lock Axis

2 - Turn Around Axis

2. Select an axis around which you want to rotate around by checking it.
3. Specify an angle. The default unit of measurement is set to degrees; you do not need to enter "°". You can change the default unit of measurement in **Preferences**.
4. Click **Apply**. You may see the yellow frame in the **3D View** turns around the selected axis of the specified angle.
5. Click again **Apply**. The yellow frame turns again around the selected axis and of the specified angle.
6. Click **Close**. The **Turn Around** dialog closes.

Note: You can combine the **Lock Axis** feature with the **Turn Around Axis** feature. In that case, the locked axis is only the available axis in the **Turn Around** dialog; the other axes are dimmed.

Creating the Built Frame

Once you have built a frame, you can create it in the database. You can use the **Set As Active Frame** option to set it as an active frame. If you leave the **Frame Creation Tool** without creating the newly built frame, a dialog opens and prompts you to create (or not) the frame.

To Create the Built Frame:

1. Check the **Set As Active Frame** option (if required).
2. Click **Create**. A new frame whose name is **OBJECTX** where **X** is its order is created in the **Models Tree**.
3. Click **Close**. The **Frame Creation** dialog closes.

Note: Press **Esc** (or select **Close Tool** from the pop-up menu) to leave the **Frame Creation Tool**.

Tip: Instead of clicking **Create**, you can also select **Create Frame** from the pop-up menu.

Shifting a Project

You can manually apply a shift to a project. It is not necessary to select it for that. Any item in the **Project Tree** can be used. This tool can be used in any processing mode (**OfficeSurvey**, **Registration** or **Modeling**).

To Shift a Project:

1. Select an item from the **Project Tree**.
2. From the **Tools** menu, select **Shift Project**. The **Shift Project** dialog opens.
3. Enter the coordinates of a vector in the **Define Vector Shift** field.
4. Click **Apply**. The **Shift Project** dialog closes.

For a station, the scanner origin changes,
For an image, its camera position changes,
For a geometry, its center changes,
Etc.

Note: You can undo the operation.

Converting to Meshes

The **Convert to Mesh** feature allows conversion a geometric entity like a cube, sphere, cylinder, cone, extruded model or plane (with holes or not) to a triangulated mesh. The created mesh is refined using parameters. This allows the application of texture to models.

To Convert a Geometric Entity to a Mesh:

1. Select a geometric object* from the **Models Tree**.
2. In the **Tools** menu, select **Convert to Mesh**. The **Convert to Mesh** dialog opens.
3. Enter a value in the **Average Triangle Edge Length** field.

The refinement consists of splitting the vertices for which the length is greater than the value set in the above field.

4. Click **Create**. The **Convert to Mesh** dialog closes.

A group whose name is "**Mesh - 'Average Triangle Edge Length' value**" is created under the current project in the **Models Tree**. A converted mesh whose name is **OBJECTX** is created and put under that group. **X** is its order.

Tip: (*) You can also select a mesh as an input. In this case, a new refined mesh is created and the selected mesh remains unchanged. You can compare the properties of both. The number of vertices and the number of triangles are changed consequently.

Creating a Merged Mesh

You can merge several meshes into a new one. A mesh can be either a fitted mesh (with points inside) or a pure mesh (no points inside).

To Create a Merged Mesh:

1. Select at least two meshes from the **Project Tree**.
2. From the **Tools** menu, select **Create a Merged Mesh**.

A merged mesh whose name is **OBJECTX** is created under the current project in the **Models Tree**. **X** is its order. This mesh has no point cloud representation inside.

Generating Key Plan

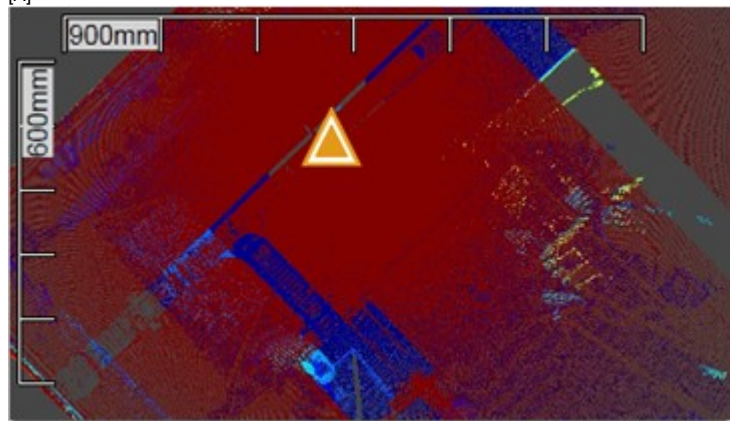
What is a **Key Plan**?. From **Trimble Scan Explorer**'s point of view, it is purely a 2D view (of the whole project or of a station (or set of stations)) with a set of triangles superimposed. Each triangle symbolizes a station's position. See [A]. From **RealWorks**'s point of view, a **Key Plan** is a **Preview** and a set of split **Ortho-images**. All are put under a folder named **Key Plan** under the **Images Tree**. See [C]. A **Key Plan** is mainly computed within **RealWorks*** and loaded in **Trimble Scan Explorer**. The computation can be done either from a unique (or a set of) **TZF** scan(s) or from a point cloud displayed in the **3D View**. For both methods, the user interaction is restricted to selecting the input. No parameters are required as they are automatically set.

One of the parameters is the **Projection Plane**, which is a plane on which points are projected. It is characterized by a projection direction (**Normal**) and a **Position**. The way this **Projection Plane** is set by default depends on the chosen method and on what is displayed in the **3D View** or not. The **Projection Plane** is a **Top View** when generating a **Key Plan** from **TZF** scans, and the focal plane when generating from the displayed point cloud.

Another parameter is an **Area Of Interest** which is used for computing a **Key Plan**. By default, the size of the **3D View** is considered as the **Area of Interest**. It is up to you to size the **3D view** to a dimension to compute a **Key Plan** to that size (only for the **Generate Key Plan From Current View** feature).

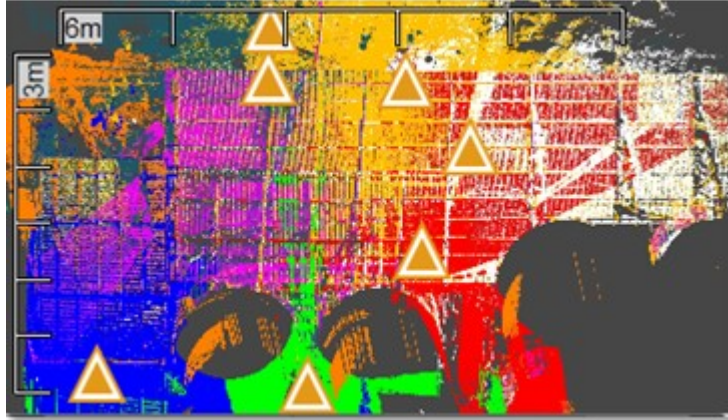
For the **Generate Key Plan from TZF Scans** feature, the option to render the computed **Key Plan** is based on the **Elevation** information. The **Elevation** value is calculated for each point based on its distance to the **Projection Plane**. Points that are far away from the **Projection Plane** are rendered **Red**. Those that are closer are rendered **Blue**.

[A]

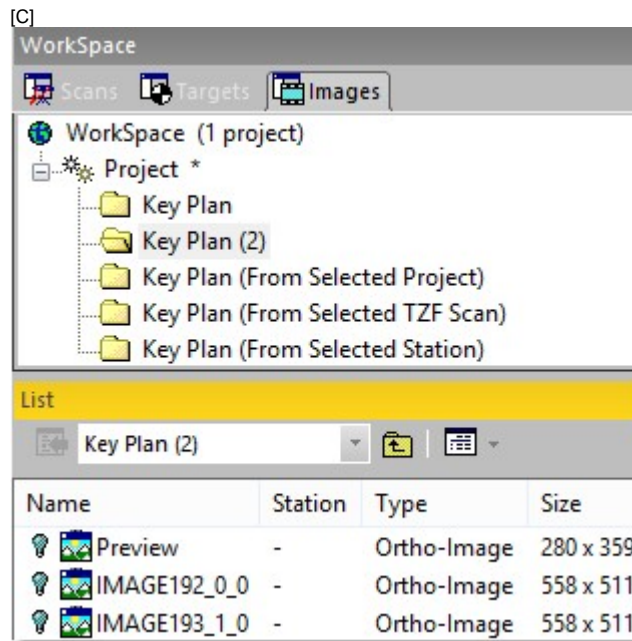


For the **Generate Key Plan From Current View** feature, the option to render the computed **Key Plan** is not based the **Elevation** information but on the **Cloud Rendering** options (**White Color**, **Cloud Color**, **Station Color**, **Scan Color**, **Gray Scaled Intensity**, **Color Coded Intensity** and **True Color**). Points are rendered according to the chosen option.

[B]



The **Preview** is an **Ortho-Image** of low resolution which is about 0.1 megapixel. The **Ortho-Image**, which is split into a set of small pieces of **Ortho-Image**, is of high resolution which cannot exceed 10 megapixels. Each split **Ortho-Image** is named as follows **ImageX_Line Index_Column Index**. It has a size (W x H) in pixels which is about 500 x 500 pixels.



Notes:

- (*) You should save your **RealWorks** project to be able to load the newly computed **Key Plan** in **Trimble Scan Explorer**.
- A file with the **RWV** extension is created per **Ortho-Image**. All **RWV** format files are put under the **RWI** folder.

Tip: All **Key Plans** are created in the root of the **Images Tree** and have the same name: **Key Plan**. Only its order indicated between brackets allows differentiating one **Key Plan** from another **Key Plan**. To make this distinction clearer and more obvious, we advise you to manually rename all **Key Plans** (in the **Name** line of the **Property** window).

From TZF Scans

This method uses the **Top** view (of the project) as a **Projection Plane** regardless of what is displayed in the **3D View**. Its **Normal** is parallel to the **Z-Axis** of the project's current frame. Its **Position** is the **Origin** or barycenter of the **Projection Plan**.

To Generate Key Plan From TZF Scans:

1. Select either a project⁽¹⁾ or a station⁽²⁾ or a **TZF Scan**⁽³⁾.
2. From the **Tools** menu, select **Generate Key Plan From TZF Scans**.

Notes:

- When selecting a project with some **TZF** format files inside, it is not necessary to be within a specific processing mode.
- The **Generate Key Plan From TZF Scans** feature is dimmed when there is no **TZF** scan in the project.
- You can be either in **Isometric** or **Perspective Projection Mode**.


Notes:

- ⁽¹⁾ All **TZF Scans** (of the project) are used for computing **Key Plan**.
- ⁽²⁾ All **TZF Scans** (of the station) are used for computing **Key Plan**.
- ⁽³⁾ Only that **TZF Scan** is used for computing **Key Plan**.

From Current View

This method uses the current camera view as the **Projection Plane**. Its **Normal** direction is perpendicular to the screen plane. Its **Position** is the **Origin** or barycenter of the **Projection Plan**.

To Generate Key plan From Current View:

1. Display a point cloud in the **3D View**.
2. Rotate the scene to specify the point of view from which you want to create a **Key Plan**.
3. From the **Tools** menu, select **Generate Key Plan From Current View** .

Notes:

- This feature is dimmed if there is no point cloud displayed in the **3D View**.
- You need to be in **Isometric Projection Mode**. Otherwise, the **Generate Key Plan From Current View** feature is grayed out.

Caution: You need to display only clouds from a single project. Otherwise an error message appears.

C H A P T E R 9

Media Tools

The **Media Tools** menu is present in all **RealWorks** products except in the **Viewer**.

Tools in the Media Tools Menu	Viewer	Base	Advanced, Advanced - Modeler, Advanced - Plant
Screen Capture (High Resolution)		✓	✓
Video Creator		✓	✓


Video Creator

The aim is to provide a tool that can generate videos from survey data. The video files are saved in the AVI (for Audio Video Interleave, a video format from Microsoft) format with customizable resolution and compression level.

Opening the Tool

No selection is required to enter the **Video Creator Tool**. Video creations are based on objects not necessary selected but displayed in the **3D View**. A selection can be done within the tool and the selected objects can be of all kinds (point cloud, mesh, geometry, etc.).

To Open the Tool:

1. Display objects in the **3D View**.
2. From the **Media Tools** menu, select **Video Creator** . The **Video Creator** dialog opens.

The **Video Creator** dialog opens as the third tab of the **WorkSpace** window and is composed of three parts. The first part (**Define Navigation Path**) contains methods for creating a video. The second part (**Define Video Parameters**) allows you to set parameters and preview the result. The last step is to save the result.

The **3D View** splits into two 3D viewers. The top 3D viewer (**Main View**) displays the global scene. The bottom 3D viewer (called either **Preview** or **Video Editing** and **Preview**) displays the view from the current keyframe. A keyframe is like a camera in a given position. The **Perspective** projection mode is set by default; you cannot swap to **Isometric**.

Defining a Navigation Path

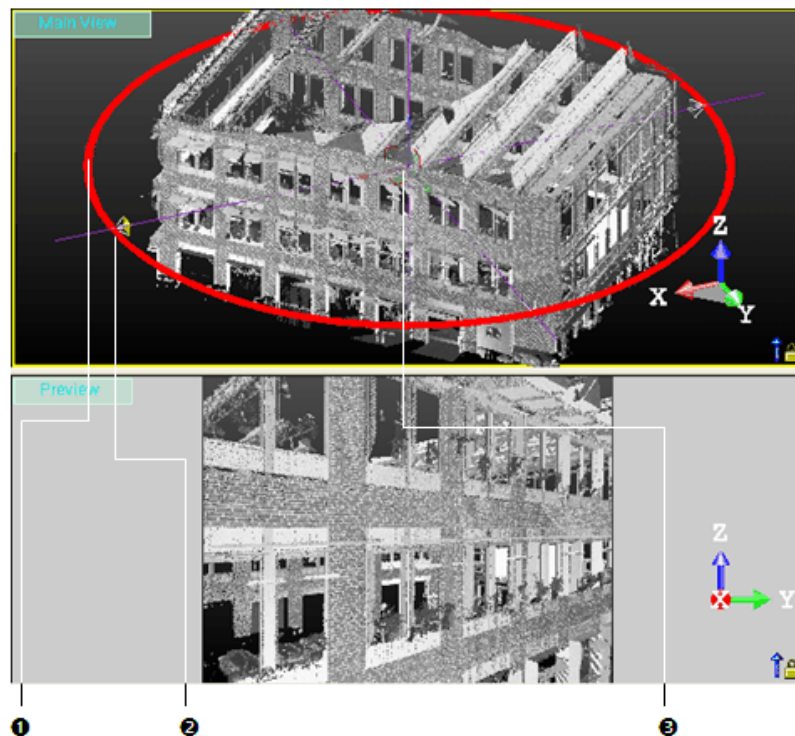
There are three modes for creating a video: **Quick Mode**, **Step-by-Step Mode** and **Path Mode**. After opening the **Video Creator Tool**, the mode that comes first is always **Quick Mode**. The **Quick Mode** uses a predetermined path (a circle). The **Step-by-Step Mode** allows you to define your own path by navigating through the scene. The **Path Mode** uses an existing polyline (or a drawn one) to describe the video path.

To Define a Navigation Path:

1. In **Step 1** of the **Video Creator** dialog, click on the pull-down arrow.
2. Select a mode among **Quick Mode**, **Step-by-Step Mode** and **Path Mode**.

Quick Mode

In the **Examiner** navigation mode, the top 3D viewer (called **Main View**) displays the global scene with a circular path (in red) and keyframes (four in all). All keyframes are directed to the center of the circular path. The bottom 3D viewer (called **Preview**) displays the view of the current keyframe (in yellow). The circular path has diameter the bounding box - that highlights the selection - diagonal. A 3D manipulator (three handles corresponding to three secant directions, each with a color (red, green and blue)) is located at the center of circular path. The **View Inwards/Outwards** button in **Step 1** of the **Video Creator** dialog is enabled.



1 - A circular path
2 - A keyframe


3 - The manipulator

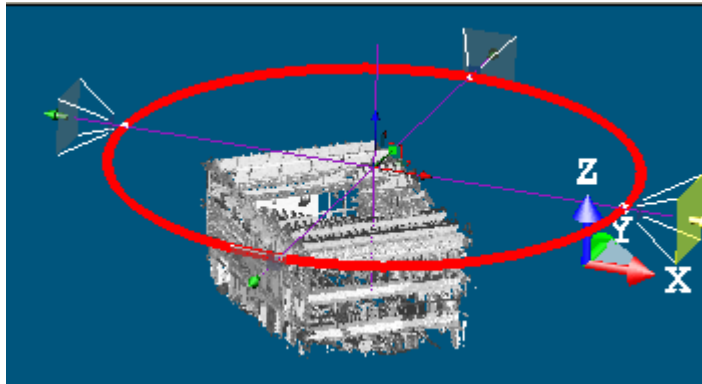
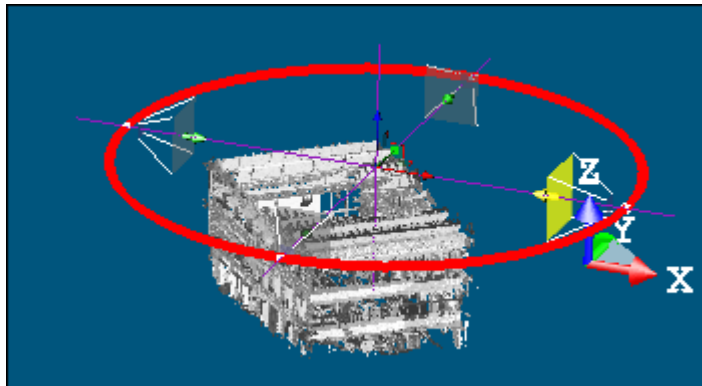
In the **Station-Based** navigation mode, the top 3D viewer displays the global scene with a keyframe. This keyframe is at the first station - in the **Project Tree** - position. The bottom 3D Viewer shows the viewpoint from that keyframe. The **View Inwards/Outwards** button in **Step 1** of the **Video Creator** dialog is dimmed.

Viewing Inwards/Outwards

You can reverse the keyframe direction so that all keyframes diverge from the center instead of converging on it.

To View Inwards/Outwards:

- In the **Video Creator** dialog, click the **View Inwards/Outwards**  icon.
- Or right-click in any 3D viewer and select **View Inwards/Outwards** from the pop-up menu.

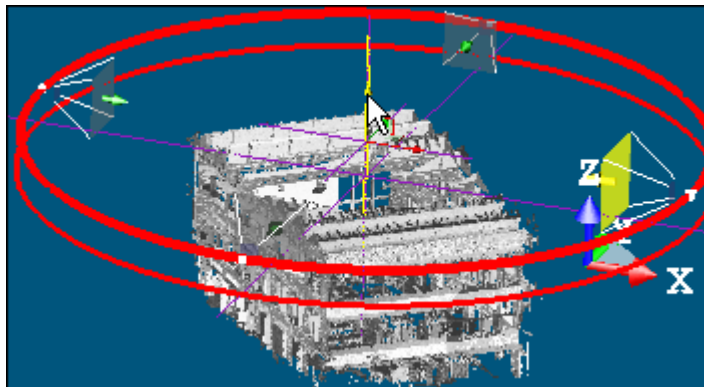
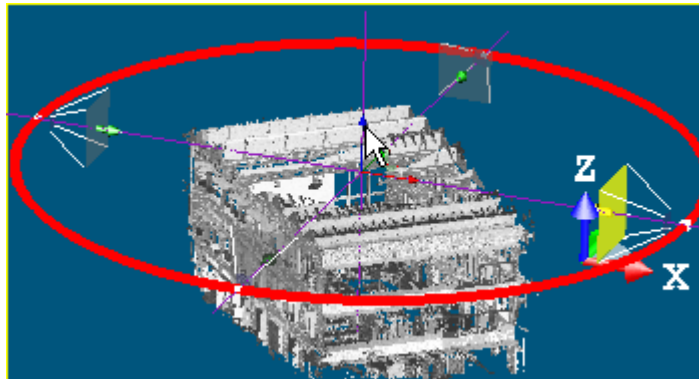


Note: You can undo (or redo) the inversion by clicking on the **Undo Operation** (or **Redo Operation**) button in the **Main** toolbar.

Moving Along a Direction

To Move the Circular Path Along a Direction:

1. Click on a handle; it turns to yellow. The direction along which you can displace the circular path (with keyframes) is highlighted in yellow and those along which you cannot displace the circular path (with keyframes) are in mauve.
2. Drag and drop to move the circular path (with keyframes) along that direction.

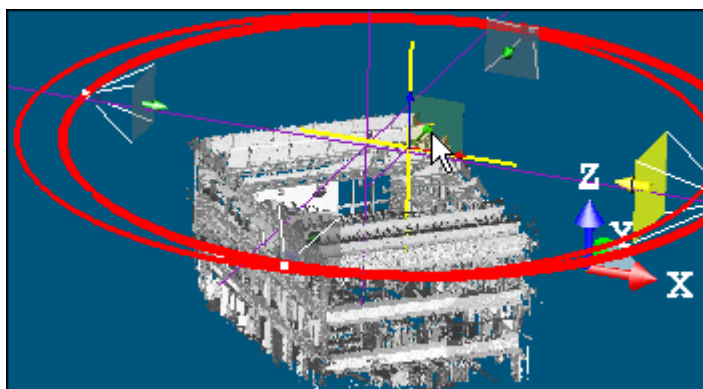
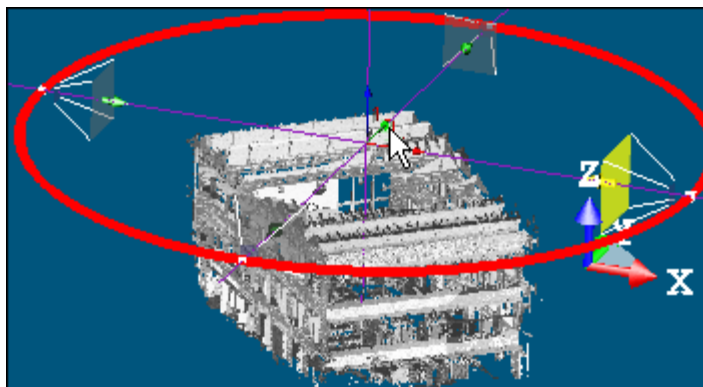


Note: You can undo (or redo) the displacement by clicking on the **Undo Operation** (or **Redo Operation**) in the **Main** toolbar.

Moving in a Plane

To Move the Circular Path in a Plane:

1. Click on a plane. It turns to yellow. The plane in which you can displace the circular path (with keyframes) appears in highlighted in yellow and those you cannot displace along are in mauve.
2. Drag and drop to move the circular path (with keyframes) in that plane.

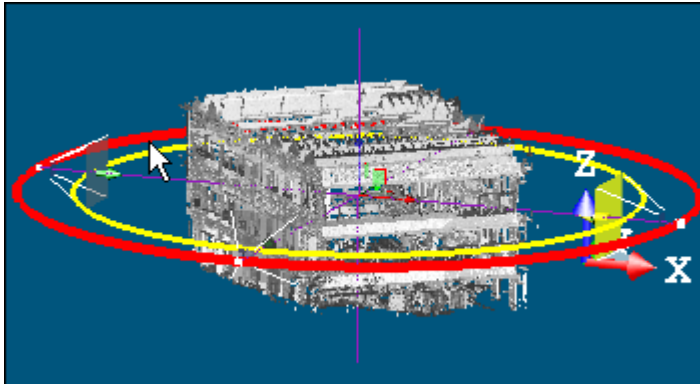
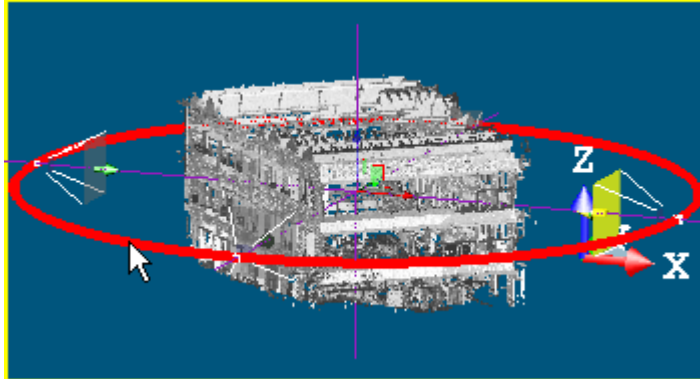


Note: You can undo (or redo) the displacement by clicking on the **Undo Operation** (or **Redo Operation**) in the **Main** toolbar.

Resizing the Circular Path

To Resize the Circular Path:

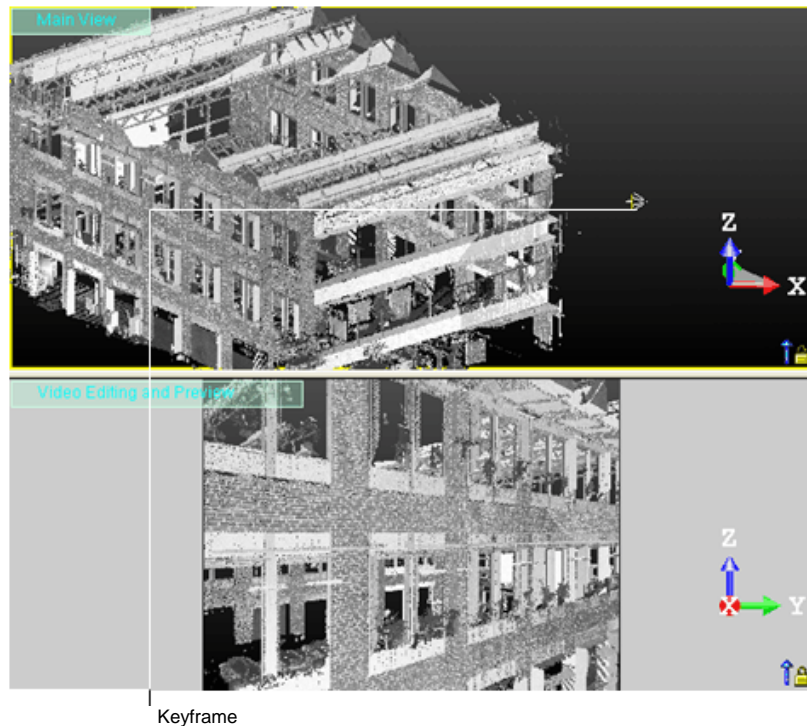
1. Click the circular path (with keyframes).
2. Drag and drop to enlarge or reduce the circular path (with keyframes) size.



Note: You can undo (or redo) the resizing by clicking on the **Undo Operation** (or **Redo Operation**) in the **Main** toolbar.

Step-by-Step Mode


In the **Examiner** (or **Walkthrough**) navigation mode, the top 3D viewer (called **Main View**) displays the global scene with an initial keyframe. The bottom 3D viewer (taking the name of **Video Editing and Preview**) displays the view from that keyframe. In both 3D viewers, the **Head Always Up** option and the **Perspective** project mode are default-set. The initial keyframe is at a position that corresponds to the current keyframe (of the previous creation mode) position with a shift. As the **Head Always Up** option is default-set; you cannot navigate with permanent constraints (**Horizontal Pan**, **Horizontal Rotation**, etc.) or temporary constraints (**Rotate** constrained around a vertical axis, **Pan** constrained along a vertical axis, etc.) in the 3D viewers. You can only zoom (in or out), pan and rotate.



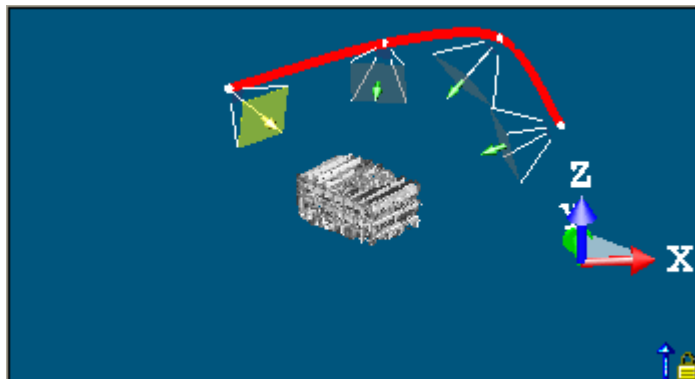
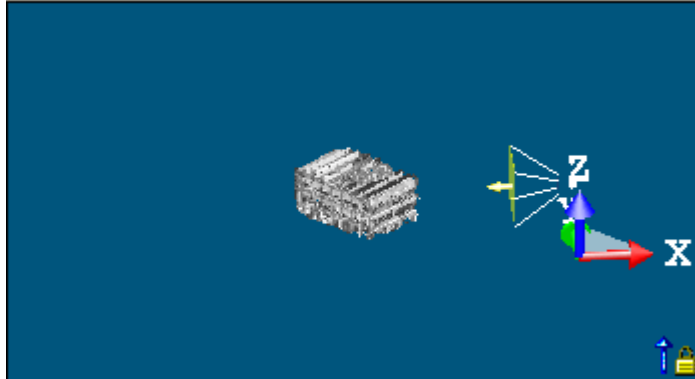
In the **Station-Based** navigation mode, the top 3D viewer remains unchanged. The bottom 3D viewer shows the view from the first station position with image(s) overlapped (if present). The initial keyframe is at the first station's position. The same navigation rules are applied in both 3D viewers except that you cannot pan in the bottom 3D viewer.

Adding Keyframes

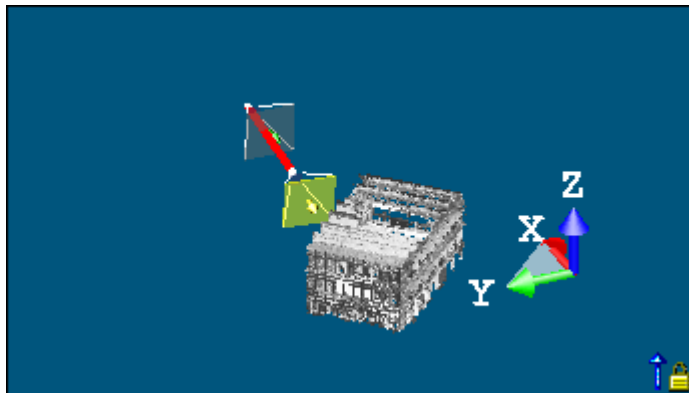
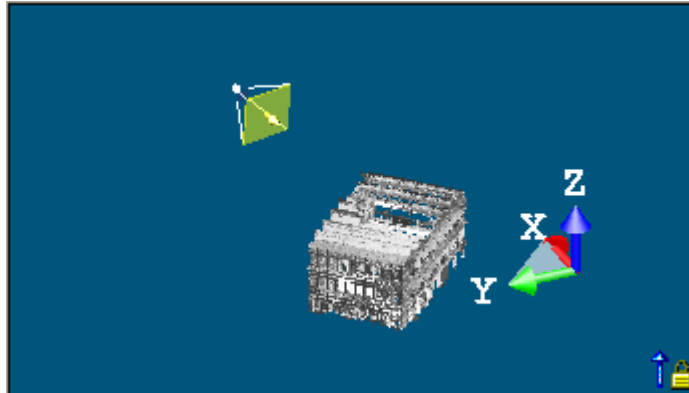
To Add a Keyframe:

1. In the bottom 3D viewer, navigate through the scene to find the right point of view.
2. In the **Video Creator** dialog, click the **Add New Keyframe**  icon. A keyframe is added in the top 3D viewer (in the first station position when you are in the **Station-Based** mode).
3. Repeat the steps from 1 to 2 to add another keyframe.

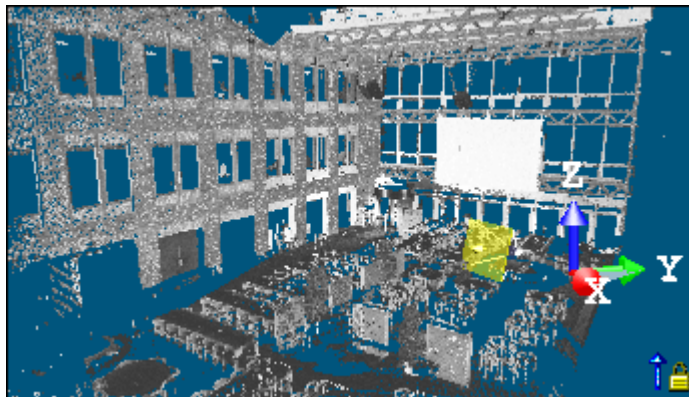
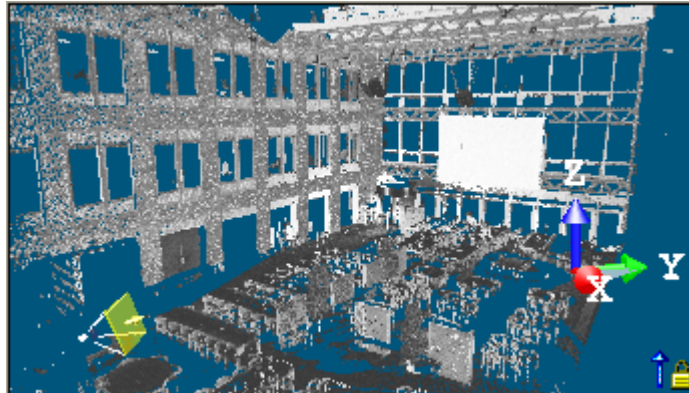
In the **Examiner** navigation mode, a new keyframe is added in the top 3D viewer. A red curve path links this new keyframe to the previous keyframe.



In the **Walkthrough** navigation mode, a new keyframe is added in the top 3D viewer at the same position as the previous keyframe but with a different direction if you tilt or look at a direction. A red curve path links this new keyframe to the previous keyframe if you pan or walk through the scene.



In the **Station-Based** navigation mode, a new keyframe is added in the top 3D viewer at the same position as the previous keyframe but with a different orientation if you rotate (or zoom) or at the second station position if you jump to the second station. There is no red curve path linking this new keyframe to the initial keyframe.



4. Repeat the steps from 1 to 2 to add other keyframes.

Tips:

- You can also click in a 3D viewer (top or bottom) and select **Add New Keyframe** from the pop-up menu.
- In the **Examiner** (or **Walkthrough**) navigation mode, use the **View Alignment** tools like **Center On Point** to help you rotating around a point.

Notes:

- Adding a new keyframe at the same position than an existing keyframe will make rotate the camera of 360°.
- You can undo (or redo) the addition of a keyframe by clicking on the **Undo Operation** (or **Redo Operation**) in the **Main** toolbar.
- You can switch from the **Examiner** navigation mode to the **Walkthrough** navigation mode and vice versa when adding new keyframes. You cannot switch to the **Station-Based** navigation mode if there are already keyframes. You need to delete all of them to be able to switch to this navigation mode.

Loading Keyframes

A keyframe file is a data file with *.dat as extension.

To Load Keyframes from a File:

1. Right-click in any 3D viewer to display the pop-up menu.
2. Select **Load Keyframes From File** from the drop-down menu. The **Load Keyframes From** dialog opens.
3. Navigate to the drive/folder where the file of keyframes is located.
4. Click on the file to select it.
5. Click **Open**.

Saving Keyframes

The **Save Keyframes to File** command is only available after adding keyframes. A keyframe file is a data file with *.dat as extension.

To Save Keyframes to a File:


1. Right-click in any 3D viewer to display the pop-up menu.
2. Select **Save Keyframes to File** from the drop-down menu. The **Save Keyframes as** dialog opens.
3. Navigate to the drive/folder where you want to store the keyframes file.
4. Enter a name in the **File Name** field. The extension is added automatically.
5. Click **Save**.

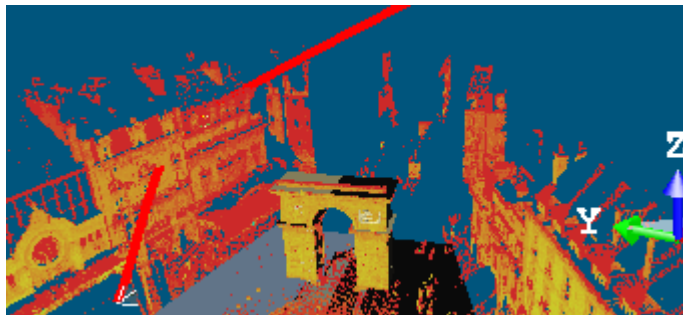
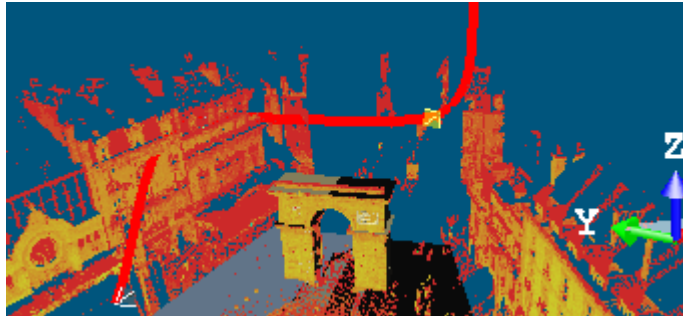
Editing Keyframes

After adding a keyframe, the **Clear All Keyframes** and **Delete Current Keyframe** icons as well as the **Save Keyframes to File** command become enabled.

Deleting a Keyframe

To Delete a Keyframe:


1. In **Step 1** of the **Video Creator** dialog, click the **Clear Current Keyframe**  icon. The current keyframe is removed from the sequence.
2. Or browse the sequence of keyframes using the navigation buttons to find the one you want to delete and click **Clear Current Keyframe**.



Note: You can undo (or redo) the deletion of a keyframe by clicking on the **Undo Operation** (or **Redo Operation**) button in the **Main** toolbar.

Deleting all Keyframes

To Delete all Keyframes:

1. In **Step 1** of the **Video Creator** dialog, click the **Clear all Keyframes**  icon.
2. Or right-click in a 3D viewer (top or bottom) and select **Clear all Keyframes** from the pop-up menu.

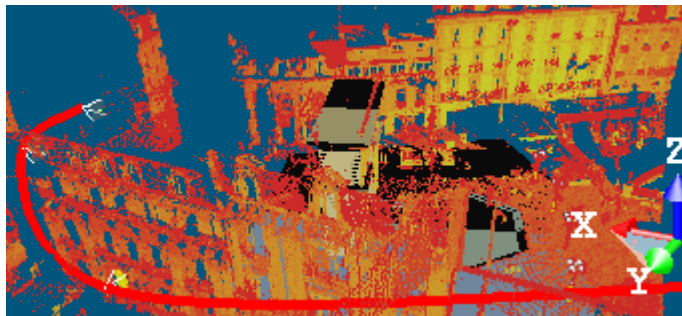
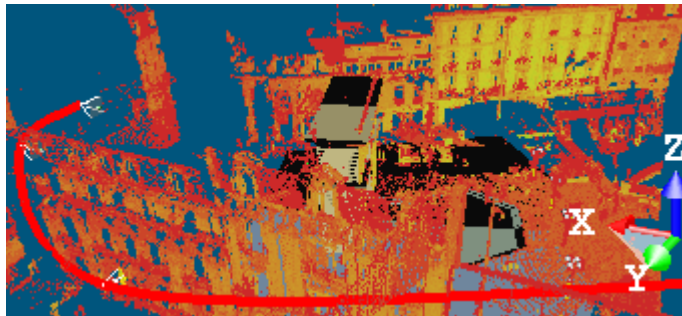
Note: You can undo the deletion of all keyframes.

Changing a Keyframe Position and Orientation

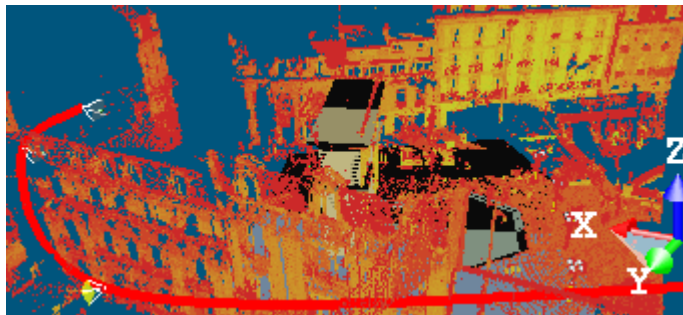
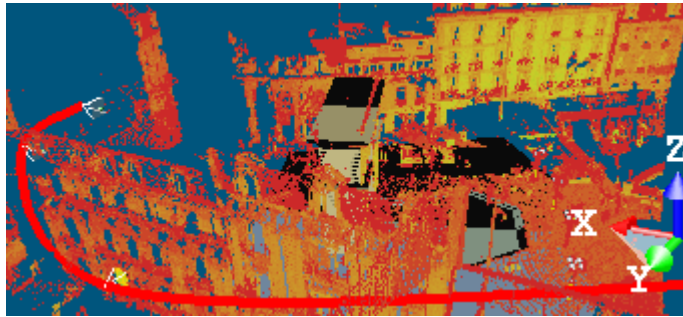
To Change a Keyframe Position and Orientation:

1. Browse the sequence of keyframes using the navigation buttons to find the one for which you want to change its position or orientation.
2. In the bottom 3D viewer, navigate through the scene to find the right point of view.

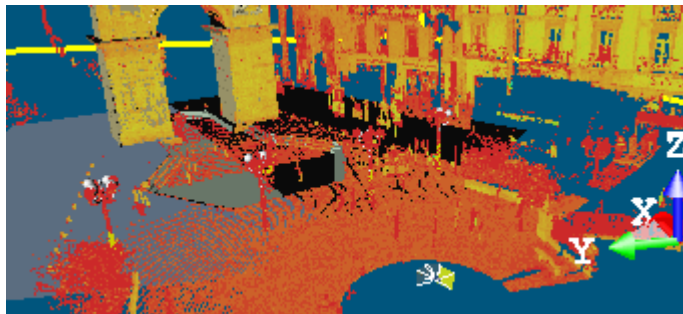
If you are in the **Examiner** mode, the current keyframe position changes as well as the path's shape.

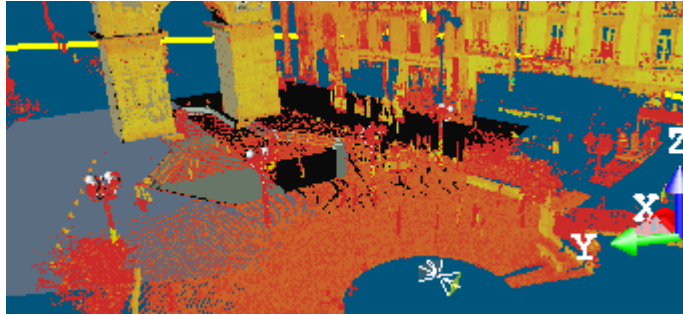


If you are in the **Walkthrough** mode, the current keyframe orientation changes while the path's shape remains unchanged.



If you are in the **Station-Based** mode, the current keyframe position changes. All are at the same position.

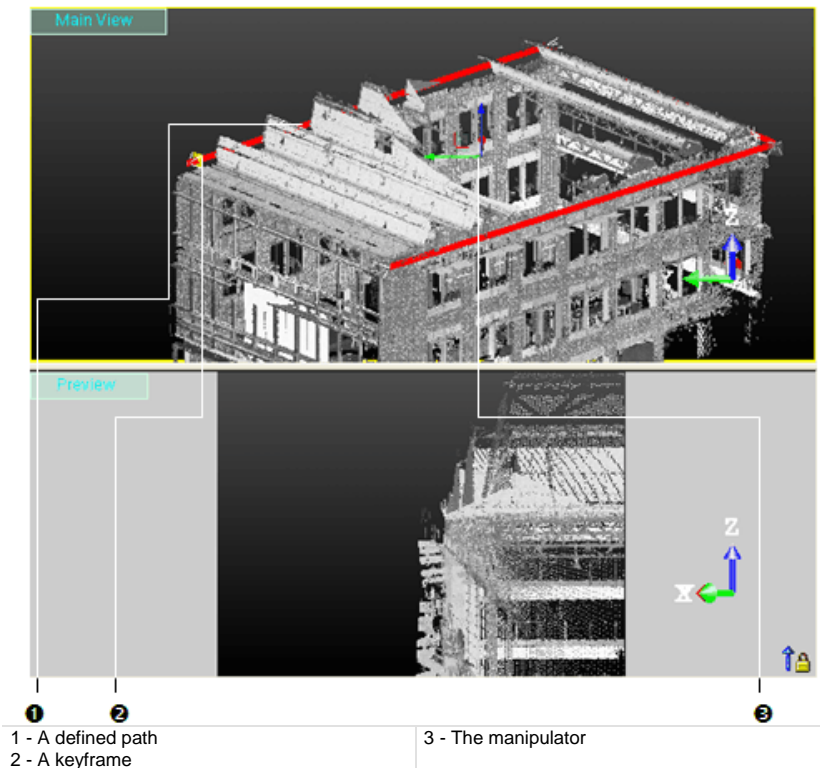




Note: You can undo (or redo) the operation by clicking on the **Undo Operation** (or **Redo Operation**) button in the **Main** toolbar.

Path Mode

The top 3D viewer displays the global scene with a path (if there is a polyline in your project) and keyframes (one at each node of the polyline). The initial keyframe is at the starting node's position. The bottom 3D viewer - taking the name of **Preview** - displays the view from the current keyframe (in yellow). If your project has no polyline; the top 3D viewer still displays the global scene but without path. The bottom 3D viewer keeps the view from the current keyframe (of the previous creation mode). In both 3D viewers, the **Head Always Up** option and the **Perspective** project mode are default-set. As the **Head Always Up** option is default-set; you cannot navigate with permanent constraints (**Horizontal Pan**, **Horizontal Rotation**, etc.) or temporary constraints (**Rotate** constrained around a vertical axis, **Pan** constrained along a vertical axis, etc.) in the 3D viewers. You can only zoom (in or out), pan and rotate.



Note: The navigation mode is restricted **Examiner** (or **Walkthrough**).

Selecting a Path

The loaded project contains one (or several) polyline(s). You can select one as a path for generating a video. The selected polyline needs to be regular (composed of one or several continuous segments with (or without) arcs).

To Select a Path:


1. Click on the **Choose Path** pull down arrow.
2. Select a polyline from the drop down list.
 - If the polyline is a set of continuous segments, a keyframe appears at each node.
 - If the polyline is a set of arcs, a keyframe appears at each node (start, mids and end).

Note: If the selected polyline contains more than twenty-two nodes, a warning message appears and prompts you to select the polyline or not.

Drawing and Creating a Path

When there is no polyline, you have to create at least one in the database. The top 3D viewer displays the global scene locked in 2D with a **2D Grid** superimposed (if not hidden previously). Movements while picking points are restricted to **Rotate** around the **Z-Axis**, **Zoom (In or Out)** along this same axis and **Pan** in the XY plane. The drawn polyline needs to be regular (composed of one or several continuous segments with or without arcs).

To Draw and Create a Path:

1. Click **Draw and Create Path in Database** . The **Drawing Tool** and **Picking Parameters** (in 2D constraint mode) toolbars appear. The mouse cursor shape changes to a pencil.
2. Draw a polyline by picking several points
3. Click **End Line**. The last picked point ends the line.
4. Click **Create** to save the drawn polyline in the database.
 - If the polyline is a set of continuous segments, a keyframe appears at each node.
 - If the polyline is a set of arcs, a keyframe appears at each node (start, mids and end).

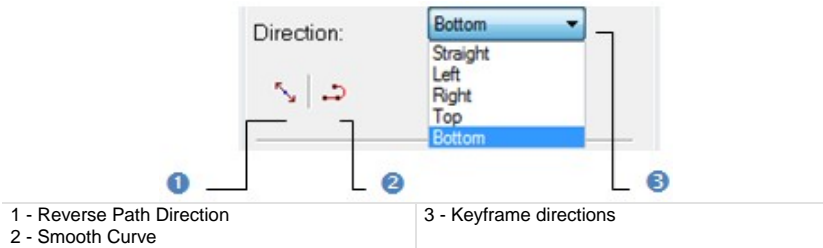
Tip: You can also select **Create** from the pop-up menu.

Notes:

- If the drawn polyline is composed of no continuous segments, an error dialog appears.
- If the drawn polyline contains more than twenty-two nodes, a warning message appears and prompts you to select the polyline or not.
- If the drawn polyline is a circle, five keyframes are generated. The first and fifth keys are in the same position. That's why only four keyframes are visible.

Setting a Direction

There is a polyline in your project or after drawing one, the **Direction** field and the **Reverse Path Direction** and **Smooth Curve** icons all become activated. Otherwise all are dimmed.

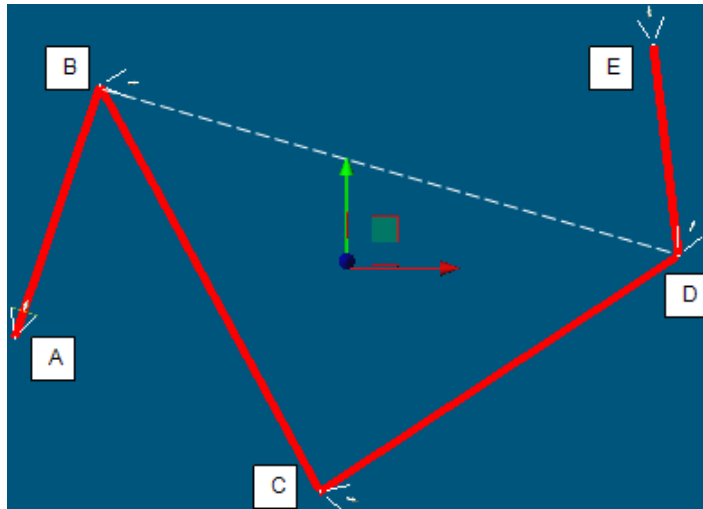


Note: **Straight** is the default direction.

Changing a Keyframe Direction

To Change Keyframe Direction:

1. Click on the **Direction** pull-down arrow.
2. Choose an item from the drop-down list.
 - If the polyline is a set of segments, **Straight** sets the start [A] and end [E] keyframes aligned respectively with the first [AB] and last [DE] segments and the other keyframes [C e.g.] parallel to the line passing through the previous and next keyframes.



- If the polyline is a set of arcs, **Straight** sets each keyframe tangent to its node.
- **Left** rotates all keyframes to the right of the **Straight** direction.
- **Right** rotates all keyframes to the left the **Straight** direction.
- **Top** rotates all keyframes so that they point upward.
- **Bottom** rotates all keyframes so that they point downward.

Note: The white dotted line in the picture above is not present in the top 3D viewer but just here for illustrating the explanation.

Reversing a Path Direction

To Reverse the Path Direction:

1. In the **Video Creator** dialog, click the **Reverse Path Direction** icon.
2. Or right-click in a 3D viewer (top or bottom).
3. Select **Reverse Path Direction** from the pop-up menu.
 - **Straight** sets the opposite direction.
 - **Left** becomes **Right**.
 - **Right** becomes **Left**.
 - **Top** becomes **Bottom**.
 - **Bottom** becomes **Top**.

Tip: You can combine the **Reverse Path Direction** feature with the **Smooth Curve** feature.

Smoothing a Path

To Smooth a Path:

1. In the **Video Creator** dialog, click the **Smooth Curve** icon.
2. Or right-click in a 3D viewer (top or bottom).
3. Select **Smooth Curve** from the pop-up menu.

Tip: You can combine the **Smooth Curve** feature with the **Reverse Path Direction** feature.

Moving Along a Direction

To Move the Path Along a Direction:

1. Click on a handle; it turns to yellow. The direction along which you can displace the path (with keyframes) is highlighted in yellow and those along which you cannot displace the path (with keyframes) are in mauve.
2. Drag and drop to move the path (with keyframes) along that direction.

Note: You can undo (or redo) the displacement by clicking on the **Undo Operation** (or **Redo Operation**) in the **Main** toolbar.

Moving in a Plane

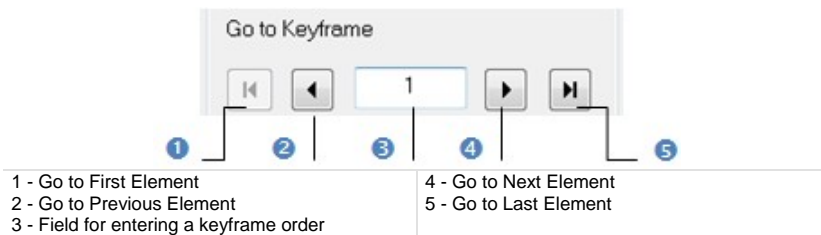
To Move the Path in a Plane:

1. Click on a plane. It turns to yellow. The plane in which you can displace the path (with keyframes) appears in highlighted in yellow and those you cannot displace along are in mauve.
2. Drag and drop to move the path (with keyframes) in that plane.

Note: You can undo (or redo) the displacement by clicking on the **Undo Operation** (or **Redo Operation**) in the **Main** toolbar.

Browsing Keyframes

A sequence of keyframes defines which movement the spectator will see, whereas the position of the keyframes on the video defines the timing of the movement. An active keyframe - the one in yellow in the top 3D viewer - sets the starting point of that movement. If more than one keyframe is available and if the active keyframe is other than the first one, you can browse through the sequence as described below:



To Browse Keyframes:

- Click **Go to Previous Element** (or **Go to Next Element**) to set the previous (or next) keyframe as active.
- Click **Go to First Element** (or **Go to Last Element**) to set the first (or last) keyframe as active.
- Key in a keyframe order in the path in the **Go to Keyframe** field to select it. You do not need to validate by pressing the **Enter** key.

Tips:

- Use the **Home** (or **End**) button of your keyboard instead of **Go to First Element** (or **Go to Last Element**).
- Use the **Up** (or **Down**) arrow of the keyboard instead of **Go to Previous Element** (or **Go to Next Element**).

Defining Video Parameters

To Define Video Parameters:

1. Enter a value in the **Duration** field.



2. Click on the pull-down arrow below the **Duration** field.
3. Choose a value among **x1**, **x2**, **x4** and **x10** from the drop-down list.

Defining a Duration

A **Duration** is second.

To Define a Duration:

- Enter a value in the **Duration** field.

Previewing a Video

To Preview a Video:

1. Click **Play**. The video is launched and the **Play** button becomes **Stop**.
2. Click **Stop** to stop the video preview.

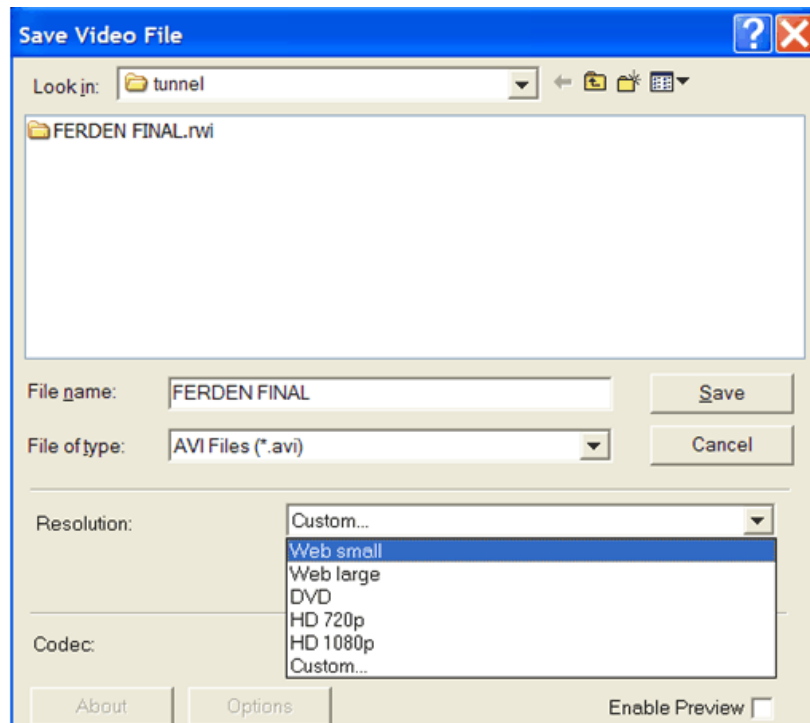
Notes:

- Press **Esc** to stop the video preview.
- The video is previewed in the 4/3 format.

Creating a Video

To Create a Video:

1. Click **Create**. The **Save Video File** dialog opens.
2. Navigate to the drive/folder where you want to store the video.
3. Enter a name in the **File Name** field.
4. Click on the **File of Type** pull-down arrow.
5. Select a type from the drop-down list.
6. Click on the **Resolution** pull-down arrow.



7. Do one of the following:
 - Use a predetermined resolution.
 - Choose among **Web small** (240x188), **Web large** (320x240), **DVD** (720x576), **HD 720p** (1280x720) and **HD 1080p** (1929x1080).

- Customize your own resolution.
 - a) Select **Custom**. The two **Pixels** fields become editable with 320 x 240 as default values.
 - b) Set your own resolution.
- 8. Click on the **Codec** pull-down arrow.
- 9. Choose "**Uncompressed**" to not compress the video. In that case, the **About** and **Options** buttons remain dimmed.
- 10. Or choose a codec from the drop-down list. Both the **About** and **Options** buttons active.
- 11. If required, click on the **About** button. A codec - for which the information is missing when clicking on the **About** button - will not open any dialog. Otherwise, an information box appears.
- 12. Click on the **Options** button and configure your own options.
- 13. If required, check the **Enable Preview** option.
- 14. Click **Save**. The **Save Video File** dialog closes. **RealWorks** will then encode the video. When encoding is complete, a box with the following information - End of operation notified, Elapsed time for the encoding and location of the video in your hard drive - appears.

Notes:

- A warning message appears when the resolution values are invalid. You can only set a value between 100 and 2000.
- Pressing **Esc** stops the video encoding. A message which prompts you to cancel the operation (or not) appears.
- If the **Enable Preview** option has been checked, you may see a keyframe in movement along the defined path and a preview of the video respectively in the top 3D viewer and bottom 3D viewer. And the preview format in the bottom 3D viewer switches from 4/3 to the resolution set in **Save Video File** dialog.
- The codecs in the drop-down list when clicking on the pull-down arrow are those installed in your computer system. For practical purposes, we recommend you to install the (free) codec to ensure that your videos reach a wide audience.

Capturing Screen (High Resolution)

This command allows you to capture the screen of the **3D View** window. The images are saved as high resolution image files in BMP format.

To Capture the Screen in High Resolution:

1. From the **Media Tools** menu, select **Capture Screen (High Resolution)**. The **Save** dialog opens.
2. Type a name for the image file. **RealWorks** assigns automatically the .BMP extension to this file.
3. Specify the location where you want to store this file by navigating through the drive/folder.
4. Click **Save**.

CHAPTER 10

Registration

When you load a file of the following formats (SIMA and TXT with Topopoints) that was never saved in the **RealWorks** format; the **Registration** processing mode is set by default. When you load a file - saved in the **RealWorks** format and in the **Registration** processing mode; that file will be opened with that processing mode set. When you are outside this processing mode and you need to use it, you have to click on the pull-down arrow in the **Tools** toolbar and choose **Registration**. When you swap the **OfficeSurvey™** processing mode for the **Registration** processing mode; you may met the following message "The Survey Configuration load state will be kept in the Registration configuration. Do you want to save the previous Registration configuration load state?".


The **Registration** processing mode is present in all **RealWorks** products. In **RealWorks Viewer**, only the **Registration Report (Target-Based)** feature is present.

Auto-Extract Targets and Register

The **Auto-Extract Targets and Register** feature allows the extraction of targets from **TZF Scans**, to match those in common and to register the stations the extracted targets belong to.

Opening the Tool

To Open the Tool:

1. Select a station, a set of stations or a project⁽¹⁾ created from **TZF** format file(s)⁽²⁾ from the **Scans Tree**.
2. In the **Registration** menu, select **Auto-Extract Targets and Register** 
 - If a station (or set of stations) has (or have) been selected, the **Auto-Extract Targets and Register** dialog opens.
 - If a project has been selected, a dialog opens and prompts you to process with all stations (or not). Click **Yes**. The dialog closes and the **Auto-Extract Targets and Register** dialog appears.
3. Choose a **Target Type**.
4. Create sampled scans.
5. Choose a Reference Station.
6. Click **OK**. The **Auto-Extract Targets and Register** dialog closes.

Notes:

- ⁽¹⁾ With a unique station (or a set of stations (or a set of groups)). Otherwise, if the input is a project with only a unique group, the tool is grayed-out.
- If the input is a group (with a station (or a set of stations)), the tool is grayed-out.

Caution: If there is no **TZF** format file inside the selected station(s) (or project), a dialog opens and asks you if you wish to continue with the remaining station(s).

Note: ⁽²⁾ If the **TZF** format file(s) has (have) not been yet processed, the **Processing TZF Scans** dialog opens and prompts you to proceed to do so.

Caution: A warning message appears in the **Auto-Extract Targets and Register** dialog when one of the selected stations has a **Level 3 Scan** within.

Note: You need to have at least one **Target Type** checked to enable the **OK** button. Otherwise, it remains dimmed.

Target Types

You can extract two types of target: **Spherical Target** or **Black and White Flat Target**.

To Choose a Target Type

1. Check both options: **Spherical Target** and **Black and White Flat Target**.
2. Or only check one type.
3. If **Spherical Target** has been checked, the **Diameter** field becomes enabled.
4. Input a value in the **Diameter** field according to the type of sphere you used during data acquisition.
5. Or click on the **Diameter** pull-down arrow.
6. And choose a value from the drop-down list.
 - There are four predefined diameters: 76.20 mm, 100 mm, 139 mm and 200 mm.

Tip: The current unit of measurement is in **Millimeters**. You do not need to enter "mm".

Note: Extracted targets are created in the database as **Spherical Targets** (or **Black and White Flat Targets**) with **Unmatched** status. All of them are gathered in the **Unmatched** folder under the **Project** node in the **Project Tree** and each one is put under its related station. An extracted target is named **TargetX** where **X** is an order, whatever its type.

Caution: The minimum distance between two targets should be 200 mm between two centers. The threshold is the same for **Spherical Targets** as for **Black and White Targets**.

Scan Creation

You can create a **Scan** based on the **Preview** of a **TZF Scan** (by getting points from the **Preview** and computing **Normals** on them). A **Scan** is always named **Preview**. The number of points for each is about two million points.

To Create a Sampled Scan:

- Keep the **Generate a Preview Scan** option checked.

Notes:

- If several stations have been selected as input, a set of **Scans** (one per station) is created in batch mode, one after the other. You can interrupt each of them by pressing **Esc**.
- If the **Generate a Preview Scan** option has been checked, you will be prompted to save the current project in the **RealWorks** database, if it is not yet saved. If the option has been kept unchecked, no prompt appears.

Tip: When you create from several **TZF Scans** within a station, all **Sans** (in that station) have not the same color. Each has its own color.

Reference Station

You have to choose a station from the project (or from the set of stations) and set it as a **Reference Station**. This means that the chosen station will be used as a reference (station remaining unchanged) and the other stations as stations to register with.

To Choose a Reference Station:

1. Click on the **Reference Station** pull-down arrow.
2. Choose a station from the drop-down list.

Notes:

- If a single station has been selected as input, no registration will occur.
- If a project has been selected as input, the first station (of the project) is by default the **Reference Station**.

Note: If a set of stations has been selected as input, the first selected station is by default the **Reference Station**. The order (of selection) is preserved.

Caution: After clicking **OK** in the **Auto-Extract and Register** dialog, if the station selected as **Reference Station** is not a **Leveled Station**, an error message appears and prompts you to change the selection. Close the **Error** message. The **Leveled Station** is automatically set as **Reference Station** in the dialog.

Caution: After clicking **OK** in the **Auto-Extract and Register** dialog, an error message appears if the input (of the tool) contains a **Topographic Station** and the station has not been chosen as **Reference Station**. Close the **Error** message. The **Topographic Station** is automatically set as **Reference Station** in the dialog.




Registering Stations

To Register Stations:

- In the **Auto-Extract Targets and Register** dialog, click **OK**.

If there are enough targets inside each station (and at least three in common between two stations), the auto-pairing of targets will be performed and the **Target-Based Registration** dialog opens.

If the auto-pairing of targets succeeds, you may see the number of paired targets in the **Station List** of the **Target-Based Registration** dialog. For a given station, the **Number of Targets** is shown as **X/Y**. **Y** is the sum of targets and **X** is the sum of matched targets. The **Adjust** button in **Step 3** of the **Target-Based Registration** dialog is dimmed (because stations are registered). The Registration Details dialog opens automatically with the **Station View** set by default.

Station List		
Station Name	Number of Targets	Res. Error
 TeachingF...	4/14	16.84 mm
 TeachingFirs...	7/10	10.27 mm
 TeachingFirs...	4/8	2.34 mm

1 - Reference Station (in bold)
2 - Matched targets in a station
3 - Total of targets (matched and unmatched) in a station

If there are not enough targets inside each station and/or if there are no common targets between stations, the auto-pairing of targets will fail and the **Adjust** button in **Step 3** of the **Target-Based Registration** dialog is enabled. The **Registration Details** dialog is not open.


Note: The **Target-Based Registration** dialog will not open if the input is a single station.

Auto-Register Using Planes (Target-Less)

The **Auto-Register Using Planes (Target-Less)** feature automatically registers leveled scans of structured environments, i.e. that contain a significant amount of predominantly flat walls. It automatically extracts all the main planes (walls, ground, ceiling, etc.), matches them automatically between scans and uses them to register the scans.

Opening the Tool

To Open the Tool:

1. Select at least two stations, a group (or set of groups)⁽¹⁾, or a project⁽²⁾ created from **TZF** format file(s) from the **Scans Tree**.
2. From the **Registration** menu, select **Auto-Register Using Planes (Target-Less)** . The **Auto-Register Using Planes (Target-Less)** dialog appears.

Notes:

- ⁽¹⁾ With at least two stations inside a group. Otherwise, if there is only a unique station in a group, the tool is grayed-out.
- ⁽²⁾ With several stations, a unique group or a set of groups. Otherwise, if the project has only a unique station within, the tool is grayed-out.

Caution: If the input does not contain at least two valid stations (with **TZF Scan** within), a warning dialog opens and the tool cannot be launched anymore.

Tip: If stations are gathered per group, auto-register first all the stations (of a group) together. Once done for all groups, auto-register then all groups together.

Reference Station

The **Reference Station** is the one whose position and orientation remain unchanged through the **Auto-Register Using Planes (Target-Less)** process. If a project (or a set of stations) has been selected as input, the first station from the set of stations (or from the project) is the default **Reference Station**. If a leveled station has been chosen as input, this station is by default **Reference Station**.

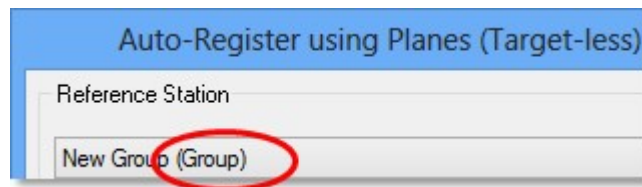
To Choose a Reference Station:

1. Click on the **Selection List** pull-down arrow.
2. Choose a station from the drop-down list.
 - If a project has been selected as input, all stations (of the project) are in the **Selection List**.
 - If a set of stations has been selected as input, only the selected stations (of the set) are in the **Selection List**.

Note: If a set of stations has been selected as input, the first selected station is by default the **Reference Station**. The order (or selection) is preserved.

Caution: After clicking **OK** in the **Auto-Extract and Register** dialog, if the station selected as **Reference Station** is not a **Leveled Station**, an error message appears and prompts you to change the selection. Close the **Error** message. The **Leveled Station** is automatically set as **Reference Station** in the dialog.

Tip: **RealWorks** can differentiate a station from a group (of stations). A group is flagged as "(Group)" in the **Auto-Register Using Planes (Target-Less)** dialog.



Registering Stations

To Register Stations:

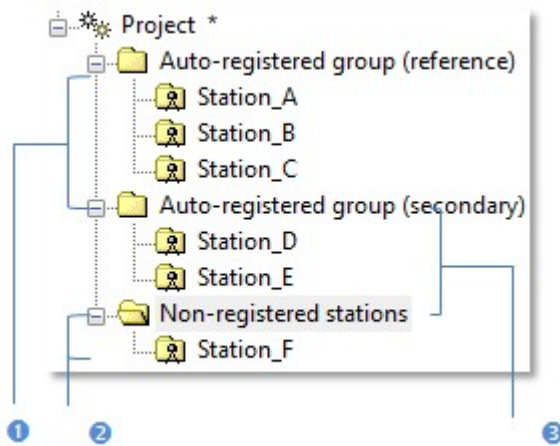
1. In the **Auto-Register Using Planes (Target-Less)** dialog, click **Start**. Stations are then registered together*.
2. If stations can be registered together, the **Registration Report** dialog opens.
3. In the **Registration Report** dialog, do one of the following:
 - Click **Save In RTF**. The **Registration Report** dialog opens.

Or

- Click **Close**. The **Registration Report** dialog closes. An **Auto-Registered Group** is created and rooted under the **Scans Tree**.

An **Auto-Registered Group** is a group gathering the stations selected for the **Auto-Register Using Planes (Target-Less)** purpose. We distinguish two different groups, one called **Reference** and the other **Secondary**. Basically an **Auto-Registered Group (Reference)** is a group that contains stations registered together and for which one of them is registered with a station chosen as the **Reference Station**. An **Auto-Registered Group (Secondary)** is a group with stations registered together and any of them cannot be registered with the **Reference Station**. An **Auto-Registered Group (Reference)** can contain a sub-group which is also an **Auto-Registered Group (Reference)**. This occurs when both share the same **Reference Station**.

Stations that cannot be registered with the others are put under a folder named "**Non-Registered Stations**".



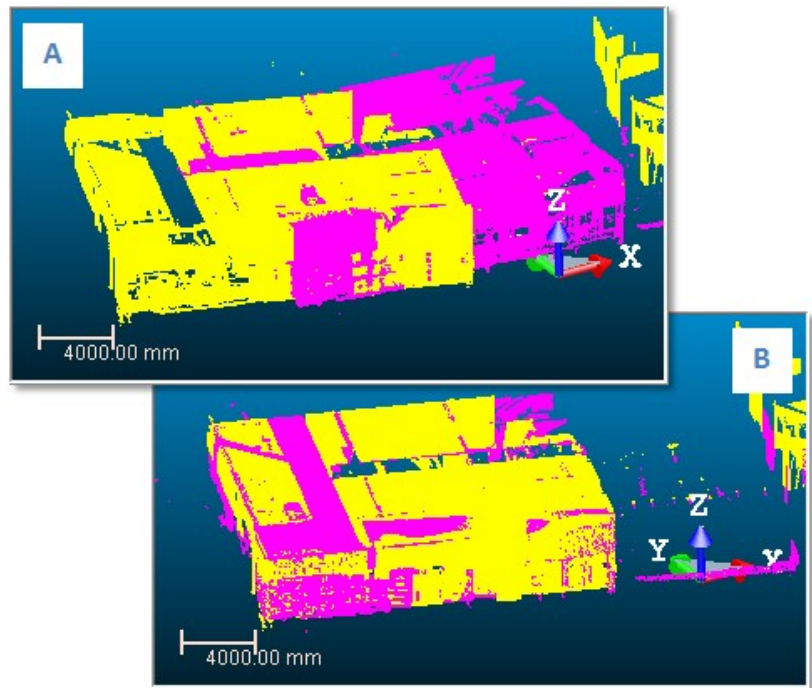
1 - Group with registered stations among which one of them is a Reference Station

2 - Group with stations for the registration with the Reference Station failed

3 - Group containing all stations that cannot be registered with others

And/or

- If the Generate a Preview Scan option has been checked, the **Registration Report** dialog remains open. Sampled scans are then created. Again in the **Registration Report** dialog, click **OK**. The **Registration Report** dialog closes.



A - Common parts of the clouds are not yet superimposed

B - Common parts of the clouds are superimposed

4. If stations cannot be registered together, an error dialog opens and suggests you to try the Cloud-Based Registration Tool.

Note: (*) The user can abort the registration (of stations) in progress by clicking **Esc**. A dialog opens and prompts you to abort or not.

Registration Report

The **Registration Report** dialog lists for each station (of the selection) the following information:

- How many station(s) each of them is registered with and the name of each.
- The deviation in a pair registered stations in the current unit of measurement.

The purpose of the **Auto-Register Using Planes (Target-Less)** feature is to register stations based on planes (paired together). The accuracy of two stations registered together is given by the **Cloud-to-Cloud Error**. This error is an average distance between paired planes (of one station) and the point cloud (of the other station). The **Cloud-to-Cloud Error** in a pair (of registered stations) is the same from one direction to the other (e.g. from Station_A to Station_B or from Station_B to Station_A).

Name	Cloud-to-cloud error	Coincident Points (%)
GH		
Station_A		
Station_B	8.12 mm	61.5%
Station_C	6.57 mm	34.9%
Station_D	3.91 mm	44.6%
Station_E	4.67 mm	51.4%
Station_B		
Station_A	8.12 mm	61.5%
Station_C	2.40 mm	57.5%
Station_D	4.25 mm	39.3%
Station_E	3.56 mm	51.9%
Overall cloud-to-cloud error: 4.30 mm		

1 - Cloud-to-Cloud Error from Station_A to Station_B

2 - Cloud-to-Cloud Error from Station_B to Station_A

- The amount of **Coincident Points** per pair (of registered stations) is in percentage. The percentage in a pair (of registered stations) is the same from one direction to the other (e.g. from Station_A to Station_B or from Station_B to Station_A).

Name	Cloud-to-cloud error	Coincident Points (%)
[-] * GH		
[-] Station_A		
Station_B	8.12 mm	61.5%
Station_C	6.57 mm	34.9%
Station_D	3.91 mm	44.6%
Station_E	4.67 mm	51.4%
[-] Station_B		
Station_A	8.12 mm	61.5%
Station_C	2.40 mm	57.5%
Station_D	4.25 mm	39.3%
Station_E	3.56 mm	51.9%
Overall cloud-to-cloud error: 4.30 mm		

1 - Common points from Station_A to Station_B | 2 - Common points from Station_B to Station_A

Note: An **Overall Cloud-to-Cloud Error** (from all the station errors) is displayed at the bottom left corner of the **Registration Report** dialog. This **Overall Cloud-to-Cloud Error** allows weighting of each station error with respect to their overlap percentage.

Saving in RTF

You can save the **Auto-Register Using Planes (Target-Less)** result in a report in **TXT** format.

To Save in RTF:

1. In the **Registration Report** dialog, enter a name for the report file in the **File Name** field.
2. Find a location where you want the report file to be stored.
3. Click **Save**. The **Registration Report** dialog closes.

Note: An **Overall Cloud-to-Cloud Error** (from all the station errors) is displayed at the beginning of the file, just before the list of stations. This **Overall Cloud-to-Cloud Error** enables to weight each station error with respect to their overlap percentage.

Options

There is one option that comes after the registration (of stations). If it has been chosen, the related process is then performed, otherwise nothing occurs.

Generating Preview Scans

The **Preview Scan** option enables to create a **Scan** by first getting points, not based on a **TZF Scan** but from its **Preview**, and by computing **Normals** on them. A **Scan** is always named **Preview**. The number of points for each is about two million points.

To Generate Preview Scans:

- Check the **Generate a Preview Scan** option.

Notes:

- If several stations have been selected as input, a set of **Scans** (one per station) are created in batch mode, one after the other.
- When you interrupt the **Generate a Preview Scan** step by pressing **Esc**, a dialog opens and prompts you to abort or not.

Notes:

- You may not see anything happen in the **3D View** if the option is not checked.
- The **Generate a Preview Scan** step is an optional step. If the **Generate a Preview Scan** option has been checked, you will prompt to save the current project in the **RealWorks** database, if it is not yet saved. If the option has been kept unchecked, no prompt appears.


Tip: When you create from several **TZF Scans** within a station, all **Sans** (in that station) have not the same color. Each has its own color.

Target-Based Registration Tool

The **Target-Based Registration Tool** allows you to register a set of stations by using targets. The targets could be those obtained while scanning, those created manually during a registration, or those obtained by using traditional surveying instruments such as **Total Stations**. The registration is based on a least-squares adjustment method using the corresponding target observations of each station. A registration report will be created after the registration. You can check the registration quality based on this report. If any of the targets are out of error tolerance, you can un-validate them and re-perform the registration.

Opening the Tool

To Open the Tool:

1. Select a station⁽¹⁾, a set of stations, a set of groups or a project⁽²⁾ from the **Scans Tree**.
2. In the **Registration** menu, select **Target-Based Registration Tool** .

Or

3. Perform an **Auto-Extract Target and Register**.

The **Target-Based Registration** dialog opens as the fourth tab of the **WorkSpace** window. It is sub-divided into three parts. Each corresponds to one step in the **Target-Based Registration** process.

Notes:

- ⁽¹⁾ Among other stations. Otherwise, if the station is alone in the project, the tool is grayed.
- ⁽²⁾ With a set of stations (or a set of groups). Otherwise, if the project has only a unique station (or group (of stations) within) the tool is grayed-out.

Notes:

- If the input is a group with a unique station (or a group with a set of stations) within, the tool is grayed.
- If the loaded project contains some scans of spherical target type which are not already fitted, **RealWorks** will prompt you to automatically fit each of them with a geometry.

Caution: You cannot open the **Target-Based Registration Tool** if the input is only of **Topographic Station** type.

Notes:

- ⁽¹⁾ When a single station has been selected, the whole project is then taken as the entry of the tool and the selected station becomes the **Reference Station**.
- ⁽¹⁾ You can also select two stations without **TZF Scan** inside.

Tip: You can also select **Target-Based Registration** from the pop-up menu.

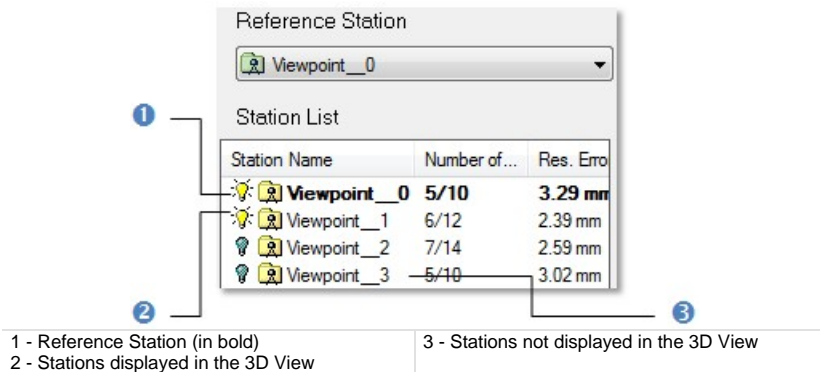
If you enter into the **Target-Based Registration Tool** with some stations that are already been registered and some not, a dialog appears and asks to register those that are not yet registered with those that are already.

Selecting a Reference Station

This step consists of fixing a station as **Reference Station**. The other station(s) is/are used to be registered with it. If a project (or a set of stations) has been selected as input, the first station (of the project) is the default **Reference Station**. If a leveled station has been chosen as input, this station is by default **Reference Station**. If one of the stations is a Topographic station, i.e. it contains points surveyed by using a traditional surveying instrument; this station is set by default as the **Reference Station**.

To Select a Reference Station:

1. Click on the pull down arrow of the **Reference Station** list.



By default, all selected stations are put in the **Station List** window. They are listed not by the order of selection but by their order (of creation). All of them are not displayed in the **3D View**. The **Reference Station** is in bold.

2. Select a station from the drop-down list.
3. Select and toggle the **Reference Station** to **On**. Its representation is shown in the **3D View**.
4. Select and toggle another station to **On**. Its representation is shown in the **3D View**.

Note: If there are several **Topographic Stations** within the project, only one is assigned as the **Reference Station**.

Caution: If the station selected as **Reference Station** is not a **Leveled Station**, an error message appears and prompts you to change the selection.


Caution: An error message appears if the selection (as input of the tool) contains a **Topographic Station** and this station has not been chosen as a **Reference Station**.

Auto-Pairing Targets

A target has two states: **Matched** or **Unmatched**. If there are already extracted targets within the input (of the tool) and if these targets have not yet been paired (**Unmatched**), all of them are gathered in the **Unmatched** folder in the **Targets Tree** and per station in the **Scans Tree**.

If there are enough targets inside each station (and at least three in common between two stations), the auto-pairing of targets will be performed automatically.

If the auto-pairing of targets succeeds, you may see the number of paired targets in the **Station List** of the **Target-Based Registration** dialog. For a given station, the **Number of Targets** is shown as X/Y. Y is the sum of the targets and X is the sum of the matched targets. The **Adjust** button (in **Step 3** of the **Target-Based Registration** dialog) is dimmed (because stations are automatically registered). The Registration Details dialog opens automatically with the **Station View** set by default.



Station Name	Number of Targets	Res. Error
TeachingF...	4/14	16.84 mm
TeachingFirs...	7/10	10.27 mm
TeachingFirs...	4/8	2.34 mm

1 - Reference Station (in bold)
 2 - Matched targets in a station
 3 - Total of targets (matched and unmatched) in a station

If there are not enough targets inside each station and/or if there are no common targets between stations, the auto-pairing of targets will then fail and the **Adjust** button (in **Step 3** of the **Target-Based Registration** dialog) is enabled. The **Registration Details** dialog is not open. The auto-pairing of targets can also fail even if there are not enough targets inside only in a station in common with other stations.

If the extracted targets have already paired (**Matched**), they are gathered per pairing group named **XXX** where **XXX** is its order. All pairing groups are rooted in the **Targets Tree**. The target pairing information is still displayed in the **Station List** as illustrated above. After entering into the tool, the stations are automatically adjusted. The **Adjust** button in **Step 3** is still enabled as the stations are automatically registered.

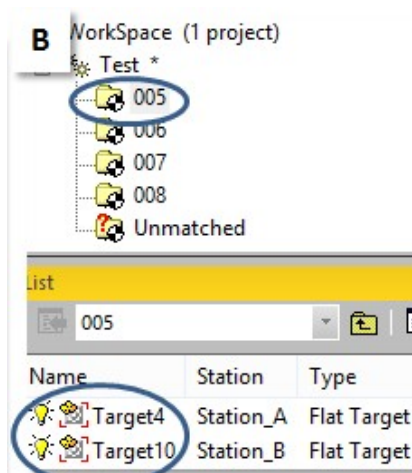
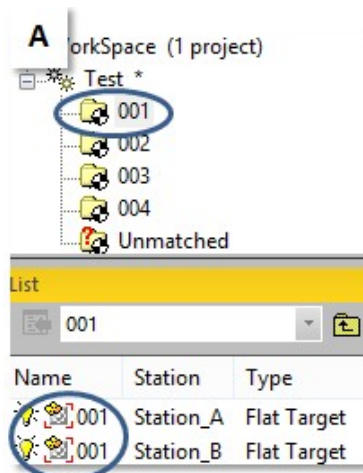
If there is no target within each station, the tool opens too. For each selected station, you may see the sum of targets **Y** and the sum of matched targets **X**, both equal to zero. An information box with the "Reference station is not registrable" text may appear.

Station List		
Station Name	Number of ...	Res. Error
Teaching...	0/0	0.00 mm
Teachi...	0/0	0.00 mm

Tip: In general, a station should have at least three targets inside. If one of the stations is a **Topographic Station** and the other a **Leveled Station**, two targets (per station) are enough.

Note: A pairing group (XXX where X is its order) is shown in the **3D View** with a label. The label's name is the group name and its color corresponds to the one that you can find in each of the targets matched together. Unmatched targets still remain in the **Unmatched** folder.

Tip: Targets, extracted by using the **Auto-Extract and Register** method and paired together immediately in the **Target-Based Registration Tool**, are renamed as well as their pairing groups. They are renamed as XXX. XXX starts at 001 and is incremented by one. See [A]. If the **Auto-Extract and Register** method is not combined with the **Target-Based Registration Tool**, paired targets are not renamed but only their pairing groups are. See [B].



Editing Targets

For each station, you can modify targets (either of spherical type or of planar type), delete those that are incorrectly fitted, and/or create additional targets in the point cloud where such a target is identified visually as having been scanned.

To Edit Targets.

- In Step 2, click **Analyze**. The Target Analyzer dialog opens.

Adjusting Stations

Once the pairing (of targets) is performed, the adjustment (of stations) is then performed automatically without user interaction. From this point on, stations are split into two categories: "Registered" and "Unregistered". Unregistered stations are those for which the targets inside are e.g. not in sufficient in quantity or not in common with other stations. They are then grayed out in the **Station List** and the **Residual Error** is equal to Zero. Registered stations are those for which the targets inside are in common and paired with other station(s). The **Residual Error** is not equal to Zero

Station Name	Number of Targets	Res. Error
TeachingFi...	5/14	14.46 mm
TeachingFirst...	7/10	10.41 mm
TeachingFirst...	4/8	2.18 mm
TeachingFirst...	0/2	0.00 mm
TeachingFirst...	3/9	5.78 mm

1 - Reference Station (in bold)
 2 - Registered station(s)
 3 - Unregistered station(s)

Caution: Changing the current **Reference Station** to a new one will NOT reset the adjustment information. The **Adjust** button, in **Step 3**, will stay grayed-out.

Note: After registering a **Leveled Station** with a **Topographic Station**, the **Up (Z direction)** of the **Leveled Station** is retained.

Checking the Adjustment

You can then check the quality of the adjustment. You can check the mean error for each target group (inside which you can find all matched observations of this target from different stations). You can also check the error for each target observation. To do this, you should select the corresponding line in the table, and all pertinent information will be shown in the dialog area below the table. If the error of a target is e.g. out of tolerance, you can select it and use the **Unmatch** button to remove it from the next registration. You can then re-perform the registration.

To Check the Adjustment:

- In **Step 3**, click on the **Check** button. The Registration Details dialog opens.

Registration Details

What is a **Fitting Error**? An extracted target is in fact a set of points fitted with a geometry. The accuracy of the fitting is given by this error (a distance value in the current unit of measurement). This distance is the deviation from the fitted geometry to the set of points. The shorter the distance, the more accurate the fitting.

As a target does not belong to only one station but to several stations and the fitting error (of this target) in a station differs from the fitting error in another station. The **Residual Error** of a target is the average of all **Fitting Errors** (of this target), each from a station observation.

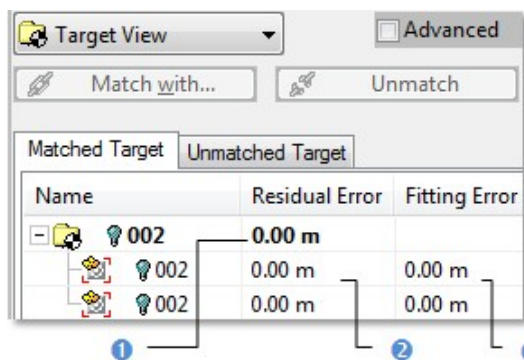
Name	Residual Error	Fitting Error
Teac...	0.00 m	
001	0.01 m	0.00 m
002	0.00 m	0.00 m
003	0.00 m	0.00 m

1 - Station's Residual Error
2 - Target's Residual Error

3 - Target's Fitting Error

A **Residual Error** of a station is the average of **Fitting Errors** of all targets (belonging to the station). The shorter the distance, the more accurate the matching of targets.

The **Overall Error** is the average of all station residual errors. This error is displayed in two places. First in **Step 1** of the **Target-Based Registration** dialog below the **Station List**. Second at the right side of the **Registration Details** dialog. The smaller the **Overall Error**, the more accurate the registration of the stations.



1 - Target group's Residual Error
2 - Target's Residual Error

3 - Target's Fitting Error

A **Target Group** is a group inside which you can find all matched observations of this target from different stations. The **Residual Error** in this case is the average of all **Fitting Errors** of this target.

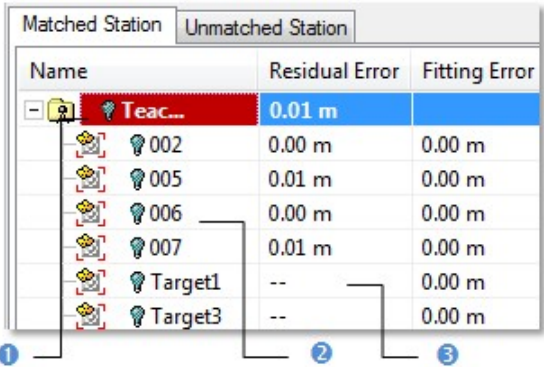
From Station View

To Check the Errors from the Station View:

1. Click on the pull-down arrow.
2. Select **Station View** from the drop-down list.

Matched Station Tab

The **Matched Station** tab lists in a table all registered stations with targets whether matched or unmatched. By default, all are **Off** (undisplayed in the **3D View**).



Name	Residual Error	Fitting Error
Teac...	0.01 m	0.00 m
002	0.00 m	0.00 m
005	0.01 m	0.00 m
006	0.00 m	0.00 m
007	0.01 m	0.00 m
Target1	--	0.00 m
Target3	--	0.00 m

1 - Registered station(s) 2 - Unmatched target(s) 3 - Matched target(s)

1. Do one of the following:
 - Toggle a registered station **On**. All targets of this station are **On** and have their representation displayed in the **3D View**.
 - Toggle a matched target **On**. Its representation is displayed in the **3D View**.
2. Do one of the following:
 - Unmatch a pair of matched targets.
 - a) Select a matched target. The **Unmatch** button becomes active.
 - b) Click **Unmatch**. This target and the one(s) in the same pair are unmatched.
 - Unmatch all matched targets in a registered station.
 - a) Select a registered station. The **Unmatch** button becomes active.
 - b) Click **Unmatch**. All targets from this station and the ones from the other registered stations are unmatched.
 - Match a matched target with.

Registered Stations

The panel below the table displays for a registered station its name, the number of station(s) it is linked to, the name of each linked station, the number of common targets and the **Mean Distance** (in the current unit of measurement).

Unmatched Targets

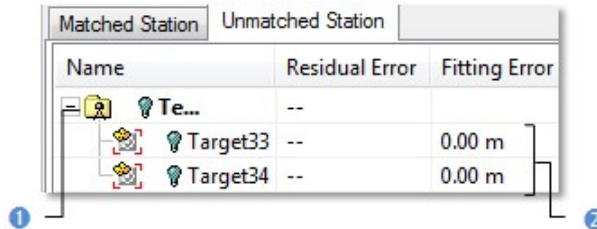
For an unmatched target, the panel displays its name and the station and group the matched target belonging to.

Matched Targets

For a matched target, the panel displays its name, the station and group it belonged to, the target(s) paired to it.

Unmatched Station Tab

The **Unmatched Station** tab lists all unregistered stations. They are only those that have been selected (as input) but for which the registration failed. There are some extracted target inside (if there is a **TZF** scan) the stations.



1 - Station selected as input of the registration

2 - Targets belonging to the unmatched station

- Do one of the following:
 - Toggle an unmatched target **On**. Its representation is shown in the **3D View**.
 - Toggle an unmatched station **On**. All targets inside this station have their representation displayed in the **3D View**.
 - Match a target with.

Notes:

- The **Unmatch** button is not available (dimmed) when selecting the **Unmatched Station** tab.
- The **Unmatched Station** tab is not present in the **Registration Details** dialog if all the selected stations have been successfully registered together.

Unregistered Stations

For an unregistered station, the panel below the table displays its name and the "0 linked station(s) text".

Unmatched Targets

For an unmatched target, the panel displays its name and the station it belongs to.

Match an unmatched target with.

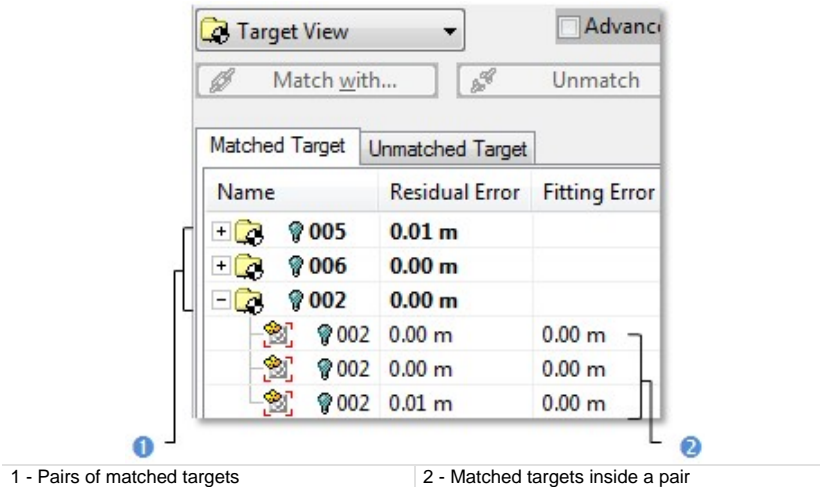
From Target View

To Check the Errors from the Target View:

1. Click on the pull-down arrow.
2. Select **Target View** from the drop-down list.

Matched Target Tab





The **Matched Target** tab lists in a table all pairs of matched targets. By default, all are **Off** (undisplayed in the **3D View**).



1. Do one of the following:
 - Toggle a pair of matched targets **On**. Both targets (one from each station) are displayed in the **3D View**.
 - Toggle a matched target **On**. Its representation is displayed in the **3D View**.
2. Do one of the following:
 - Unmatch a pair of matched targets.
 - a) Select a pair of matched targets. The **Unmatch** button becomes active.
 - b) Click **Unmatch**. This pair of targets is unmatched.
 - Unmatch a matched target.
 - a) Select a matched target. The **Unmatch** button becomes active.
 - b) Click **Unmatch**. This target and the one in the same pair are unmatched.
 - Match a target with.

Unmatched Target Tab

The **Unmatched Target** tab lists all unmatched targets. All are put in the **Unmatched** folder.

Matched Target		Unmatched Target	
Name	Residual Error	Fitting Error	
 Unmatched	--		
 Target1	--	0.00 m	
 Target3	--	0.00 m	
 Target5	--	0.00 m	

- Do one of the following.
 - Toggle an unmatched target **On**. Its representation is displayed in the **3D View**.
 - Toggle the **Unmatched** folder **On**. All unmatched targets inside this folder have their representation displayed in the **3D View**.
 - Match a target with.

Note: The **Unmatch** button is not available (dimmed) when selecting the **Unmatched Target** tab.

Auto-Match All

The **Auto-Match All** feature allows you to first un-adjust stations that had previously been adjusted and then adjust them again. No selection is required. Stations are those selected as the input of the **Target-Based Registration** (or **Auto-Extract Targets and Register**).

Note: The user can be in either the **Station View** or the **Target View**.

Auto-Match Station

The **Auto-Match Station** feature allows you to auto-adjust a selected station from the **Registration Details** dialog. If the selected station is already adjusted, it is then un-adjusted and adjusted again. If it is not already adjusted, it is then automatically adjusted. A selection is required. It must be done in the **Station View** from either the **Matched Station** tab or the **Unmatched Station** tab.

Saving the Adjustment Result

If you are satisfied with the adjustment result, you can use the **Apply** (or **Group**) button to save this result. You can continue to perform other registrations or to quit the tool by using the **Close** button.

Tip: **Close** can also be selected from the pop-up menu.

Applying

To Apply the Adjustment:

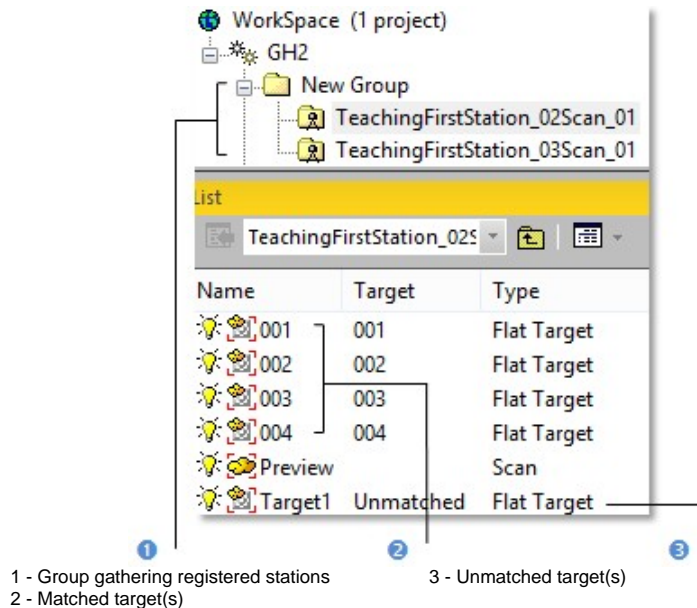
1. Click **Apply**. The **Target-Based Registration** dialog remains open. Paired targets remain paired. Stations are adjusted.
2. Click **Close**. The **Target-Based Registration** dialog closes.

Tip: You can perform two undo operations, one for the adjustment (of stations) and one for the auto-pairing (of targets).

Grouping

To Apply the Adjustment and Group the stations:

1. Click **Group**. A dialog opens and asks you to apply all changes before grouping stations.
2. Do one of the following:
 - Click **Yes**. The **Target-Based Registration** dialog closes. A new folder named **New Group** is created in the **Scans Tree**. All adjusted stations are put under that folder while all unadjusted stations are outside. Targets and target groups, instead of being named **TargetX** and **mTargetX** where **X** is an order, they are renamed as **XXX**. **XXX** starts at 001.



Or

- Click **No**. A new dialog opens and asks you to apply all changes to the database.
 - a) Click **Yes**. The **Target-Based Registration** dialog closes. Targets remain paired. Stations are adjusted and no new folder is created.

- b) Or click **No**. The **Target-Based Registration** dialog closes. Targets are unpaired. Each target keeps its default name: **TargetX** (where **X** is an order). Stations are not adjusted.

Note: You can perform two undo operations, one for both the grouping and the adjustment (of stations) and one for the auto-pairing (of targets).

Applying and Grouping

To Apply the Adjustment and Group the Stations:

1. Click **Apply**. The **Target-Based Registration** dialog remains open. Paired targets remain paired. Stations are adjusted.
2. Click **Group**. The **Target-Based Registration** dialog closes. A new folder named **New Group** is created in the **Scans Tree**. All adjusted stations are put under that folder while all unadjusted stations are outside.

Note: You can perform three undo operations, one for the station grouping, one for the adjustment (of stations) and one for the auto-pairing (of targets).

Refine Registration Using TZF Scans

When working with **TZF Scans**, this functionality refines the registration using the scan data. A target-less registration report is generated, showing residual errors and percentage of common points between matched stations. The report can be saved as an RTF file

Opening the Tool

To Open the Tool:

1. Select at least two stations, a group (or set of groups)⁽¹⁾, or a project⁽²⁾ created from **TZF** format file(s)⁽³⁾ from the **Scans Tree**.
2. Or select an **Autoregister Group** (for which the **Cloud Refinement** option has not been chosen).
3. From the **Registration** menu, select **Refine Registration Using TZF Scans**.
 - If a set of stations has been selected, the **Refine Registration Using TZF Scans** dialog opens.
 - If a project has been selected, a dialog opens and prompts you to process with all stations (or not). Click **Yes**. The dialog closes and the **Refine Registration Using TZF Scans** dialog appears.

Notes:

- ⁽¹⁾ With at least two stations inside a group. Otherwise, if there is only a unique station in a group, the tool is grayed-out.
- ⁽²⁾ With several stations, a unique group or a set of groups. Otherwise, if the project has only a unique station within, the tool is grayed-out.

Caution: If there is no **TZF** format file inside the selected station(s) (or group or project), a dialog opens and asks you if you wish to continue with the remaining station(s).

Reference Station

The **Reference Station** is the one whose position and orientation remain unchanged through the **Refine Registration Using TZF Scans** process. If a project (or a set of stations) has been selected as input, the first station (of the project) is the default **Reference Station**. If a leveled station has been chosen as input, this station is by default **Reference Station**.

To Choose a Reference Station:

1. Click on the **Selection List** pull-down arrow.
2. Choose a station from the drop-down list.
 - If a project has been selected as input, all stations (of the project) are in the **Selection List**.
 - If a set of stations has been selected as input, only the selected stations (of the set) are in the **Selection List**.

Caution:

- If the station selected as **Reference Station** is not leveled, an error message appears and prompts you to change the selection.
- If some of the selected stations are leveled and the **Reference Station** is not, an error message appears and prompts to set all stations to unlevelled.

Refining Registration

To Refine the Registration:

- Click on the **Start** button. Stations are then registered together*.
 1. If stations can be registered together, the **Registration Report** dialog opens.
 2. In the **Registration Report** dialog, do one of the following:
 - Click **Save In RTF**. The **Registration Report** dialog opens.

Or

- Click **Close**. The **Registration Report** dialog closes. An **Auto-Registered Group** is created and rooted under the **Scans Tree**.
- 3. If the refinement cannot be performed, an error dialog opens and suggests you try the **Cloud-Based Registration Tool**.

Note: (*) The user can abort the refinement in progress by clicking **Esc**. A dialog opens and prompts you to abort or not.

Registration Report

The **Registration Report** dialog lists for each station (of the selection) the following information:

- How many station(s) each of them is registered with and the name of each.
- The deviation in a pair registered stations in the current unit of measurement.

The purpose of the **Refine Registration Using TZF Scans** feature is to register stations based on the common parts of point clouds. The accuracy of two stations registered together is given by its **Cloud-to-Cloud Error**. This error is an average distance error of points present in the common parts of the two clouds. The **Cloud-to-Cloud Error** in a pair (of registered stations) is the same from one direction to the other (e.g. from Station_A to Station_B or from Station_B to Station_A).

Name	Cloud-to-cloud error	Coincident Points (%)
GH		
Station_A		
Station_B	8.12 mm	61.5%
Station_C	6.57 mm	34.9%
Station_D	3.91 mm	44.6%
Station_E	4.67 mm	51.4%
Station_B		
Station_A	8.12 mm	61.5%
Station_C	2.40 mm	57.5%
Station_D	4.25 mm	39.3%
Station_E	3.56 mm	51.9%
Overall cloud-to-cloud error: 4.30 mm		

1 - Cloud-to-Cloud Error from Station_A to Station_B

2 - Cloud-to-Cloud Error from Station_B to Station_A

- The amount of **Common Points** per pair (of registered stations) in percentage. The percentage in a pair (of registered stations) is the same from one direction to the other (e.g. from Station_A to Station_B or from Station_B to Station_A).

Name	Cloud-to-cloud error	Coincident Points (%)
GH		
[-] Station_A		
Station_B	8.12 mm	61.5%
Station_C	6.57 mm	34.9%
Station_D	3.91 mm	44.6%
Station_E	4.67 mm	51.4%
[-] Station_B		
Station_A	8.12 mm	61.5%
Station_C	2.40 mm	57.5%
Station_D	4.25 mm	39.3%
Station_E	3.56 mm	51.9%
Overall cloud-to-cloud error: 4.30 mm		

1 - Common points from Station_A to Station_B

2 - Common points from Station_B to Station_A


Note: An **Overall Cloud-to-Cloud Error** (from all the station errors) is displayed at the bottom left corner of the **Registration Report** dialog. This **Overall Cloud-to-Cloud Error** enables to weight each station error with respect to their overlap percentage.

Target Analyzer Tool

This tool helps you to analyze a project before you register the stations that are inside. For each station, you can check if there are enough targets (either of spherical type or of planar type), modify or delete those that are incorrectly fitted, and/or create additional targets in the point cloud where such a target is identified visually as having been scanned.

Opening the Tool

To Open the Tool:

1. Select a project (or a group of stations⁽¹⁾ or a single station⁽¹⁾) from the **Project Tree**.
2. In the **Registration** menu, select **Target Analyzer Tool** .

Or

3. In **Step 2** of the **Target-Based Registration** dialog, click **Analyze**.

The **Target Analyzer** dialog opens as the fourth (or fifth) tab of the **WorkSpace** window and is composed of five parts. The first part allows you to select a station for analyzing. The second part is to check targets/surveying points and scans that are in the selected station. The third part is to repair (or correct) a given target or to create a new one. The fourth part is to update the network⁽²⁾. The fifth part is to save the result, close the tool and give access to the online help. The number of scans and targets in the selection appear in text below the selection box.

Notes:

- If the input contains some scans of spherical target type which are not already fitted, **RealWorks** will prompt you to automatically fit each of them with a primitive.
- ⁽¹⁾ If the **TZF** format files have not yet been processed, the **Processing TZF Scans** dialog opens and prompts you to proceed to do so.
- ⁽²⁾ This part is only available when launching the **Target Analyzer Tool** through the **Target-Based Registration Tool**.

Tips:

- When a single station has been selected, the whole project is then taken as the entry of the tool.
- You can also right-select on a project (or a group of stations⁽¹⁾ or a single station⁽¹⁾) from the **Project Tree** and **Target Analyzer Tool** from the pop-up menu.

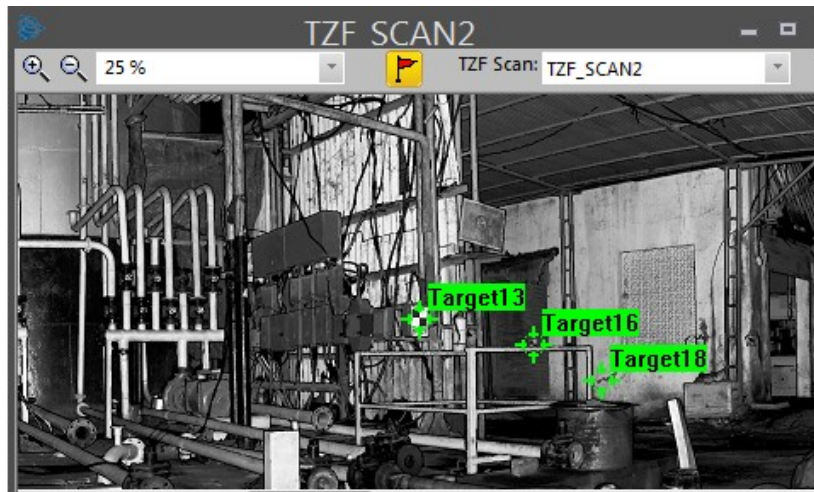
Selecting a Station

If a set of stations has been selected as the input (of the tool), no matter in which order has been selected each station, the first from the list is the one that is displayed in **Step 1**. The same rule is applied when selecting a project.

To Select a Station for Analyzing:

1. Click on the pull down arrow.
2. Select a station from the drop-down list.
3. Or click **Go to Next Station** (or **Go to Previous Station**).

For each station, the number of scans and the number of targets are displayed as well as the **Residual Error** (in the current unit of measurement). If the station has not yet been registered, its **Residual Error** is equal to Zero. By default, the **Main Scan** within the selected station is displayed in a **2D Viewer** as a **2D Preview Image**.



You can zoom an area of this **2D Preview Image In** or **Out** using the **Zoom In** and **Zoom Out** commands, zoom the whole image **In** or **Out** using the mouse wheel or by defining a zoom factor. If the image is zoomed **In** more than the **2D Viewer** can display, you can pan it in any direction in order to view the hidden areas.

Spherical Targets, Black and White Flat Targets or Point Targets extracted from a TZF Scan by using e.g. the Auto-Extract Targets and Register feature, once created, are displayed within the TZF Scan. You can display (or hide) all labels by clicking on the Show/Hide Labels icon.

Note: The 2D Viewer will not appear anymore when there is no TZF Scan within the input (of the tool).

Caution: If there is no TZF format file inside the selected station, a dialog opens and warns you that the TZF format file cannot be opened. It may be absent, corrupted or blocked. The 2D Viewer disappears after closing the warning dialog.

Selecting a TZF Scan

As a station can contain more than one TZF Scan, you can manually choose to display the one you want other than the Main Scan.

To Select a TZF Scan:

1. In the 2D Viewer, click on the TZF Scan pull-down arrow.
2. Choose a TZF Scan to display.

Tip: If a TZF Scan has been chosen (as input of the Target Analyzer Tool), it is then displayed in the 2D Viewer instead of the Main Scan.

Focusing on Targets

Targets and scans of the station selected in **Step 1** are listed according to the category they belong to. A target can be either of spherical shape or of flat shape. It can also be a surveying point. Only fitted targets can be used for registration (see the **Target-Based Registration Tool** for full details). Fitted targets are put together in the **Fitted** list and this list is accessed by selecting its corresponding tab.

Similarly, unfitted targets and scans are respectively in the **Not fitted** and **Scan** lists. The first item of the **Fitted** list is shown in the **3D View** but none is selected. The information box at the top right corner of the **3D View**, which is here to display the selected item, is blank. Both the **Go to Next Target** and the **Go to Previous Target** in the dialog are dimmed.

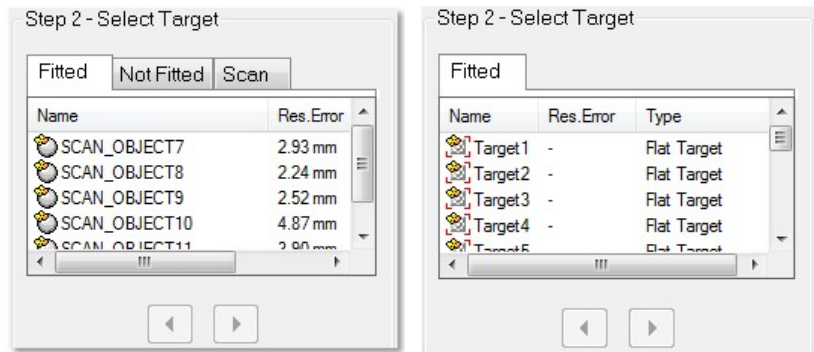
Properties depend on where the selected item is. If the selected item belongs to the **Scan** and **Not Fitted** lists, its name and number of points are listed in the information box. If the selected item comes from the **Fitted** list, you will find two other items of information (**Standard Deviation** (also called RMS error) and **Scanner Distance**) in addition to its name and number of points. If the selected item is from a leveled station, you will also see the **Target Height** and **Scanner Up Direction** information.

To Focus on a Target:

1. If the station (selected in **Step 1**) has no TZF scan inside, **Step 2** looks as shown in [A].
2. If the station (selected in **Step 1**) has a TZF scan inside, **Step 2** looks as shown in [B].
3. Select an item from the current list. Both the **Go to Next Target** and the **Go to Previous Target** buttons become enabled.
4. Click **Go to Next Target** (or **Go to Previous Target**) to navigate through the list (of target).
5. Or press **Down** or **Up** on your keyboard.

[A]

[B]



A **Target** selected from **Step 2** is highlighted in the **TZF Scan** and centered on the **3D View** and on the **2D Viewer** as shown below.



Notes:

- You can only view the target height in the **3D View** if its absolute value is greater than zero.
- You can first select any item from the current list and use **Page Up** and **Page Down**. The first and last item of this list becomes consecutively selected and its representation is shown in the **3D View**.

Creating/Editing Targets

If the station selected in **Step 1** contains already fitted items, you can focus on each item of the **Fitted** list from the first to the last. Visually compare each of the selected item's representations (points and geometry) in the **3D View** and if required check the **RMS Error** value in the information box. The smaller this value, the more precise the fitting. Those that are not correctly fitted can be modified or deleted. **Step 3** (of the **Target Analyzer** dialog) appears as shown in [A], [B] and [C] when selecting respectively a spherical item, a flat target and a survey point.



If already fitted items are not sufficient, you can create additional items with the **Fitting Tool**. You should first select an item from one of the two lists (**Not Fitted** and **Scans**). If the selected item is from the **Not Fitted** list and is of spherical shape (or flat shape (or survey point)), the dialog appears as shown in [D], (or [E] (or [F])).



If the selected item is from the **Scans** list, the dialog looks as shown in [H]. From each item of the **Scans** list, you can extract a 3D point as in the **3D Point Creation Tool**.



Those, that are not correctly fitted and that belong to a leveled station (or from a survey instrument), can also be modified and deleted. If the selected item is from the **Fitted** list, you can edit its height (see [H] when selecting a flat item). If it is either from the **Not Fitted** list or from the **Scan** list, you cannot edit any height as the **Target Height** is grayed out (see [I] when selecting a flat item).



If the station selected in **Step 1** contains fitted items extracted from a TZF scan, the dialog looks as shown in [J] (for an un-leveled station) and [K] (for a leveled station).

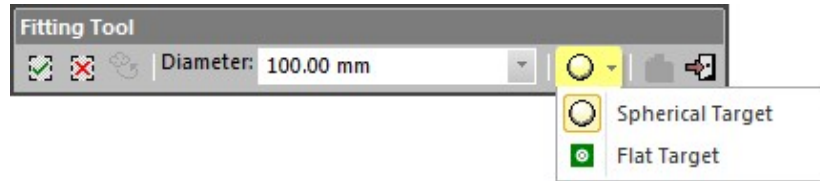


Note: Please, be aware that the **Re-fit** and **Fit** buttons are replaced by the following icon

Fitting Geometry to Cloud [From Scan Items]

To Fit a Geometry to Point Cloud:

1. Select a scan from the **Scan** list.
2. Click **Fit**. The **Fitting Tool** toolbar appears.



3. Fence a set of points by drawing a polygon.
4. Click on the **Fit Geometry to Cloud** pull-down arrow.
5. Do one of the following:
 - Fit with a **Spherical Target**.
 - a) If required, select **Spherical Target** from the drop-down list.
 - b) Click on the **Diameter** pull-down arrow.
 - c) Select **AUTO** to do a free fitting.
 - d) Or key a diameter value in the **Diameter** field*.
 - e) Or select 100 mm (one the standard Trimble white **Spherical Target** diameters) to perform constrained fitting.
 - f) In the **Fitting Tool** toolbar, click again **Spherical Target**. A **Spherical Target** is fitted to the fenced points.
 - Fit with a **Flat Target**.
 - a) Select **Flat Target** from the drop-down list.
 - b) In the **Fitting Tool** toolbar, click **Flat Target** again. A **Flat Target** is fitted to the fenced points.
6. Click **Create Fitted Geometry**.
7. Click **Close Tool**.

Notes:

- You can also right-click anywhere in the **3D View** and select a command from the pop-up menu.
- The value entered in the **Diameter** field will no longer be kept. If you close the **Fitting Tool** without creating the fitted geometry.

A set of points, once fitted, is put with its geometry in the **Fitted** list and under the **Unmatched** folder of the **Targets Tree** and under the active group of the **Scans Tree**. Undoing the fitting removes the set of points with its primitive from the **Fitted** list, the **Unmatched** folder and the active group.

Note: Please, be aware that the **Flat Target** icon looks like this .


Creating 3D Points

To Create a 3D Point:

1. Select a scan from the **Scan** list.
2. Click **Pick Point to Create 3D Point**  icon. The **Picking Parameters** toolbar appears in 3D constraint mode and the cursor becomes a cross.
3. Pick a point on the point cloud displayed in the **3D View**. A **3D Point** whose name is **Scan_ObjectX** where **X** is its order is created. This **3D Point** which is an unmatched target is put the current station in the **Scans Tree** and in the **Unmatched** folder in the **Targets Tree**.

Deleting Targets

To Delete a Target:

1. Select a target from the **Fitted** list.
2. Click **Delete the Selected Target** . A warning dialog appears.
3. Do one of the following:
 - To delete both the geometry and the points, click **Delete Scan and Target**.
 - To delete only the geometry, click **Delete Target Only**.
 - To cancel, click **Cancel**.

Caution: The deletion is definitive. You cannot undo.

Editing the Target Height

To Edit the Target Height:

1. Select a target from the **Fitted** list.
2. Enter a value in the **Target Height** field.
3. Or keep the default value.
4. Type **Enter**.

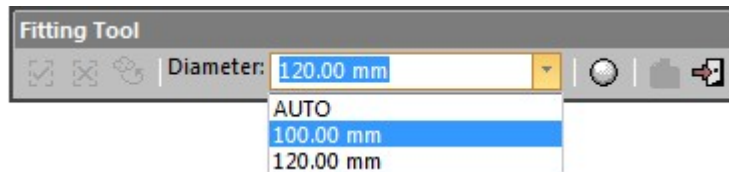
Notes:

- The selected target needs to belong to a leveled station.
- Instead of editing the **Target Height** value from the **Target Analyzer** dialog, you can also do so in the **Property** window.

Fitting Geometry to Cloud [From Unfitted Items]

To Fit a Geometry to Point Cloud:

1. Select an unfitted item from the **Not fitted** list.
2. Click **Fit**. The **Fitting Tool** toolbar appears.
3. If the selected item is of spherical type, the **Fitting Tool** toolbar appears as shown below.



4. If the selected item is of flat type, the **Fitting Tool** toolbar appears as shown below.



5. If required, fence the target by drawing a polygon.
6. Do one of the following:
 - If the target is of **Spherical Target** type.
 - a) Click on the **Diameter** pull-down arrow.
 - b) Select **AUTO** to perform a free fitting.
 - c) Or key a diameter value in the **Diameter** field.
 - d) Or select 100 mm (one of the standard Trimble white **Spherical Target** diameters) to perform a constrained fitting.
 - e) In the **Fitting Tool** toolbar, click again **Spherical Target**.
 - If the target is of **Flat Target** type.
 - a) In the **Fitting Tool** toolbar, click again **Flat Target**
7. Click **Create Fitted Geometry**.
8. Click **Close Tool**.

Notes:

- You can also right-click anywhere in the **3D View** and select a command from the pop-up menu.

- The value entered in the **Diameter** field will no longer be kept if you close the **Fitting Tool** without creating the fitted geometry.

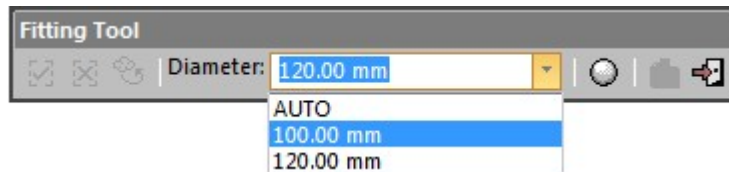
A target scan, once fitted, is removed from the **Not Fitted** list and put in the **Fitted** list and under the **Unmatched** folder in the **Targets Tree**. Undoing the fitting replaces the target scan again in the **Not Fitted** list and removes it from the **Unmatched** folder.

Note: Please, be aware that the **Flat Target** icon looks like this .

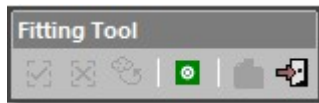
Re-Fitting Geometry to Cloud [From Fitted Items]

To Re-Fit a Geometry to Point Cloud:

1. Select a fitted item from the **Fitted** list.
2. Click **Re-Fit**. The **Fitting Tool** toolbar appears. The geometry representation of the selected item is then hidden.
3. If the selected item is of spherical type, the **Fitting Tool** toolbar appears as shown below.




4. If the selected item is of flat type, the **Fitting Tool** toolbar appears as shown below.



5. If required, fence the target by drawing a polygon*.
6. Do one of the following:
 - If the target is of **Spherical Target** type.
 - a) Click on the **Diameter** pull-down arrow.
 - b) Select **AUTO** to perform a free fitting.
 - c) Or enter a diameter value in the **Diameter** field.
 - d) Or select 100 mm (one of the standard Trimble **Spherical Target** diameters) to perform a constrained fitting.
 - e) In the **Fitting Tool** toolbar, click **Spherical Target** again.
 - If the target is of **Flat Target** type.
 - a) In the **Fitting Tool** toolbar, click **Flat Target** again.
7. Click **Create Fitted Geometry**.
8. Click **Close Tool**.

Notes:


- You can also right-click anywhere in the **3D View** and select a command from the pop-up menu.

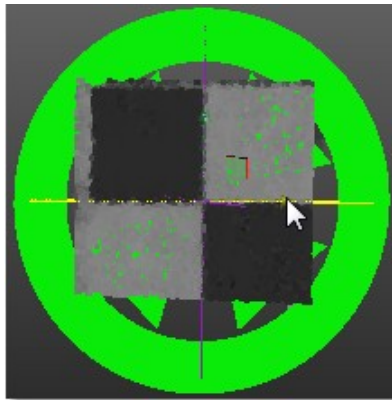
- The value entered in the **Diameter** field will no longer be kept if you close the **Fitting Tool** without creating the fitted geometry.
- (*) If some points have been removed from the point cloud when fencing, the **Residual Error** of the fitted item changes as well as the **RMS**, **Standard Deviation** and **Number of Points**.
- Please, be aware that the **Flat Target** icon looks like this .

Modifying Target Position

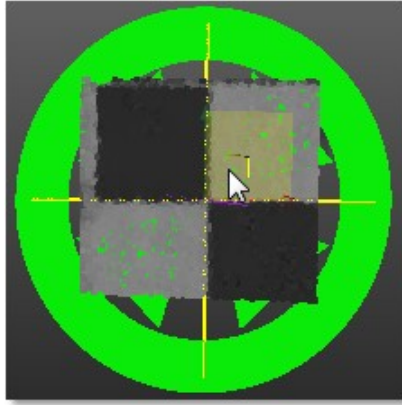
You can adjust the extracted target (of flat type) so that it fits exactly the points of the scanned target. A **Manipulator** (with two axis handles and a plane) appears. You can adjust the fitted geometry by moving it with to the manipulator. You can pan the fitted geometry along a direction or in the plane.

To Modify the Target Position:

1. Select a fitted item (of flat type) from the **Fitted** list.
2. Click **Show Manipulators to Modify Target Position** . A **Manipulator** (with two axis handles and a plane) appears.
3. Click on a handle; it turns to yellow. The direction along which you can displace the geometry is highlighted in yellow and the one along which you cannot displace it is in magenta.
4. Move the fitted geometry along that direction.



5. Click on the translucent plane. It turns to yellow. The plane in which you can displace the fitted geometry turns to yellow.
6. Move the created target in that plane.



Extracting

The **Extract** feature allows the user to extract **Spherical Targets** and/or **Black and White Flat Targets** and/or **Point Targets** from **TZF** scans and use the extracted targets to register the stations they belong to.

To Extract:

1. In **Step 1**, select a station from the station list.
2. In **Step 3**, click **Extract**. The **Target Creator Tool** dialog opens.

Note: The extract method, which appears in the **Target Creator Tool**, is the last used one.

Spherical Targets

To Extract Spherical Targets:

1. If required click on the pull-down arrow.
2. Choose **Spherical Target** as object type. The **Target Creator Tool** Toolbar appears as shown below.



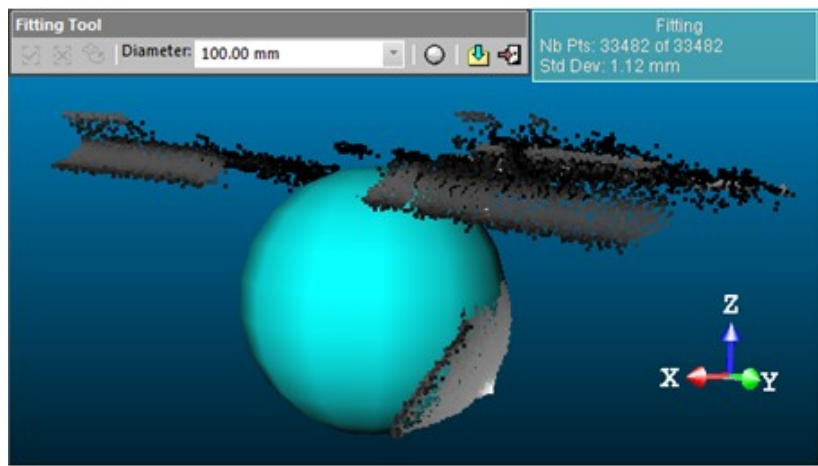
Note: There are four predefined diameters: 76.20 mm, 100 mm, 139 mm and 200 mm.

Picking

To Auto-Extract a Spherical Target:

1. Click on the **Auto-Extract Target** button.
2. Pick a point in the **Input Data 2D Viewer**.

If the target extraction succeeds; points of the created scan with a fitted geometry are displayed in the **3D View** and the **Fitting Tool** toolbar opens as shown below. A scan (named **TargetX**) is created and put under the current station.



An information box at the top right corner of the **3D View** displays the number of points in the created scan as well as the **Standard Deviation** information (except when the extraction fails). The **Delete Last** button in the dialog becomes enabled.

Note: An error dialog opens when **RealWorks** cannot find a **Spherical Target** close to the picked point.

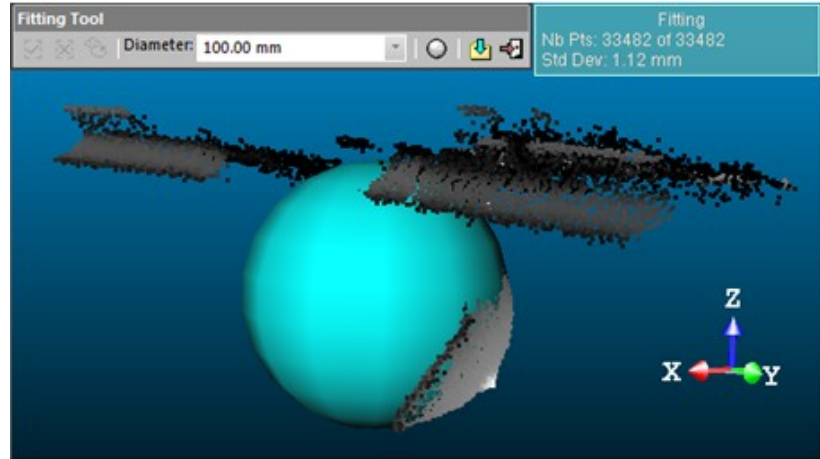
Fencing

You need to define an area from which a target will be created. This area is to be defined on the 2D image data in the 2D viewer.

To Select a Zone:

1. Click the **Polygonal Selection**  icon.
2. Pan (or zoom **In** or **Out**) the 2D image data (if needed).
3. Draw a polygonal fence by picking and double-click to close the polygon.

If the target extraction succeeds; points of the created scan with a fitted geometry are displayed in the **3D View** and the **Fitting Tool** toolbar opens as shown below. A scan (named **TargetX**) is created and put under the current station.




An information box at the top right corner of the **3D View** displays the number of points in the created scan as well as the **Standard Deviation** information (except when the extraction fails). If the area contains no points; nothing occurs.

Tips:


- Instead of double-clicking, press on the **Space Bar** of your keyboard.
- Press **Esc** (or select **New Fence** or **Close Polygon Tool** from the pop-up menu) to undo the polygonal fence in progress.

Creating

To Create the Fitted Geometry:


- In the **Fitting Tool** toolbar, click the **Create Fitted Geometry**  icon [A].

Or



- In the **Fitting Tool** toolbar, first click the **Spherical Target**  icon.
- And then, click the **Create Fitted Geometry**  icon [B].

The created scan, whose name is **TargetX**, is of **Spherical Target** type. It is displayed in the **Input Data 2D Viewer** and **3D View**. The **Fitting Tool** toolbar closes on its own. This **Spherical Target** is assigned as "Unmatched" and put in the **Unmatched** folder in the **Targets Tree**.

In the [A] method, the **Spherical Target** diameter is not constrained but results from the fencing of points (or auto-extracting). In the [B] method, the **Spherical Target** diameter is constrained to the value in the **Diameter** field in the dialog (or **Diameter** field in the **Fitting Tool** toolbar).

Properties	
<input checked="" type="checkbox"/> General	
Type	Spherical Target
Name	Target12
Number of Points	4 322
Number of Points Loaded	4 322
Color of Cloud	 RGB(198,217,241)

Properties of a scan (of Spherical Target type)

Properties	
<input checked="" type="checkbox"/> General	
Type	Spherical Target
Name	Target12
Number of Points	4 322
Number of Points Loaded	4 322
Color of Cloud	 RGB(198,217,241)
Standard Deviation	2.04 mm
<input checked="" type="checkbox"/> Geometry	
Color of Geometry	 RGB(198,217,241)
Center	-2773.34 mm; 3561.96 mm; 99.18 mm
Radius	50.00 mm
Direction of Axis	0.00; 0.00; 1.00

Properties of a scan (of Spherical Target type) fitted with a geometry

Note: A dialog appears if the user decides to close the **Fitting Tool** without creating the fitted geometry.

Black and White Flat Targets


To Extract Black and White Flat Targets:

1. If required click on the pull-down arrow.
2. Choose **Black and White Flat Target** as object type. The **Target Creator Tool** Toolbar looks as shown below.

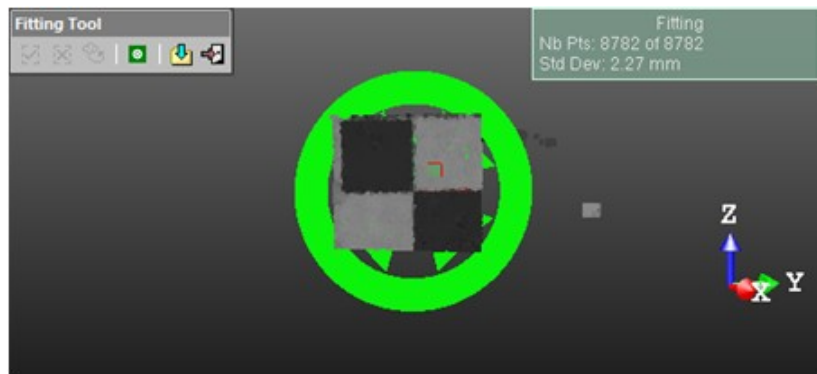


Picking

To Auto-Extract a Black and White Flat Target:

1. Click the **Auto-Extract Target**  icon.
2. Pick a point in the **Input Data 2D Viewer**.

If the target extraction succeeds; points of the created scan with a fitted geometry are displayed in the **3D View** and the **Fitting Tool** toolbar opens as shown below. A scan (named **TargetX**) is created and put under the current station.




An information box at the top right corner of the **3D View** displays the number of points in the created scan as well as the **Standard Deviation** information (except when the extraction fails).

Note: An error dialog opens when **RealWorks** cannot find a **Black and White Flat Target** close to the picked point.

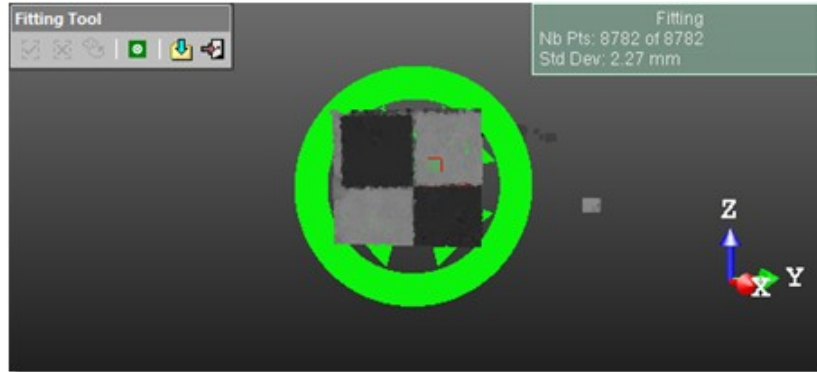
Fencing

You need to define an area from which a target will be created. This area is to be defined on the 2D image data in the 2D viewer.

To Select a Zone:

1. Click the **Polygonal Selection**  icon.
2. Pan (or zoom **In** or **Out**) the 2D image data (if needed).
3. Draw a polygonal fence by picking and double-click to end.

If the target extraction succeeds; points of the created scan with a fitted geometry are displayed in the **3D View** and the **Fitting Tool** toolbar opens as shown below. A scan (named **TargetX**) is created and put under the current station.




An information box at the top right corner of the **3D View** displays the number of points in the created scan as well as the **Standard Deviation** information (except when the extraction fails).

Tips:


- Instead of double-clicking, press on the **Space Bar** of your keyboard.
- Press **Esc** (or select **New Fence** or **Close Polygon Tool** from the pop-up menu) to undo the polygonal fence in progress.

Creating


To Create the Fitted Geometry:

- In the **Fitting Tool** toolbar, click the **Create Fitted Geometry**  icon.



Or

- In the **Fitting Tool** toolbar, first click the **Flat Target**  icon.
- And then, click the **Create Fitted Geometry**  icon.

The created scan, whose name is **TargetX**, is of **Flat Target** type. It is displayed in the **Input Data 2D Viewer** and **3D View**. The **Fitting Tool** toolbar closes on its own. This **Flat Target** is assigned as "**Unmatched**" and put in the **Unmatched** folder in the **Targets Tree**.

Properties	
<input checked="" type="checkbox"/> General	
Type	Flat Target
Name	Target30
Number of Points	8 782
Number of Points Loaded	8 782
Color of Cloud	 RGB(198,217,241)

Properties of a scan (of Black and White Flat Target type)

Properties	
<input checked="" type="checkbox"/> General	
Type	Flat Target
Name	Target30
Number of Points	8 782
Number of Points Loaded	8 782
Color of Cloud	 RGB(198,217,241)
Standard Deviation	2.27 mm
<input checked="" type="checkbox"/> Geometry	
Color of Geometry	 RGB(198,217,241)
Center	-3350.23 mm; 2317.73 mm; -664.18 mm
Direction of Normal	0.96; -0.28; 0.04

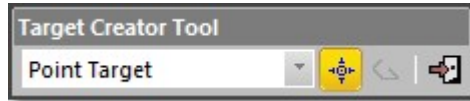
Properties of a scan (of Black and White Flat Target type) fitted with a geometry

Note: A dialog appears if the user decides to close the **Fitting Tool** without creating the fitted geometry.

Point Targets


To Extract Point Targets:

1. If required click on the pull-down arrow.
2. Choose **Point Target** as object type. The **Target Creator Tool** Toolbar looks as shown below.

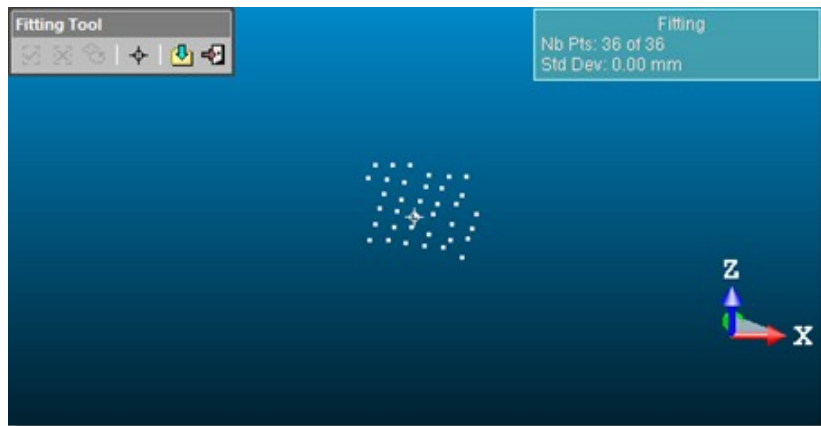


Picking

To Auto-Extract a Point Target:

1. Click the **Auto-Extract Target**  icon.
2. Pick a point in the **Input Data 2D Viewer**.

If the target extraction succeeds; points of the created scan with a fitted geometry are displayed in the **3D View** and the **Fitting Tool** toolbar opens as shown below. A scan (named **TargetX**) is created and put under the current station.




An information box at the top right corner of the **3D View** displays the number of points in the created scan as well as the **Standard Deviation** information. The **Delete Last** button in the dialog becomes enabled.

Note: An error dialog opens when **RealWorks** cannot find a **Point Target** close to the picked point.

Creating

To Create the Fitted Geometry:

- In the **Fitting Tool** toolbar, click the **Create Fitted Geometry**  icon.

The created scan, whose name is **TargetX**, is of **Survey Point** type. It is displayed in the **Input Data 2D Viewer** and in the **3D View**. The **Fitting Tool** toolbar closes on its own. This **Survey Point** is assigned as "Unmatched" and put in the **Unmatched** folder in the **Targets Tree**.

Properties	
<input checked="" type="checkbox"/> General	
Type	Survey Point
Name	Target11
Number of Points	36
Number of Points Loaded	36
Color of Cloud	<input type="checkbox"/> RGB(255,255,255)

Properties of a scan (of Point Target type)

Properties	
<input checked="" type="checkbox"/> General	
Type	Survey Point
Name	Target11
Number of Points	36
Number of Points Loaded	36
Color of Cloud	<input type="checkbox"/> RGB(255,255,255)
Standard Deviation	0.00 mm
<input checked="" type="checkbox"/> Geometry	
Color of Geometry	<input type="checkbox"/> RGB(255,255,255)
Center	-3348.86 mm; 2313.72 mm; -662.71 mm

Properties of a scan (of Point Target type) fitted with a geometry

Note: A dialog appears if the user decides to close the **Fitting Tool** without creating the fitted geometry.

Updating Network

This step, which only appears when using the **Target Analyzer Tool** within the **Target-Based Registration Tool**, allows the user to redo the adjustment of the stations after modifying the extracted targets e.g. refitting, deleting, etc.

Applying the Result

Once you are satisfied with the result, you can select another station and perform the same operations. When all selected stations are analyzed, you can save all results by using the **Apply** button. You can then evoke the **Target-Based Registration Tool** for registering stations together.

Note: The Projection Mode in use by default (in the **Target Analyzer Tool**) is **Perspective**. If you are in **Isometric** (before entering the tool), the projection mode automatically switches to **Perspective**. Once the tool is closed, the projection mode is restored.


Cloud-Based Registration Tool

The purpose of this tool is to register two selected scanning stations (or two station groups). The basic idea behind this tool is that user should pick three corresponding point pairs from both the point clouds to initialize the registration. Then the software can refine this registration by using the common parts of the two point clouds. The registration error will be shown as an average distance between the two point clouds. User can also check the registration results visually by using the **Cutting Plane Tool**. The **Cloud-Based Registration Tool** is available only in the **Registration** mode. In order to use this tool, you should select at least a set of two stations from the **Scans Tree**.

Opening the Tool

You should select two objects from a single project in the **Scans Tree**. You have the choice between three combinations: two groups of stations, two stations or one group of stations and one station. You cannot select two projects.

To Open the Tool:

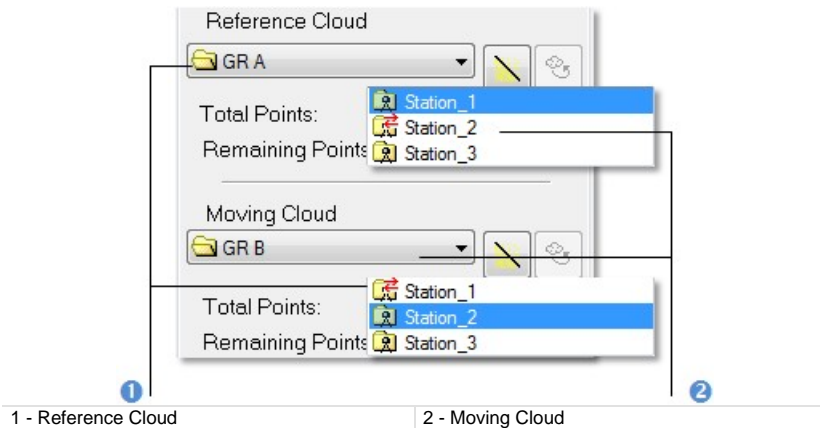
1. Select two different stations from the **Scans Tree**.
2. From the **Registration** menu, select **Cloud-Based Registration Tool** . The **Cloud-Based Registration** dialog opens.

Tips:

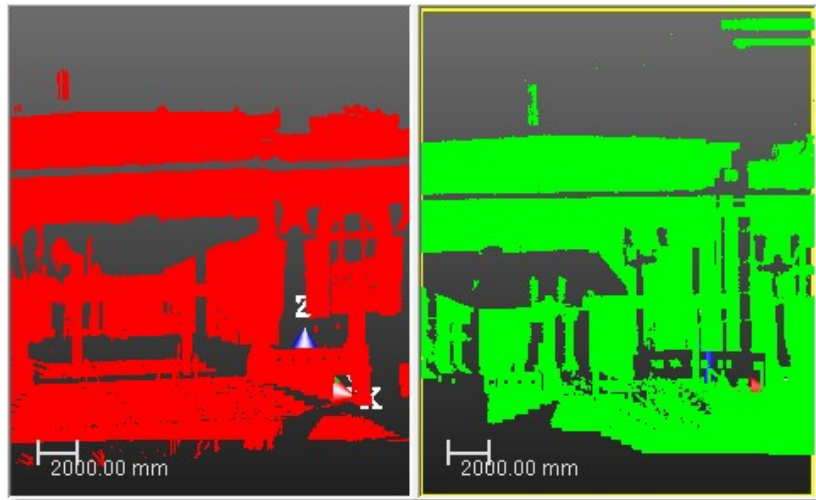
- The **Cloud-Based Registration Tool** icon can also be selected from the **Tools** toolbar.
- The input (of the selection) can be also a single station (with stations inside).

Reference Cloud and Moving Cloud

The first selected station becomes a **Reference Cloud**. Its name is highlighted in the **Reference Cloud** field. The second selected station is a **Moving Cloud**. Its name is displayed in the **Moving Cloud** field. The **Total Points** and the **Remaining Points** of each are displayed in the dialog. The **Reference Cloud** (or **Moving Cloud**) has a specific representation which can be seen when dropping down the **Reference Cloud** (or **Moving Cloud**) list.



The **3D View** is split into two sub-views. The left sub-view displays the **Reference Cloud** in red and the right sub-view the **Moving Cloud** in green. Only one sub-view can be active at once; it's the one with a yellow frame. As the default layout is two sub-viewers; you can use the **View Manager** to display in one full view window or to switch from one sub-view to another.



Because the refinement of the registration is based on the common parts of the two selected stations (or groups), you can use the **Segmentation Tool** to pre-select these common parts or the **Sampling Tool** to simplify the clouds for registration refinement. The **Create** command for these two tools is disabled. This means that you cannot save the result.

- For either the **Reference Cloud** or the **Moving Cloud**, click on the **Segmentation** (or **Sampling**) icon,
- Inside each sub-view, select **Sampling Reference Points** (or **Sampling Moving Points**) from the pop-up menu.

Notes:

- Sampling (or segmenting) the **Reference Cloud** (or **Moving Cloud**) will update the **Remaining Points** (of the **Reference Cloud** (or **Moving Cloud**)).
- When a group of stations has been selected as input, all scans of the group are displayed. The **Total Points** is the sum of all points of all scans.

Selecting

To Select the Reference and Moving Clouds:

1. Click on the **Reference Cloud** pull-down arrow.
2. Select a station from the **Reference Cloud** list.
3. Click on the **Moving Cloud** pull-down arrow
4. Select a station from the **Moving Cloud** list.

Note: The **3 Points** button in **Step 2** of **Cloud-Based Registration** dialog is enabled after opening the **Cloud-Based Registration Tool**. This means that you can start selecting points for the registration instead of sampling (or segmenting).

Changing

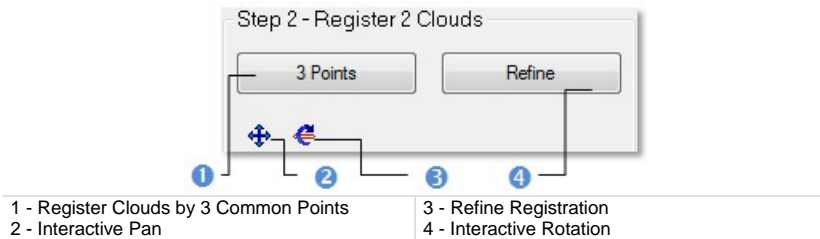
You can change your selection in the **Reference Cloud** and **Moving Cloud** fields without leaving the **Cloud-Based Registration Tool** if your project (or group) contains more than two stations (or group of stations). You only need to drop down the selection list in each combo box. If you want to change your selection by one that does not belong to your current project (or group), you should leave first the **Cloud-Based Registration Tool**, select the stations (or group of stations) and open again the tool.

To Change the Reference and Moving Clouds:

1. Drop-down the **Reference Cloud** list and select a new station (or group of stations).
2. Drop-down the **Moving Cloud** list and select a new station (or group of stations).

Displacing the Moving Cloud


Before picking points; you can use the **Interactive Pan** and **Interactive Rotation** to e.g. orientate the **Moving Cloud** in the direction of the **Reference Cloud** so that both have the same viewing direction or to place the **Moving Cloud** close to **Reference Cloud** if both are far from each other.

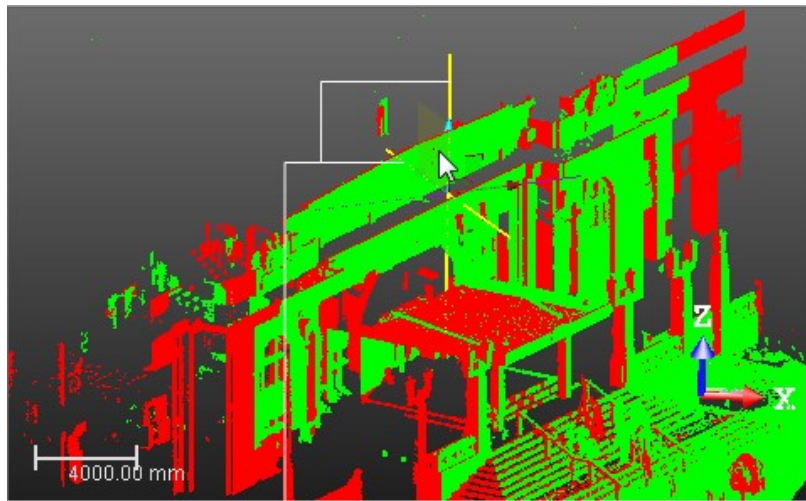


Tip: You can switch from **Interactive Pan** to **Interactive Rotation** and vice versa by clicking on its related button or by selecting its related command from the pop-up menu.

Interactive Pan

To Pan the Moving Station:


1. Click **Interactive Pan** . The two sub-views are merged into one. The cloud of each station remains with its own color (red for the **Reference Cloud** and green for the **Moving Cloud**). A **Manipulator** appears. It is composed of (three **Axis Handles** corresponding to three secant directions, each with a color (red, green and blue) and has as origin the **Moving Cloud's** center. In addition to the three **Axis Handles**, the user can find three **Plane Handles**. With that **Manipulator**, you can pan the **Moving Cloud** along a direction or in a plane.
2. Click on an **Axis Handle**; it turns to yellow. The direction along which you can displace the **Moving Cloud** is highlighted in yellow and those along which you cannot displace the **Moving Cloud** are in mauve.
3. Move the **Moving Cloud** along that direction.
4. Click on a **Plane Handle**. It turns to yellow. The plane in which you can displace the **Moving Cloud** appears in highlighted in yellow and those you cannot displace along are in mauve.
5. Move the **Moving Cloud** in that plane.

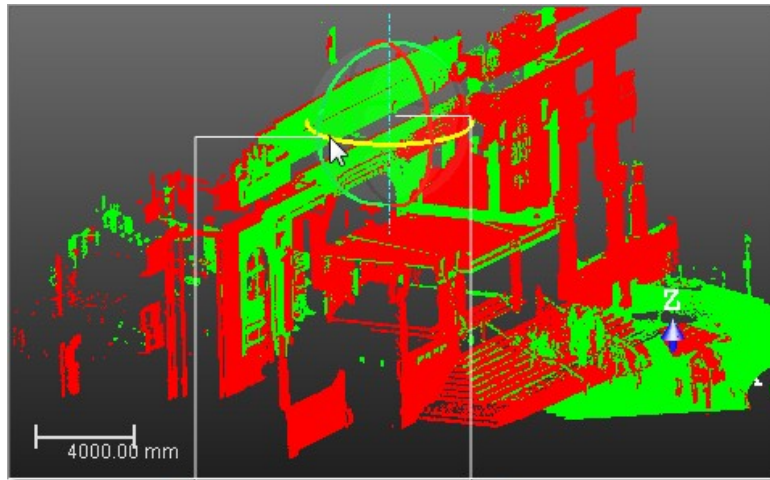


Directions of Pan

Interactive Rotation

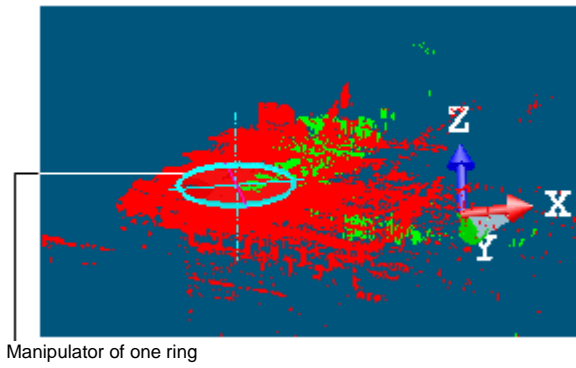
To Rotate the Moving Station:

1. Click **Interactive Rotation** . The two sub-views are grouped in one. The cloud of each station remains with its own color (red for the **Reference Cloud** and green for the **Moving Cloud**). A **Manipulator** appears. It is composed of three **Ring Handles** corresponding to three directions, each with a color (red, green and blue) and has as shared center the **Moving Cloud's** center. With this **Manipulator**, you can rotate the **Moving Cloud** around an axis passing through each ring center and perpendicular to it.
2. Click on a **Sphere Handle**; it turns to yellow. The axis the **Moving Cloud** can be rotated around is dotted and in green color.
3. Move the **Moving Cloud** around that axis.



1 - Selected ring 2 - Axis of rotation

4. For stations from a leveled instrument or from an instrument that is leveled and setup over a known point - respectively of blue color and green color in the **Scans Tree** - you can only rotate them around the Z-axis of the active coordinate frame; this is to preserve the leveling information on stations.



Picking Three Pairs of Points

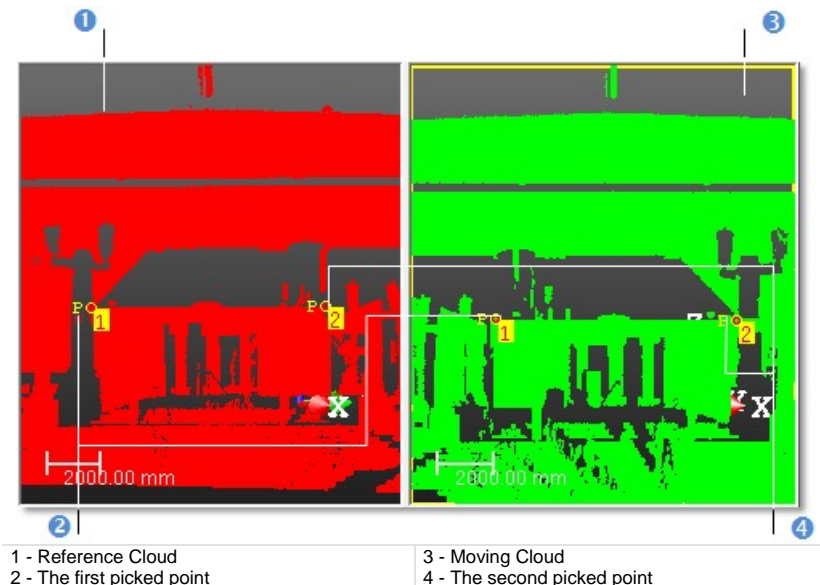
The picking three point mode is default set; you can go to pick three pairs of corresponding points from the clouds of both stations. Each pair of points should be picked on a similar position of the scene. Once the three pairs of points are selected, an initial registration is calculated and the clouds of two stations are superimposed.

To Pick Three pairs of Points:

1. Select a sub-view by clicking inside.
2. Pick a series of three points. Picking should be on the displayed cloud.
3. Select the other sub-view by clicking inside.
4. Pick another series of three points. Picking should be on the displayed cloud too.

Or

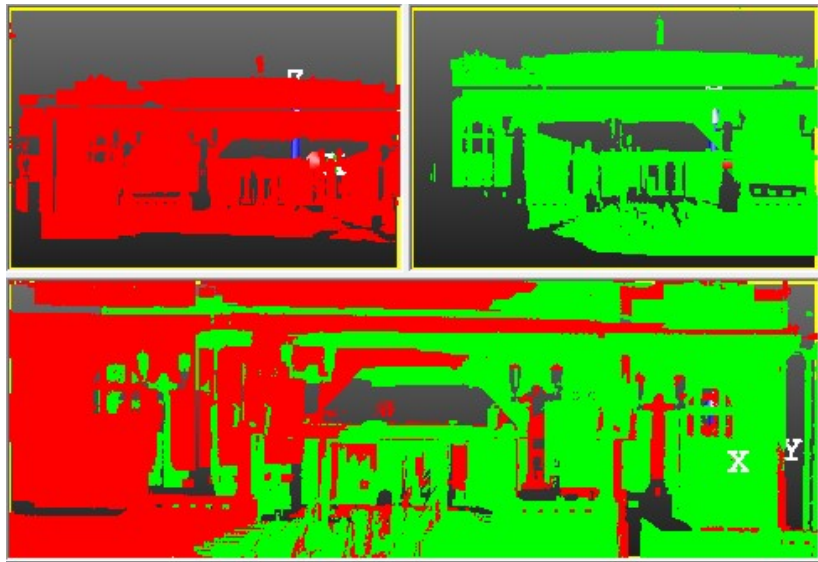
5. Select a sub-view by clicking inside.
6. Pick one point. Picking should be on the displayed cloud.
7. Go to the other sub-view.
8. Pick one point. Picking should be on the displayed cloud.
9. Go back to the first sub-view and pick another.
10. Repeat the steps until you reach three points on each displayed cloud.



A point once selected in the **Reference Cloud** (or **Moving Cloud**) is indicated by a yellow square with a number inside. This number increases one-by-one from **One** to **Three** each time you progress in your selection. You cannot exceed three. Once three pairs of points are selected, an initial registration is calculated and the clouds of the two selected stations (or group of stations) are superimposed. The two sub-views merge into one and the **3 Points** button becomes enabled. This means that you are no longer in the selection mode.

You have to click again on the **3 Points** button if you want to perform a new selection. In each sub-view, you can navigate (like **Zoom**, **Rotate**, etc.) while you select points. Just click inside (except on displayed object(s)) to select it and a yellow frame appears. Otherwise, you select a registration point. You cannot select an object in display in each sub-view.

Note: To split the **3D View** into three sub-views (the top right and left sub-views display the **Reference Cloud** and **Moving Cloud** and the third sub-view displays the registration result), select **Restore Default Layout** from the pop-up menu.



Notes:

- To cancel the registration you have just done and start a new one, click again **3 Points**.

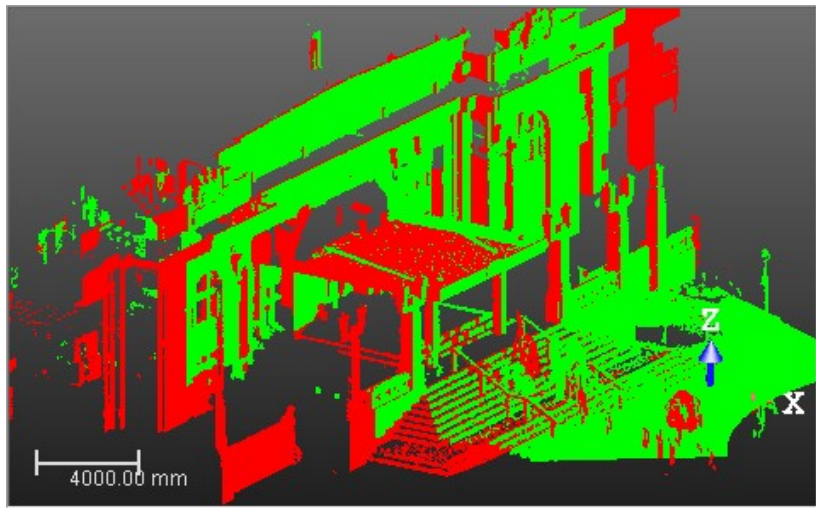
- Once the registration of the two stations done, you can still carry out a sampling on each of them (or a segmentation on both). To do it, select one of the available commands: **Segmentation**, **Sampling Reference Points**, **Sampling Moving Points**, **Reloading Reference** and **Reloading Moving Points** from the pop-up menu.

Refining the Initial Registration

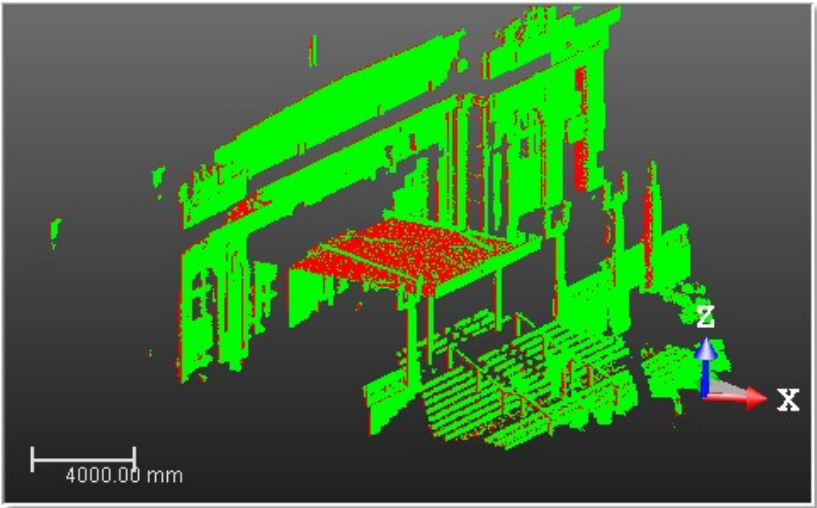
You can now refine the initial registration, if necessary. The refinement uses an iterative method. So you can perform several iterations to improve the results. The registration error shown in the **Cloud-Based Registration** dialog is the average distance error of the points present in the common parts of the two clouds. The average distance error is expressed in millimeter. You can change the unit of measurement by selecting the **Preferences** command from the **Edit** menu.

To Refine the Initial Registration:

1. Click **Refine** to refine the result.
2. If required, click again **Refine** until you obtain the best registration result.



3. Once you are satisfied with the registration result, you can overlap the selected stations using the common parts.

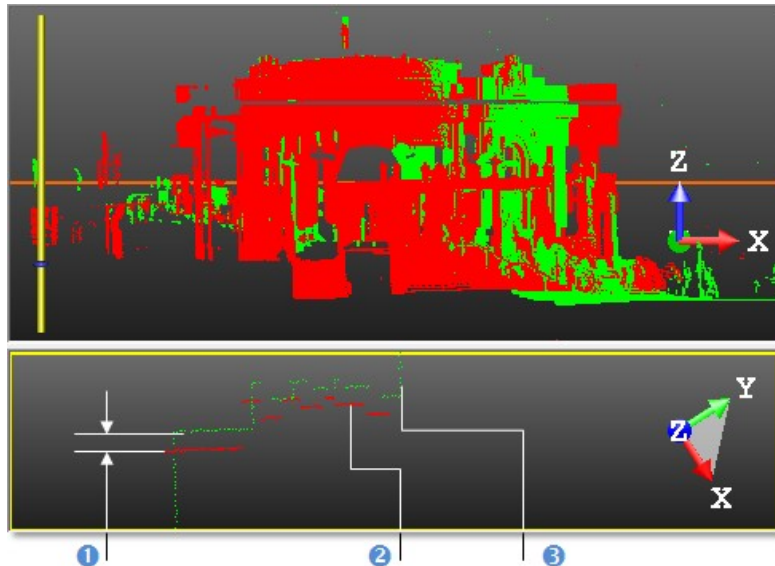


Checking the Registration Results

You can either visually check this result in the **3D View** because each station (or group of stations) still remains with each own color* or control the errors displayed in the **Control Quality** field. You have to use the **Cutting Plane** and **Measurement Tools** for visually check the result. The registration error is the average distance error of the points presented in the common parts of the two clouds.

To Check the Registration Result:

1. Click **Cutting Plane**. The **Cutting Plane** dialog opens.
2. In the **3D View**, define a cutting plane.



- | | |
|--|---|
| 1 - Gap between the Reference Cloud and the Moving Cloud | 2 - Reference Cloud (red)
3 - Moving Cloud (green) |
|--|---|

3. Move the slider from **Up** to **Down** and vice versa until you have the best cutting plane position.
4. In the planar view (the one under the **3D View** window), check visually the gap between the **Reference Cloud** and the **Moving Cloud**.
5. Click **Measurement Tool** in the **Tools** toolbar. The **Measurement Tool** toolbar appears.
6. Measure the gap between the **Reference Cloud** and **Moving Cloud** by picking one point from each cloud.
7. Click **Close Tool** in the **Measurement Tool** toolbar.

8. Click **Close** in the **Cutting Plane** dialog.

Note: (*) Clouds are always rendered in **Red** and **Green**, regardless of the **Rendering** mode.

Saving the Registration Results

You can save the registration result either by using the **Apply** (or **Group**) button. In the latter case, both stations will be put into a group which is automatically created under the current active folder of the **Scans Tree**.

To Save the Registration Result:

1. Click **Apply** to validate the registration in the database.
2. Click **Group** to validate the registration and set the two objects in a new group.
3. Click **Close**.


Note: You can continue to register other stations without quitting the tool. If you use the **Group** command to put the stations just registered together, you can use it to register with another station. In this way, you can structure the **Scans Tree** in such a way that it reflects the history of your registration procedure.

Orientation Tool

The **Orientation Tool** provides the user with tools to easily orientate a 3D scene after it has been locally registered. It is assumed that, in most cases, the instrument (used to acquire the 3D scene data) is leveled, and the **Z** axis is correct. The **Orientation Tool** then allows orienting the scene in 2D in order to re-define the **X** and **Y** axis.

Opening the Tool

To Open the Tool:

1. Perform a display in the **3D View**.
2. From the **Registration** menu, select **Orientation Tool** . The **Orientation Tool** toolbar appears.
 - The 3D scene is locked in a 2D plane in the **Top** view (in the **XY** plane) with a 2D grid superimposed (if not hidden previously).
 - A temporary yellow frame appears:
 - If only one station is displayed in the **3D View**, the origin of the yellow frame is the origin of the station.
 - If several stations are displayed in the **3D View**, the origin of the yellow frame is the origin of the last station (from the project).
 - If several stations (with a **TZF** scan in each) are registered and displayed in the **3D View**, the origin of the yellow frame matches the origin of the **Reference Station**.

Tips:

- To close the **Orientation Tool**, press **Esc**.
- Tools present in the **Orientation Tool** toolbar can also be reached from the pop-up menu.



Caution: You can enter in the tool without displaying anything. But this has no sense because most of the tools (in the **Orientation Tool**) are based on picking on object(s) displayed in the **3D View**. The **Automatic Axis Definition** icon in the **Orientation Tool** toolbar is dimmed.

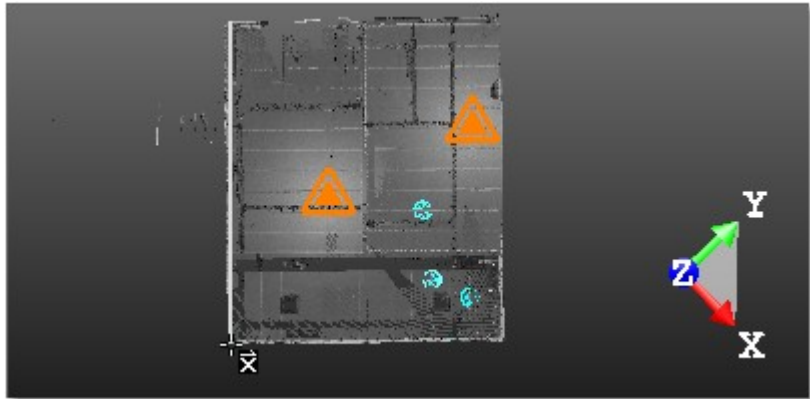
Note: There is no way to unlock the 3D scene from the 2D lock. Once in that 2D lock position, you can only **Pan** in the **YZ** plane, **Zoom In** (or **Out**) along the **Z** axis or **Rotate** around the **Z** axis.

Caution: Frame transformations cannot be applied to projects linked to remote datasets (those extracted from **Scan Explorer**) (or to remote projects). When you attempt to perform such operations, an error dialog appears.

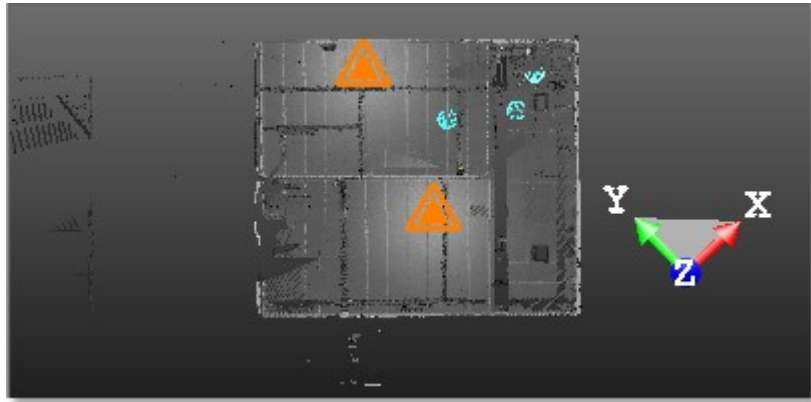
Define Horizontal Axis by Picking Two Points

To Define the Horizontal Axis by Picking Two Points:

1. Click **Define Horizontal Axis by Picking Two Points** . The cursor changes as follows . The **Picking Parameters** toolbar appears in the 2D constraint mode. The yellow frame disappears from the **3D View**.
2. Pick a 3D point in the **3D View**.
3. Pick another 3D point in the **3D View**.
 - The two picked points define an axis.



- The 3D scene is then rotated (in the **XY** plane and around the **Z** axis of the current frame) so that this axis becomes horizontal.



- The yellow frame reappears in the **3D View**. It remains unchanged in position but not in orientation. Its **X** axis is then parallel to the picked axis.

Notes:

- (*) To leave the picking mode, press **Esc**.
- Picking should not be necessary on the displayed object.

Automatic Axis Definition

The **Automatic Axis Definition** feature allows the user to find the correct orientation based on the **Normal X**, **Normal Y** and **Normal Z** information that are in the displayed point cloud.

To Automatically Define Axis:

- Click **Automatic Axis Definition** 

Rotating Counterclockwise 90°



To Rotate Counterclockwise 90°:

- Click **Rotate Counterclockwise 90°** . The whole 3D scene is then rotated of 90° counterclockwise. The yellow frame remains unchanged in position and in orientation.

Picking the Origin

The **Pick Origin** feature allows the user to associate a picked point with a **Known Point**.

To Pick the Origin:

- Click **Pick Origin** . The cursor changes as follows . The **Picking Parameters** toolbar appears in the 3D constraint mode
- Pick a 3D point in the **3D View**. The **Define Origin** dialog opens. The 3D coordinates of the picked point are displayed in the **Picked 3D Point** field. The values in this field are not editable.
- Input **Known Point** coordinates in the **New Coordinates** field.
- Click **OK**. The **Define Origin** dialog closes. The origin of the yellow frame is then moved to the picked point.


Notes:

- (*) To leave the picking mode, press **Esc**.
- Picking should be the on displayed cloud. You may hear a warning sound when picking an empty point.

Tip: You can select the 3D coordinates that appear in the **Picked 3D Point** field (after picking a point).


Applying Transformation

To Apply Transformation:

1. Click **Apply Transformation** .
2. Or press **Enter**. A dialog opens and prompts you to apply the transformation the **Home Frame**.
3. Click **Yes**. All coordinates of the current project are then modified.
4. Or click **No** to not apply.

Closing the Tool

To Close the Tool:

1. Click **Close Orientation Tool** .
2. Or press **Esc**. A dialog opens and prompts you to save the new orientation or not.
3. Click **Yes** to apply.
4. Or click **No** to not apply.


Georeferencing Tool

Georeferencing describes the process of locating an object in "real world" coordinates. For example, you can georeference your house by determining its latitude and longitude coordinates. In **RealWorks**, the objective of this tool is to allow you to georeference a station (or a group of stations or a project) to a known coordinate system. To do this, you have to assign for some targets (or points) of the station (or group of stations) the corresponding known coordinates. Once you assign at least three pairs, a least squares fitting method will be used to calculate the best transformation. You can also import a control network surveyed by traditional surveying instruments, and use these control points to assign coordinates. If you apply this procedure station by station, this amounts to performing registration sequentially (in contrast to **Target-Based Registration** where the least squares adjustment is applied simultaneously to all selected stations).

Opening the Tool

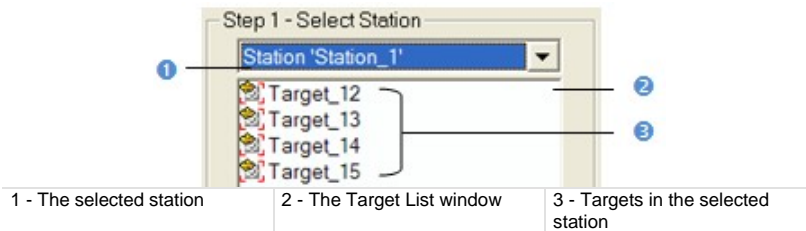
To open the **Georeferencing Tool**, you need to select a station (or a group of stations or a project); no matter if the selection contains or not targets.

To Open the Tool:

1. Import a survey network file including measured points into your project, if required.
2. Select a station (or a project) from the **Scans Tree** and display it if required.
3. In the **Registration** menu, select **Georeferencing Tool** . The **Georeferencing** dialog opens.

This dialog box opens as the fourth tab of the **WorkSpace** window and is composed of four parts. Each of them corresponds to one step in the georeferencing procedure.

- If the input is a lonely station, that station is by-default selected. All targets (if existed) of this station are listed in the **Target List** window.
- If the input is a group of stations, the station selected by-default is the first by selection order. All targets (if existed) of this station are listed in the **Target List** window.
- If the input is a project, that project is by-default selected. All targets (if existed) of this project are listed in the **Target List** window.



Note: A measured point may have two states: **Matched** or **Unmatched**. All measured points when unmatched are gathered into a folder named **Unmatched** and rooted under the **Project** node in the **Targets Tree**. This folder can be reached by selecting the **Targets** tab.

Tip: **Georeferencing Tool** can also be selected from the pop-up menu.

Selecting Stations

You should first select a station for which you want to georeference. Once it is selected, all targets (if existed) of this station are listed in the **Target List** window.

To Select a Station:

1. Click on the **Select Station** pull-down arrow.
2. Select a station from the **Select Station** list. This station is displayed in the **3D View** and all targets that are inside are listed in the **Target List** window.

Note: Multi-selection is forbidden.

Assigning Coordinates to Targets

If there are targets, you can select and assign one of them with known coordinates. You have two methods. If there is a control network imported, you can choose the corresponding one and use it to assign the coordinates. Otherwise, you can key-in the coordinates in the text field. A target, once assigned with known coordinates, will be removed from the **Target List** window and put in a list under the **By Target** and **By Picking** buttons. Similarly, it will be displayed and numbered in the **3D View**.

To Assign Coordinates to Targets:

1. Select a target in the **Target List** window. The **By Target** button becomes active.
2. Click on the **By Target** button. The **Coordinates Assigning** dialog opens.

Selected Target

Name:

Coordinates:

Associated Known Coordinates

TopoPoint:

	X	Y	Z
	0.00 mm	0.00 mm	0.00 mm
	12 (730.96 mm; 380.80 mm; 332.55 mm)		
	13 (755.39 mm; 408.46 mm; 330.32 mm)		
	14 (737.15 mm; 418.58 mm; 330.70 mm)		

Name:

1 - Name of the selected target (not editable) 2 - Coordinates of the selected target (not editable) 3 - The Manual Edit method 4 - Topo points

This dialog is composed of three parts. For a target selected (from the **Georeferencing** dialog or from the **3D View**), its name and coordinates appear in the first part of the dialog and you cannot modify them. The second part enables to assign coordinates to the selected target. You can either edit the coordinates by hand or select a measured point and assign its coordinates. You can then validate the operation in the third part.

3. Do one of the following:
 - Edit known coordinates by hand.

- a) The **Manual Edit** in the **Topo Point** field is set by default. If not, select it.
 - b) Enter a known coordinate in the **X** field*.
 - c) Enter a known coordinate in the **Y** field*.
 - d) Enter a known coordinate in the **Z** field*.
- Select known coordinates from the survey network file.
- a) Click on the **Topo Point** pull-down arrow.
 - b) Select a Topo point from the list.
4. Keep the default name **TopoPoint 1**.
 5. Or enter a new name in the **Name** field
 6. Click **OK**. The **Coordinates Assigning** dialog opens.

Notes:

- You can mix the two ways of assigning coordinates (**By Picking** and **By Target**) in a single georeferencing operation without leaving the tool.
- (*) In the **X, Y, Z Coordinate System**.

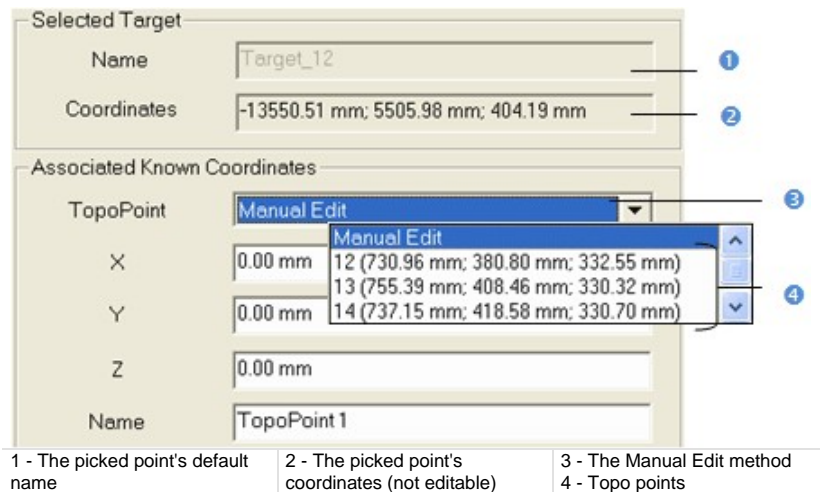
Tip: For assigning known coordinates to a target, you can select it from the **Target List** window or pick on it in the **3D View**. A selected target is highlighted in the **3D View**.

Assigning Coordinates to Picked Points

If there are no targets, the **By Target** button in the **Georeferencing** dialog is unavailable; you can then assign known coordinates to points picked in the **3D View**. You have two methods for assigning the coordinates. If there is a control network imported, you can choose the corresponding one and use it to assign the coordinates. Otherwise, you can key-in the coordinates in the text field.

To Assign Coordinates to Picked Points:

1. In the **Georeferencing** dialog, click on the **By Picking** button.
2. Pick one point on the displayed object(s). The **Coordinates Assigning** dialog opens.



This dialog is composed of three parts. For a point picked in the 3D scene, a name by default **PickPoint1** and its coordinates appear in the first part of the dialog. You can rename this picked point but you cannot modify its coordinates. The second part enables to assign know coordinates to the picked point. You can either edit the coordinates by hand or select a measured point and assign its coordinates. You can then validate the operation in the third part.

3. Keep the default name **PickPoint 1**.
4. Or enter a new name in the **Name** field.
5. Do one of the following:

- Edit known coordinates by hand.
 - a) **Manual Edit in the Topo Point** field is set by default. If not, select it.
 - b) Enter a known coordinate in the **X** field.
 - c) Enter a known coordinate in the **Y** field.
 - d) Enter a known coordinate in the **Z** field.
 - Select known coordinates from the survey network file.
 - a) Click on the **Topo Point** pull-down arrow.
 - b) Select a Topo point from the list.
6. Keep the default name **TopoPoint 1**.
 7. Or enter a new name in the **Name** field.
 8. Click **OK**.

Notes:

- If you have selected a station, you can only pick points of this station.
- You can leave the picking mode by selecting **Exit Picking Mode** from the pop-up menu.
- You can mix the two ways of assigning coordinates (**By Picking** and **By Target**) in a single georeferencing operation without leaving the tool.
- (*) In the **X, Y, Z Coordinate System**.

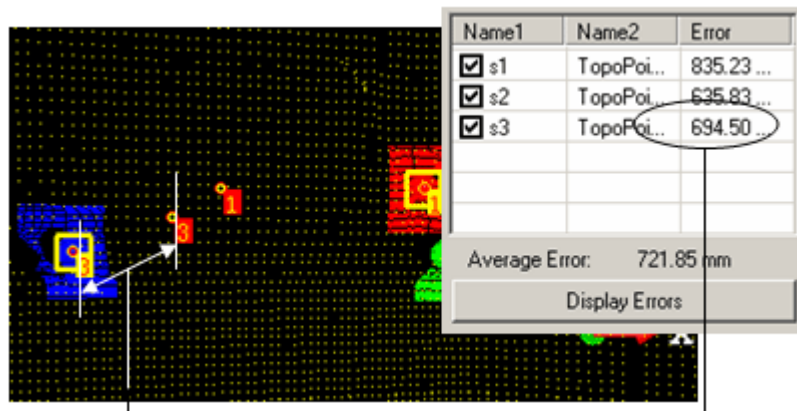
Tip: You can remove the Topo point labels from the **3D View** by first selecting **Rendering**, then **Display 3D Labels** from the **3D View** menu.

Saving the Results

Once at least three known coordinates have been assigned to targets or picked points, a least-squares fitting method will be automatically applied to calculate a best transformation. You can select and delete an already assigned target (or picked point). An assigned target when deleted will be removed from the list below the **By Target** and **By Picking** buttons and put again in the **Target List** window for a new assignment. The error for each target (or picked point) will be expressed in terms of distance. You can un-check targets (or picked points) having the greatest error; the fitting method will be applied each time you check or un-check targets (or picked points). You can visualize errors in the **3D View** by clicking **Display Errors**.

To Save the Results:

1. Click **Display Errors**.



The error is expressed in millimeters

2. Click **Apply**.
3. Click **Close**.

Notes:

- A measured point (from the survey network file) once assigned is set as matched.
- Leaving the **Georeferencing Tool** without applying the georeferencing will display an error message which prompts to abort or continue the operation.

Tip: You can quit the **Georeferencing Tool** by pressing **Esc** or by selecting **Close** from the pop-up menu.

Bundle Adjustment

In this tool, all targets of the entire project are matched without user intervention. This tool shares the same dialog as the **Target-Based Registration Tool**.

To Bundle Adjust:

1. Select a project from the **Project Tree**.
2. In the **Registration** menu, select **Bundle Adjustment**. The **Target-Based Registration** dialog opens as well as the **Registration Details** dialog.

In the **Target-Based Registration** dialog, targets are automatically matched. The **Adjust** button in **Step 3** of the **Target-Based Registration** is grayed-out. In the **Registration Details** dialog, the **Station View** is set by default. Each target is automatically paired with other targets.

In the **Targets Tree**, targets matched together are put in a folder named **mTARGET** and rooted under the **Project** node.

3. Click **Apply**. The **Target-Based Registration** dialog closes. All the changes are applied to the database.
4. Or click **Close**. An information box appears and prompts user to apply the changes in the database or not.

Name-Based Bundle Adjustment

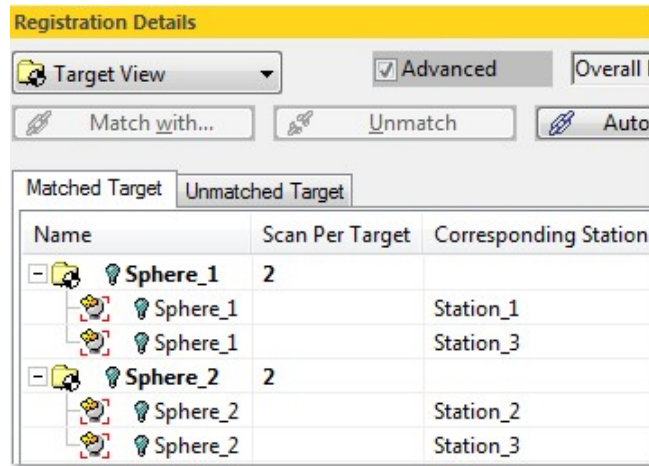
The **Name-Based Bundle Adjustment** feature first un-matches any already matched target* and then matches all targets of the whole project by associating them by name. This feature shares the same dialog as the **Target-Based Registration Tool**.

To Bundle Adjust by Name:

1. Select a project from the **Project Tree**.
2. In the **Registration** menu, select **Name-Based Bundle Adjustment**. The **Target-Based Registration** dialog opens as well as the **Registration Details** dialog.

Targets are automatically matched by their name. The **Adjust** button in **Step 3** of the **Target-Based Registration** is grayed-out. In the **Registration Details** dialog, the **Station View** is set by default.

3. Switch to the **Target View**. Targets that are paired by their name are put in a folder named by the target name.



4. Click **Apply**. The **Target-Based Registration** dialog closes. All the changes are applied to the database.
5. Or click **Close**. An information box appears and prompts user to apply the changes in the database or not.

Note: When you select a project for which any target can be matched by name, you will be automatically direct to the basic **Target-Based Registration Tool**.

Tip: (*) You can also select a project for which no target has been matched.

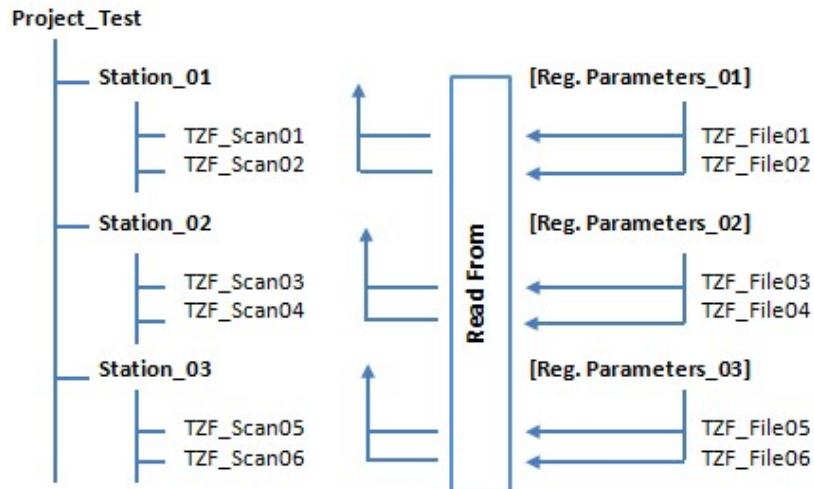
Note: Targets that are matched together are not renamed.

Station Registration Parameters

A **TZF** format file holds registration parameters which are **Vector of Translation**, **Axis of Rotation** and **Angle of Rotation**. On first import of **TZF** format files, in the case of a new project e.g., **RealWorks** creates new stations and automatically initialize them with the registration parameters of the **TZF** format files. These registration parameters will no longer to be read (or written) anymore after the initialization even if they are changed in the meantime. The only way these parameters can be read (or written) is by performing an explicit import (or export).

Importing from TZF Files

The **Import Station Registration Parameters from TZF Files** feature, when selected (from the **Registration** menu), allows the reading of registration information from **TZF** format files and applying them to the station they belong to, as illustrated below

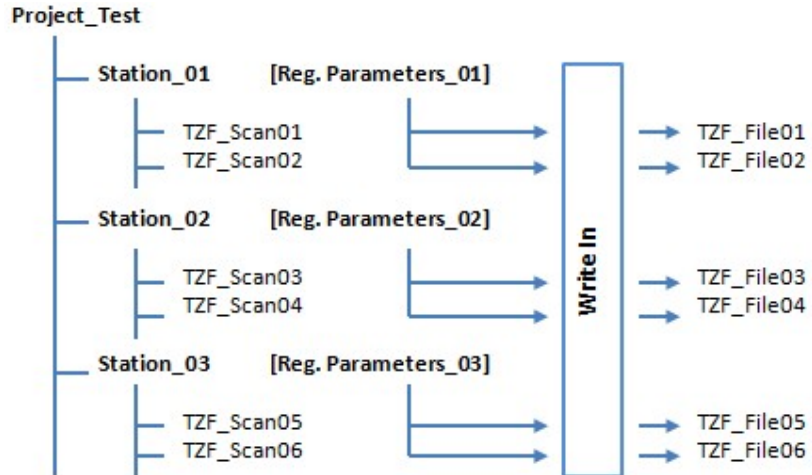


Notes:

- No selection is required but you need to have at least a project loaded in **RealWorks**.
- If the project contains a station that is not valid (with no **TZF Scan(s)** within), a dialog opens and asks you to proceed with the remaining station(s). Choosing **No** will interrupt the import process.

Exporting to TZF Files

The **Export Station Registration Parameters to TZF Files** feature, when selected (from the **Registration** menu), allows the writing of the registration information of a station into all its related **TZF** format files, as illustrated below.



Note: You need to have at least a project loaded in **RealWorks** and you need to perform a selection from the project to enable this feature, whatever the selection.

Exporting to RMX Files

This feature lets you export the **Station Registration Parameters** that have been applied to stations (empty or not) in batch (or interactive) processing mode. The registration parameters (**Vector of Translation**, **Axis of Rotation** and **Angle of Rotation**) are stored in a file of **RMX** format which is an **ASCII** format file. One **RMX** format file will be created for each station. The **RMX** format file has the same name as the registered station.

To Export Station Registration Parameters to RMX Files:

1. First apply transformation to stations (if required)*.
2. Then select either a project (or a station (or a set of stations)).
3. From the **Registration** menu, select **Export Station Registration Parameters to RMX Files**. The **Select New File Folder** dialog opens.
4. Navigate to the drive/folder where you want to store the **RMX** format files in the **In** field.
5. Click **Open**. The **Select New File Folder** dialog closes.

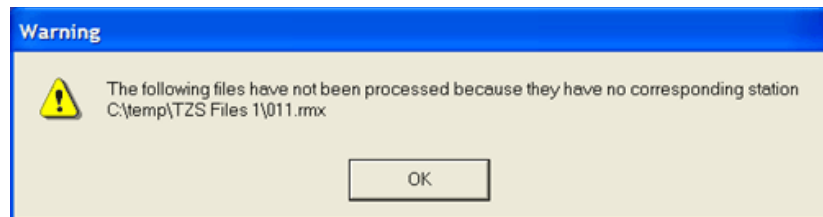
Note: (*) If any transformation has been applied to stations; the registration parameters (**Vector of Translation**, **Axis of Rotation** and **Angle of Rotation**) are equal to zero.

Register Stations With Imported RMX Files

This feature allows you to import and apply registration parameters to stations (empty or not) in batch (or interactive) processing mode. The registration parameters (**Vector of Translation**, **Axis of Rotation** and **Angle of Rotation**)* are stored in a file of **RMX** format which is an **ASCII** format file. You need to have one **RMX** format file per station. The **RMX** format file has the same name than the station to register.

To Register Stations With Imported RMX Files:

1. Select a project or a station (or a set of stations).
2. From the **Registration** menu, select **Register Stations With Imported RMX Files**. The **Select New File Folder** dialog opens.
3. Navigate to the drive/folder where the **RMX** format files are stored in the **In** field.
4. Select and open the folder by double-clicking on it**.
 - The **RMX** format files will be processed one after the one.
 - For a given **RMX** format file, if **RealWorks** finds a related station; the registration parameters will be applied to the station. If the **RMX** format file has no corresponding station; the warning dialog below appears. Click **OK**. The registration parameters won't be applied.



Notes:

- (*) 3D coordinates in millimeters.
- (**) Otherwise the above warning dialog appears.

Modifying Stations

The "Modify Stations" menu gathers the operations the user can apply to stations.

Forcing Leveled

A station for which the leveling information is missing because the instrument had not been initially leveled can be forced leveled.

To Force Leveled:

1. Select an un-leveled station from the Scans Tree.
2. In the Registration menu, select Modify Station / Force Leveled.

In the Scans Tree, the selected item switches from  to .

In the Property window, the "Scanner Leveling" state switches from "False" to "True". The "Instrument Height" and "Projected Instrument Position" appear with respectively 0.00 m as distance value for the first and 0,0,0 as coordinates for the second.

Scanner ID	Trimble Scan
Scanner leveling	False
Scanner Origin	0.00 m; 0.00 m; 0.00 m
Scanner Right Direction	1.00; 0.00; -0.00
Scanner View Direction	0.00; 1.00; 0.00
Scanner Up Direction	0.00; 0.00; 1.00

Scanner ID	Trimble Scan
Scanner leveling	True
Instrument height	0.00 m
Projected Instrument Position	0.00 m; 0.00 m; 0.00 m
Scanner Origin	0.00 m; 0.00 m; 0.00 m
Scanner Right Direction	1.00; 0.00; -0.00
Scanner View Direction	0.00; 1.00; 0.00
Scanner Up Direction	0.00; 0.00; 1.00

Notes:

- You can undo the operation.
- You cannot select a set of stations.

Modifying the Instrument Height

To Modify the Instrument Height:

1. Select a leveled station from the **Scans Tree**.
2. In the **Registration** menu, select **Modify Station / Modify Instrument Height**. The **Modify Station Height** dialog opens.
3. Enter a distance value in the **Station Height** field.
4. Click **Apply**. The **Modify Station Height** dialog closes.

In the **Property** window, the current value of the "Instrument Height" line changes. In the example below, the keyed-in value 1.50 m is displayed instead of 0.

Scanner ID	Trimble Scan
Scanner leveling	True
Instrument height	0.00 m
Projected Instrument Position	0.00 m; 0.00 m; 0.00 m
Scanner Origin	0.00 m; 0.00 m; 0.00 m
Scanner Right Direction	1.00; 0.00; -0.00
Scanner View Direction	0.00; 1.00; 0.00
Scanner Up Direction	0.00; 0.00; 1.00

Scanner ID	Trimble Scan
Scanner leveling	True
Instrument height	1.50 m
Projected Instrument Position	0.00 m; 0.00 m; 0.00 m
Scanner Origin	0.00 m; 0.00 m; 1.50 m
Scanner Right Direction	1.00; 0.00; -0.00
Scanner View Direction	0.00; 1.00; 0.00
Scanner Up Direction	0.00; 0.00; 1.00

The new set value is added to the Z coordinate of all items (point, geometry, scanner origin, etc.) of the selected station.

Tip: You can also select and right-click on a leveled station in the **Project Tree** and select **Modify Instrument Height** from the pop-up menu.

Caution: You can undo the operation.

Note: If some targets of the selected station have been previously paired with targets of other stations, a warning message appears and warns you that a Bundle Adjustment of the project is now necessary to adjust corresponding stations.

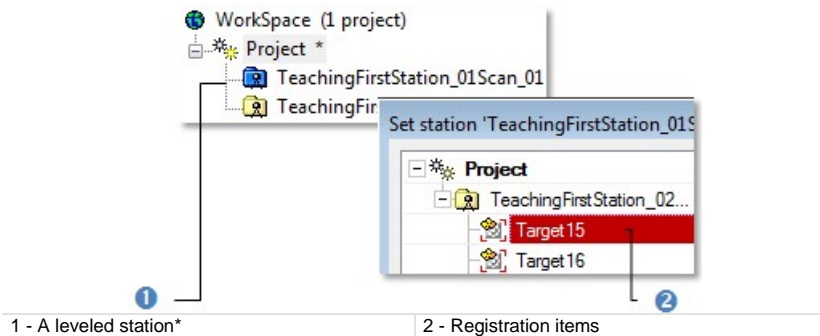
Tip: If the selected station has several **TZF Scans** within, you can select a unique **TZF Scan** and set an **Instrument Height** to it. This parameter is then applied to the station the **TZF Scan** belongs to.

Setting Over a Known Point

Setting-up a station over a **Known Point** consists of associating both together. A **Known Point** can be a fitted target (sphere or flat target), a **Survey Point** or a **Topo Point**. It must belong to a different station than the selected one.

To Setup Over a Known Point:

1. Select a leveled station from the **Scans Tree**.
2. In the **Registration** menu, select **Modify Station / Set Over Known Point**. The **Set Over Known Point** dialog opens.
3. Expand the **Project Tree** if required.



4. In the **Set Over Known Point** dialog, the **Project Tree** gathers all registration items except those belonging to the selected station.
5. Select an item. The **OK** button becomes enabled.
6. Click **OK**. The **Set Over Known Point** dialog closes.

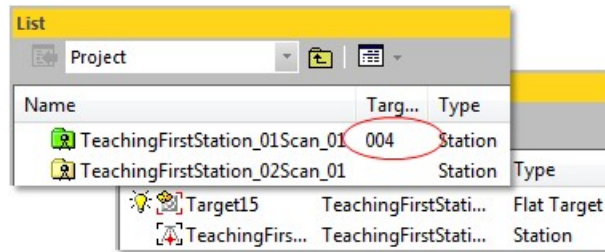
In the **Scans Tree**, the station item switches from to .

In the **Property** window, the **"Over a Known Point"** line appears. The selected item's name appears in that line. The **"Projected Instrument Position"** and **"Scanner Origin"** lines are updated respectively with the selected item's 3D coordinates and the selected item's 3D coordinates plus the **Instrument Height** value for the **Z** value.

Scanner ID	Trimble Scan
Scanner leveling	True
Instrument height	1.50 m
Projected Instrument Position	0.00 m; 0.00 m; 0.00 m
Scanner Origin	0.00 m; 0.00 m; 1.50 m
Scanner Right Direction	1.00; 0.00; -0.00
Scanner View Direction	0.00; 1.00; 0.00
Scanner Up Direction	0.00; 0.00; 1.00

Scanner ID	Trimble Scan
Scanner leveling	True
Instrument height	1.50 m
Over a Known Point	Target15
Projected Instrument Position	-3.64 m; -0.18 m; 2.21 m
Scanner Origin	-3.64 m; -0.18 m; 3.71 m
Scanner Right Direction	1.00; 0.00; 0.00
Scanner View Direction	0.00; 1.00; 0.00
Scanner Up Direction	0.00; 0.00; 1.00

7. In the **Targets Tree**, a matched folder is created with the selected station and the known point inside.



Caution: You can undo the operation.

Notes:

- If some targets of the selected station have been previously paired with targets of other stations, a warning message appears and warns you that a Bundle Adjustment of the project is now necessary to adjust corresponding stations.
- (*) It is not necessary to have a leveled station as input.


Caution: The **Set Over Known Point** feature is dimmed if there is no target inside the other station.

Removing a Known Point

Removing a **Known Point** from a station consists of dissociating one from the other; this doesn't require any parameter and no dialog appears. You can change the current **Known Point** of a station; first remove it and then associate a new **Known Point**.

To Remove a Known Point:

1. Select a station setup over a known point from the **Scans Tree**.
2. In the **Registration** menu, select **Remove Known Point**.

In the **Scans Tree**, the selected station switches from  to .

In the **Property** window, the "Over a Known Point" line disappears. Note that the coordinates in the "Projected Instrument Position" and "Scanner Origin" lines remain unchanged.

Scanner ID	Trimble Scan
Scanner leveling	True
Instrument height	0.00 m
Over a Known Point	Target7
Projected Instrument Position	-8.43 m; 17.87 m; 2.15 m
Scanner Origin	-8.43 m; 17.87 m; 2.15 m
Scanner Right Direction	0.97; -0.24; -0.00
Scanner View Direction	0.24; 0.97; 0.01
Scanner Up Direction	-0.00; -0.01; 1.00




Scanner ID	Trimble Scan
Scanner leveling	True
Instrument height	0.00 m
Projected Instrument Position	-8.43 m; 17.87 m; 2.15 m
Scanner Origin	-8.43 m; 17.87 m; 2.15 m
Scanner Right Direction	0.97; -0.24; -0.00
Scanner View Direction	0.24; 0.97; 0.01
Scanner Up Direction	-0.00; -0.01; 1.00



Caution: You can undo the operation.

Forcing Unleveled

To Force Unleveled:

1. Select a leveled (or setup over a known point) station from the **Scans Tree**.
2. In the **Registration** menu, select **Force Leveled**.

In the **Scans Tree**, the selected item switches from  (or ) to .

In the **Property** window, the "**Scanner Leveling**" state switches from "**True**" to "**False**". For a  leveled station, the "**Instrument Height**" and "**Projected Instrument Position**" information are lost. For a  setup over a known point station, the "**Instrument Height**", "**Over a Known Point**" and "**Projected Instrument Position**" information are lost.

Scanner ID	Trimble Scan
Scanner leveling	True
Instrument height	0.00 m
Projected Instrument Position	0.00 m; 0.00 m; 0.00 m
Scanner Origin	0.00 m; 0.00 m; 0.00 m
Scanner Right Direction	1.00; 0.00; -0.00
Scanner View Direction	0.00; 1.00; 0.00
Scanner Up Direction	0.00; 0.00; 1.00

Scanner ID	Trimble Scan
Scanner leveling	False
Scanner Origin	0.00 m; 0.00 m; 0.00 m
Scanner Right Direction	1.00; 0.00; -0.00
Scanner View Direction	0.00; 1.00; 0.00
Scanner Up Direction	0.00; 0.00; 1.00

Caution: You can undo the operation.

Modifying Target Matching

The "Modify Target Matching" menu gathers the operations the user can apply to targets.

Matching With

Matching a pair of targets consists of pairing one with the other. The **Match With** command can be reached from the **Registration** menu or within e.g. the **Target-Based Registration Tool** in the **Registration Details** dialog after auto-pairing of targets. You can select an unmatched item (or matched item) as input of that tool.

To Match an Item with Another Item:

1. Select a registration item (matched or not) from the **Project Tree**.
2. From the **Registration** menu, select **Modify Target Matching / Match With**. The **Match With** dialog opens.
3. In the **Match With** dialog, expand the **Project Tree** if required.

In the **Match With** dialog, the **Project Tree** gathers all unmatched registration items except those belonging to the selected station.

4. Select an unmatched item from the **Match With** dialog. The **OK** button becomes active.
 - If the selected item is not yet matched, both are matched together and put under a matched folder in the **Targets Tree**.
 - If the selected item is already matched, both are also matched together but no matched folder is created in the **Targets Tree**. The item selected for matching (in the **Match With** dialog) is put under the existing matched folder (the one inside which reside the selected item and its pair).

Tip: You can also right-click on an unmatched item from the **Project Tree** and select **Match With** from the pop-up menu.

Matching Targets

You can manually pair an unmatched target with another unmatched target. Both must not reside under the same station. You can also pair two different stations. Both need to be leveled and setup over a **Known Point**.

Notes:

- You can undo the operation.
- Use the **Ctrl** (or **Shift**) key combined with the left-click for multi-select items.

Targets

To Match Targets:

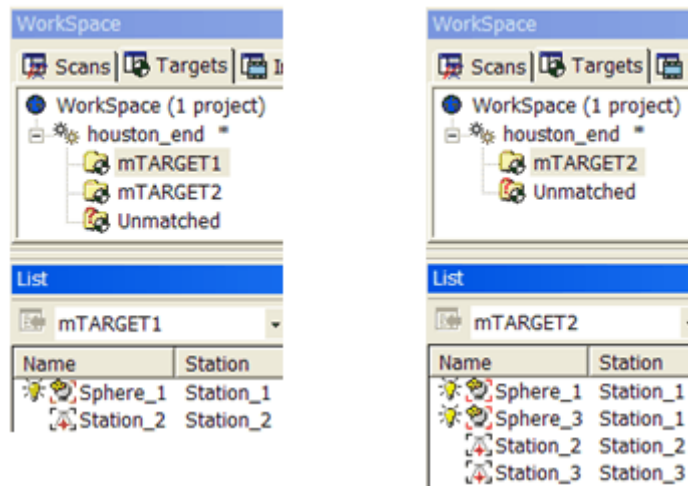
1. Select an unmatched target from a station in the **Project Tree**.
2. Select another unmatched target from a different station in the **Project Tree**.
3. In the **Registration** menu, select **Modify Target Matching / Match Targets**.

Targets (once paired) are put under a matched folder in the **Scans Tree**.

Stations

To Match Stations:

1. Select two stations (leveled and setup over a known point) from the **Scans Tree**.
2. From the **Registration** menu, select **Modify Target Matching / Match Targets**. A dialog opens and prompts you to accept the removal of a target (or not).
3. Click **Yes** to accept.



Notes:

- You can undo the operation.
- Use the **Ctrl** (or **Shift**) key combined with the left-click for multi-select items.

Un-matching Targets

Un-matching a pair of matched targets consists of dissociating one from the other. Selecting a matched target will unmatch the pair that it belongs to. Selecting a station will unmatch all pairs of matched targets that are inside.

To Unmatch a Target:

1. In the **Registration Details** dialog*, select a matched entity from a station in the **Station View** (or from a pair in the **Target View**). The **Match With** and **Unmatch** buttons become enabled.
2. Click on the **Unmatch** button.

Or

3. Select a matched entity from a pair.
4. From the **Registration** menu, select **Modify Target Matching / Unmatch Target**.

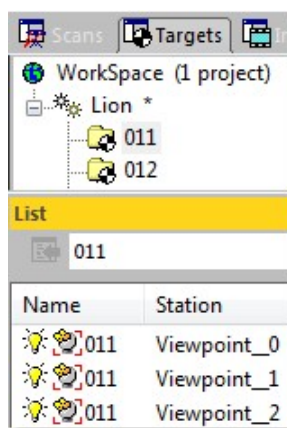
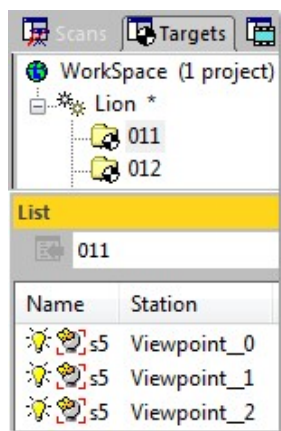
Tip: You can also right-click on a matched item from the **Project Tree** and select **Un-match Target** from the pop-up menu.

Renaming Targets

Auto-Pairing Targets: Targets and target groups are renamed. Instead of being named **TargetX** and **mTargetX** where **X** is an order, they are renamed as **XXX**. **XXX** starts at 001 and is incremented by one. **Manual-Pairing Targets:** Targets keep their default name and target groups are renamed as described above. The **Rename Targets** feature allows renaming of targets in case the manual method has been used.

To Rename Targets:

1. Select a target group from the **Targets Tree**.
2. From the **Registration** menu, select **Modify Target Matching / Rename Targets**. Targets are renamed according to the target group name they belong to.



Tip: You can also right-click on a target group and select **Rename Targets** from the pop-up menu.

Creating Points

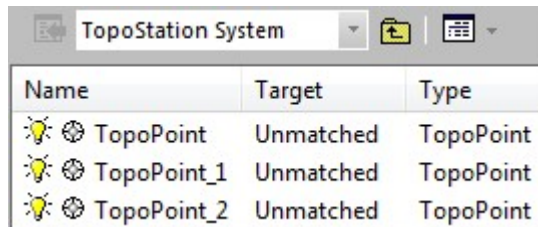
The "Create Points" menu gathers the operations related to the creation of points.

Creating Topo Points

To Create a Topo Point:

1. Select anything from the Project Tree.
2. In the Registration menu, select Create Points / Create Topo Point. The Create New Topo Point dialog opens.
3. Keep the default name TopoPoint.
4. Or enter a new name in the Topo Point Name field.
 - If the given name does not exist, jump to the next step.
 - If the given name already exists, a dialog appears and prompts to set a new name instead. Click Yes to use the new name.
5. Enter new coordinates in the Coordinates field.
6. Click Create. The Create New Topo Point dialog closes.

An unmatched Topo Point is created and placed under a station (named TopoStation System) in the Scans Tree. This Topo Point is shown in the 3D View.



Name	Target	Type
TopoPoint	Unmatched	TopoPoint
TopoPoint_1	Unmatched	TopoPoint
TopoPoint_2	Unmatched	TopoPoint

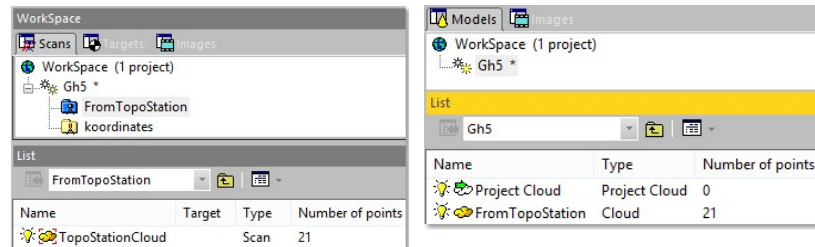
Note: You can undo the operation.

Tip: You can display (or hide) the Topo Point's 3D Labels by selecting Rendering/Display 3D Labels from the 3D View menu. The Display 3D Labels feature once selected has a check mark on its side.

Creating Point Cloud from Topo Points

To Create a Point Cloud from Topo Points:

1. Select a **Topo Point** (or a set of **TopoPoints**) from the **Scans Tree**.
2. In the **Registration** menu, select **Create Points / Create Point Cloud from Topo Points**.



Each **Topo Point** is converted to a 3D coordinate point. In the **Scans Tree**, all are gathered in a scan named **TopoStationCloud** which is placed under a leveled station (blue color) named **FromTopoStation**. This scan is displayed in the **3D View**. In the **Models Tree**, a new point cloud (named **FromTopoStation**) is created.

Notes:

- You cannot undo the operation.
- By default, the **Project Cloud** does not contain the newly created point cloud.

Creating 3D Points

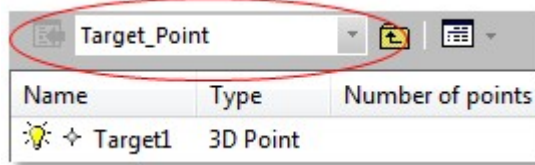
You can convert any registration entity (matched or unmatched) to a 3D coordinate point. A registration entity can be a **Spherical Target**, a **Flat Target**, a **Survey Point** coming from a survey network file, an imported **Topo Point** or a **Topo Point** created within a **RealWorks** tool such as the **Georeferencing Tool** or a target obtained by geometry fitting. The idea of this feature is to convert the barycentre of a registration entity to a **3D Point**. This barycentre is the averaged center position when selecting a set of matched targets.

Creating 3D Points From Targets

To Create a 3D Point from a Target:

1. From the **Scans Tree**, select a set of matched (or unmatched) entities from a station.
2. From the **Registration** menu, select **Create Points / Create 3D Points From Targets**. An information box opens.
3. Click **OK**. The information box closes.

A group named **Target_Point (x)** is created and rooted under the current project in the **Models Tree**; where X is its creation order. This group gathers the entities that are converted to **3D Points**. There is a **Target_Point** group per conversion. Each **3D Point** has the name of the entity it is issued from.



Note: A selection is always from the **Scans Tree**. When you select an unmatched entity from the **Unmatched** folder (or a matched entity from a pair) in the **Targets Tree**, only the **From Matched Targets** command is available. For both a warning message appears and warns that the selection is not valid.

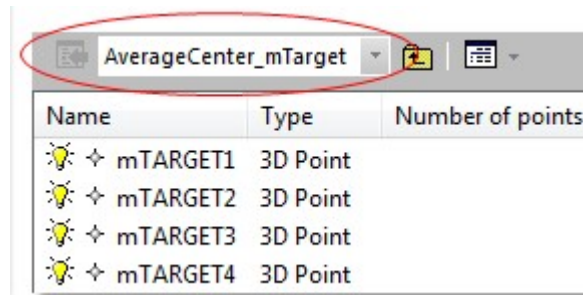
Tip: You can display (or hide) the **3D Point's 3D Labels** by selecting **Rendering/Display 3D Labels** from the **3D View** menu. The **Display 3D Labels** feature once selected has a check mark on its side.

Creating 3D Points From Matched Targets

To Convert a Pair of Matched Entities to a 3D Point:

1. From the **Targets Tree**, select at least a pair of matched entities.
2. From the **Registration** menu, select **Create Points / Create 3D Points From Matched Targets**. An information box opens.
3. Click **OK**. The information box closes.

A group of **AverageCenter_mTarget** is created and rooted under the current project in the **Models Tree**. This group contains the average barycenter of the matched entities. Each **3D Point** has the name of the pair (of matched entities) it is issued from.



Notes:

- There is no **Undo**.
- The **Create 3D Points From Matched Targets** command is available even if you select a single matched entity from a pair. But a warning message appears and warns you that the selection is not valid. No **3D Point** is then created in the **Models Tree**.

Tip: You can display (or hide) the **3D Point's 3D Labels** by selecting **Rendering/Display 3D Labels** from the **3D View** menu. The **Display 3D Labels** feature once selected has a check mark on its side.

Creating a Registration Report (Target-Based)

The **Registration Report (Target-Based)** feature lets the user to create a report after on a registration (based on paring of targets) in an **RTF** (Rich Text Format) file. The **RTF** specification is a method of encoding formatted text and graphics for easy transfer between applications. Currently, users depend on special translation software to move word-processing documents between different MS-DOS®, Windows, OS/2, Macintosh, and Power Macintosh applications. The **RTF** specification provides a format for text and graphics interchange that can be used with different output devices, operating environments, and operating systems. This feature is only available in **Registration**. You can only create one report per project. If several projects exist under the **Project Tree**, the report concerns the project which contains the active group. For a given project, if any registration has been performed, the report is empty of information.

To Create a Registration Report (Target-Based):

1. Select a project from the **Project Tree**.
2. From the **Registration** menu, select **Registration Report (Target-Based)**. The **Registration Report (Target-Based)** dialog opens.
3. Navigate to the drive/folder where you want the report file to be stored in the **Look In** field.
4. Enter a name in the **File Name** field. The extension **RTF** is added automatically.
5. Click **Save**.

Note: The results in the report are spilt into two categories: **By Stations** and **By Targets**.

CHAPTER 11

OfficeSurvey

When you load a file of any format - except SIMA and TXT with topopoints - that had never been previously saved in the **RealWorks** format; the **OfficeSurvey™** processing mode is set by default. When you load a file - saved in the **RealWorks** format and in the **OfficeSurvey™** processing mode; that file will be opened with that processing mode set. When you are out of this processing mode and you need to be in it; you have to click on the pull-down arrow in the **Tools** toolbar and choose **OfficeSurvey™**.

The **OfficeSurvey** processing mode is present in all **RealWorks** products. Tools in the **OfficeSurvey** menu* differ from a license to another. The table below lists the contents the user can have according to the license (of **RealWorks**) he has.

Tools (in)	Viewer	Base	Advanced Advanced - Modeler Advanced - Plant
Cutting Plane		✓	✓
Contouring		✓	✓
2D-EasyLine		✓	✓
Mesh Creation		✓	✓
Mesh Editing		✓	✓
Ortho-Projection		✓	✓
Multi-Ortho-Projection			✓
Image Rectification		✓	✓
Polyline Drawing		✓	✓
Image Matching		✓	✓
Volume Calculation		✓	✓
Profile Cross-Section		✓	✓
Feature Set			✓
2D Polyline Inspection			✓
Twin Surface Inspection			✓
Surface to Model Inspection			✓
Inspection Map Analyzer			✓
Profile Matcher			✓
EasyProfile			✓
Fitting		✓	✓
3D Inspection			✓
3D Inspection Analyzer			✓

Note: (*) In **RealWorks Viewer**, there is no **OfficeSurvey** menu.


Cutting Plane Tool

The **Cutting Plane Tool** enables to cut a selected entity (point cloud or mesh) with a plane whose position and orientation need to be defined by the user. The result of this cut is a sectioned point cloud or a polyline. This tool can be used alone as a main tool or inside a main tool as a sub-tool. In the latter case, it is mainly used as a visual quality checking tool and no results can be created.

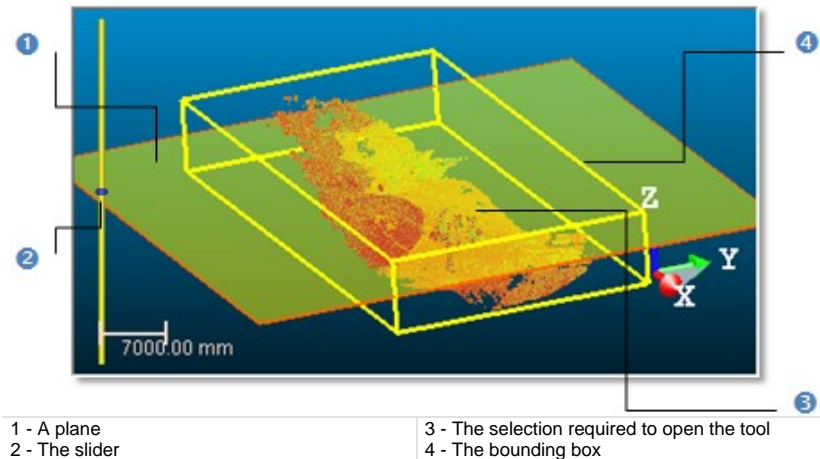
Opening the Tool

To open the tool, you should first select either a point cloud or a mesh. If a point cloud is selected, you can clean it by removing for example parasite points or reduce it by sampling or by fencing an area. These can be done thanks to the two following sub-tools (**Sampling** and **Segmentation**). If a mesh is selected, these two sub-tools are unavailable.

To Open the Tool:

1. Select an object (point cloud or mesh) from the **Project Tree**.
2. From the **OfficeSurvey** menu, select **Cutting Plane Tool** . The **Cutting Plane** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window and is composed of four parts. The first part (**Define Cutting Plane**) contains tools for defining a plane. The second part (**Define Slice**) allows you to set a thickness for the plane defined in the previous step and to choose between **Single Slice** and **Multi Slice**. The third part (**Control**) is to preview the result(s). The last step is to save the result(s).



A planar view with a 2D grid that you can customize opens below the **3D View**. A plane perpendicular to the screen and a slider appear in the **3D View**. This plane runs across a point that corresponds to the centre of the selection (required to open the tool). This point sets the altitude (**Offset**) of that plane. In this case, it is 0 mm*. The bounding box that highlights the selection delineates the size of that plane.

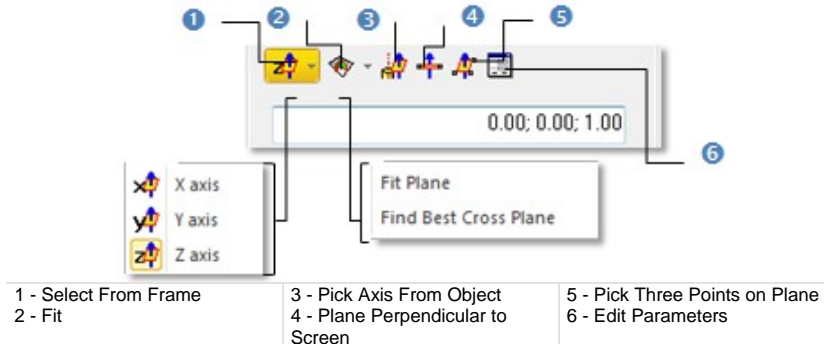
3. From now, you should define a plane. First, you should set its orientation. Then you can define its position.

Note: (*) The current unit of measurement is in Millimeter. You can change it in the **Preferences** dialog.

Defining the Orientation of a Plane

There are four methods for precisely defining the orientation of a plane. The first method is to select an axis (from the active frame) so that the initial plane becomes perpendicular to it. The second method is by picking an object's local frame. The third method is to specify the coordinates of its normal vector. The fourth method is to edit parameters.

There are three methods for visually defining the orientation of a plane. The first method is to pick two points. The initial plane will pass through the line defined by these two points and perpendicular to the screen plane. The second method is to pick three points. The initial plane will pass through the plane drawn by these three points. The third method is to define a plane. The initial plane will be parallel to the defined plane.



Setting Coordinates







To Set Coordinates:

- Enter the coordinates of a plane's normal vector in the **Normal** field.

Selecting a Frame Axis

This method consists of selecting an axis from the active frame as **Normal** direction. The initial plane will be moved so that its **Normal** will be parallel to the selected axis. Its position in the 3D scene will be kept and its **Offset** (altitude) will be reset.


To Select a Frame Axis:

1. Click on the **Set from Frame** pull down arrow.
2. Choose among **X Axis** , **Y Axis**  and **Z Axis**  (in the X, Y and Z **Coordinate System**).
3. Or choose among **North Axis** , **East Axis**  and **Elevation Axis**  (in the **North, East and Elevation Coordinate System**).

Picking an Object Local Frame

This method consists of picking an object's local frame. The initial plane will be moved so that its **Normal** will be parallel to the picked local frame. Its position in the 3D scene and its **Offset** (altitude) will be set by the picked point.

To Pick an Object Local Frame:


1. Click the **Pick Axis from Object**  icon. The initial cutting plane disappears from the **3D View**.
2. In the **Workspace** window, click on the **Models** tab.
3. Right-click on the selection to display the pop-up menu.
4. Select **Display Geometry**.
5. Click an object.

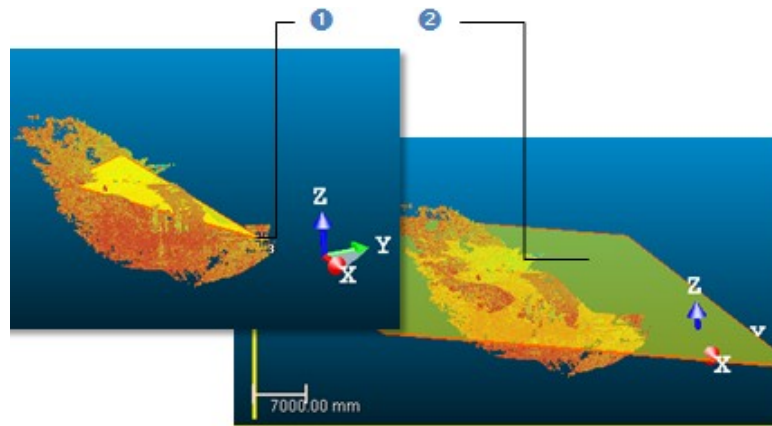
Note: Pressing **Esc** while the picking is in progress will cancel the selected point(s) and makes appeared the last defined plane.

Picking Three Points

This method consists of picking three points. The initial plane will be moved so that it will pass through the three picked points. Its position in the 3D scene and its **Offset** (altitude) will be the barycentre of the three picked points.

To Pick Three Points:

1. Click the **Pick 3 Points on Plane**  icon. The **Picking Parameters** toolbar appears.
2. Pick three points (free or constrained). Picking is always on the displayed object.



1 - Picked points


2 - The cutting plane passes through the picked points

Note: Pressing **Esc** while the picking is in progress will cancel the selected point(s) and makes appeared the last defined plane.

Picking Two Points

This method consists of picking two points. The initial plane will be moved so that it will pass through the two picked points and perpendicular to the screen.



To Pick Two Points:

1. Click the **Plane Perpendicular to Screen**  icon.
2. Pick two points. No need to pick on the displayed object.

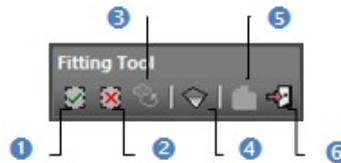
Note: Pressing **Esc** while the picking is in progress will cancel the selected point(s) and makes appeared the last defined plane.

Fitting With a Geometry

To Fit With a Geometry:

1. Click on the **Fit** pull-down arrow.
2. Choose one of the following:
 - Select **Fit Plane** . The **Fitting Tool** toolbar appears as well as an information window at the top right corner of the **3D View**.
 - Select **Find Best Cross Plane** . The **Fitting Tool** toolbar appears as well as an information window at the top right corner of the **3D View**.

Fitting a Plane



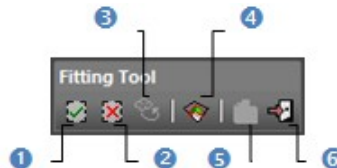
- | | |
|--|--|
| 1 - In (i)
2 - Out (o)
3 - Display Un-partitioned Points | 4 - Plane
5 - Create Fitted Geometry
6 - Close Tool (Escape) |
|--|--|

To Fit a Plane:

1. Fence a set of points for which you want to fit with a plane.
2. Click the **Plane** icon. Kept points are fitted with a plane.

Note: It is not necessary to fence a set of points; fitting a plane can be applied to the entire point cloud.

Finding the Best Cross Plane



- | | |
|--|---|
| 1 - In (i)
2 - Out (o)
3 - Display Un-partitioned Points | 4 - Plane Normal
5 - Create Fitted Geometry
6 - Close Tool (Escape) |
|--|---|


To Find the Best Cross Plane:

1. Fence a set of points with which you want to fit a plane.
2. Click the **Plane Normal** icon. Kept points give the projection plane's normal direction.

Note: It is not necessary to fence a set of points; finding the best cross plane can be applied to the entire point cloud.

Editing Parameters

To Edit Parameters:


1. Click the **Edit Parameters**  icon. The **3D Plane Editing** dialog opens.
2. Click on the pull down arrow and do one of the following:
 - Chose **Normal + Point** to define a normal and a position.
 - a) Enter a direction in the **Normal** field.
 - b) Enter a point position in the **Point** field.
 - Chose **Point to Point** to define two points. **Points(To) - Point (From)** defines a normal and **Point (From)** gives a position.
 - a) Enter a point position in the **Point (To)** field.
 - b) Enter a point position in the **Point (From)** field.
3. Click **OK**. The **3D Plane Editing** dialog closes.

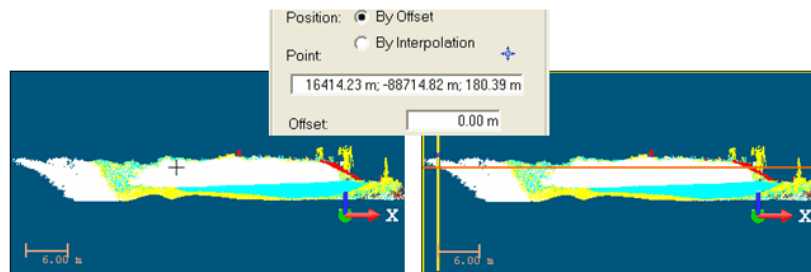
Defining the Position of a Plane

There are three methods for defining a plane's position. The first method is to define this position by one point. You can either pick this point in the displayed scene or key in its coordinates. We call this method **By Offset**. The second method is to define this position by two points. This can be used, for example, to find the center plane of two parallel walls of a building. We call this method **By Interpolation**. You can either pick the points in the displayed cloud or give their exact coordinates. Then you can use the **Ratio** field to define the position of the cutting plane precisely between these two points. **Ratio 0** will put the position coincident with the first point, and **Ratio 1** with the second point. The third method is to use the slider at the left side of the **3D View** to move the plane. This can be used to visually define the position of a plane and is often used for visual checking of registration quality.

By Offset

To Define a Position By Offset:

1. Check the **By Offset** option.
2. Enter a point position in the **Point** field.
3. Or click **Pick Point** . The **Picking Parameters** toolbar appears in 3D constraint mode.
4. Pick one point in the 3D scene.
 - The cutting plane passes through that point.
 - The **Offset** value is set to 0.00*.




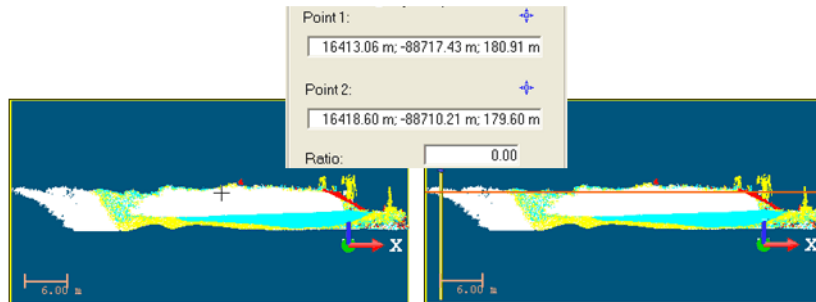
Notes:


- (*) In the current unit of measurement. You can change it in **Preferences**.
- Moving the slider **Up** (or **Down**) will increase (or decrease) the **Offset** value.

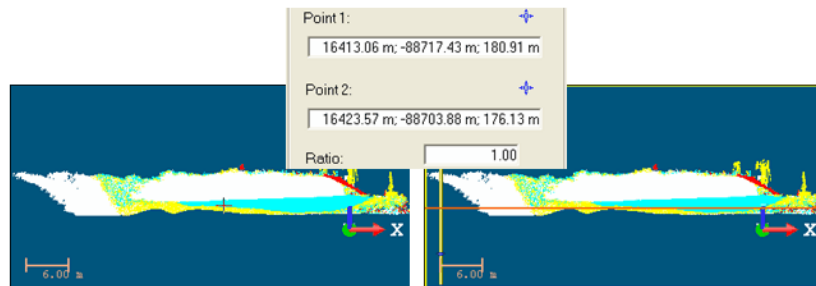
By Interpolation

To Define a Position By Interpolation:

1. Check the **By Interpolation** option.
2. Enter a point position in the **Point 1** field.
3. Or click the **Pick First Point**  icon. The **Picking Parameters** toolbar appears in 3D constraint mode and the cursor becomes as shown below.
4. Pick a point in the 3D scene.
 - The initial plane will pass through **Point 1**.
 - The **Ratio** value is equal to 0.



5. Enter another point position in the **Point 2** field.
6. Or click the **Pick Second Point**  icon. The **Picking Parameters** toolbar appears in 3D constraint mode and the cursor becomes as shown below.
7. Pick another point in the 3D scene.
 - The initial plane will pass through **Point 2**.
 - The **Ratio** value is equal to 1.



8. Define the exact position of the plane by entering a value between 0 and 1 in the **Ratio** field.

Defining a Slice

After defining a plane, you now need to decide whether to perform a **Single Slice** or **Multiple Slice** cutting. In the case of a **Single Slice**, the cutting will be along the defined plane. You will then choose the thickness of the slice in order to cut the point clouds. In the case of a **Multiple Slice**, you also need to define the interval between two slices. The slices will be propagated from the position of the defined plane in two directions with the given interval. The number of slices indicated at the bottom of the dialog is calculated in such a way that slices will span the whole range of the point cloud along the normal direction of the defined plane.



Single Slice

To Define a Single Slice:

1. Check the **Single Slice** option.
2. Enter a value in the **Thickness** field.
3. Or use the **Up**  and **Down**  buttons to select a value.

Multiple Slice

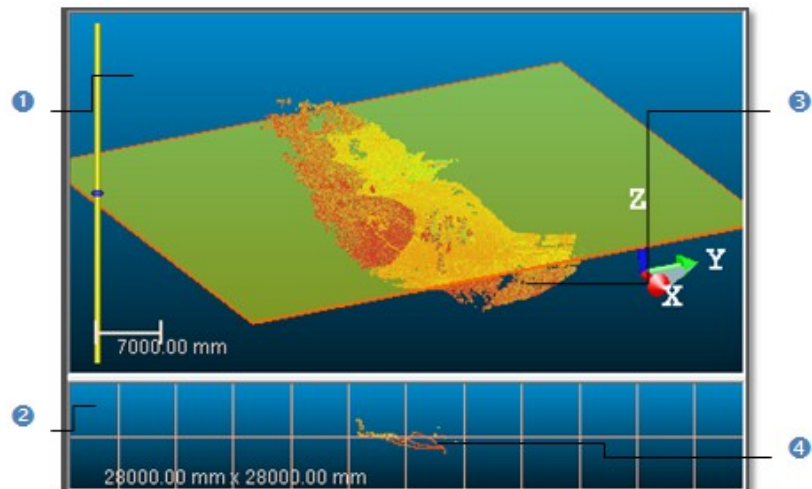
To Define a Multiple Slice:

1. Check the **Multiple Slice** option.
2. Enter a value in the **Thickness** field.
3. Enter a value in the **Interval** field.
4. Or use the **Up**  and **Down**  buttons to select a value.

Note: The **Thickness** value cannot exceed the **Interval** value. It can only be the same.

Previewing a Single Slice

If the **Single Slice** option has been chosen, the **Preview** button remains inactive. The top window displays the selected object (point cloud or mesh) with the defined plane. The planar view displays in real time the cutting result. There is by default a 2D Grid superposed on the displayed sliced cloud; you can choose to change the grid size or to hide it by using the corresponding items from the pop-up menu. Note that the **View Manager** toolbar appears at the bottom of the **3D View**. You can use the icons to change the configuration of the two sub-views.



1 - The 3D View
2 - The Planar View

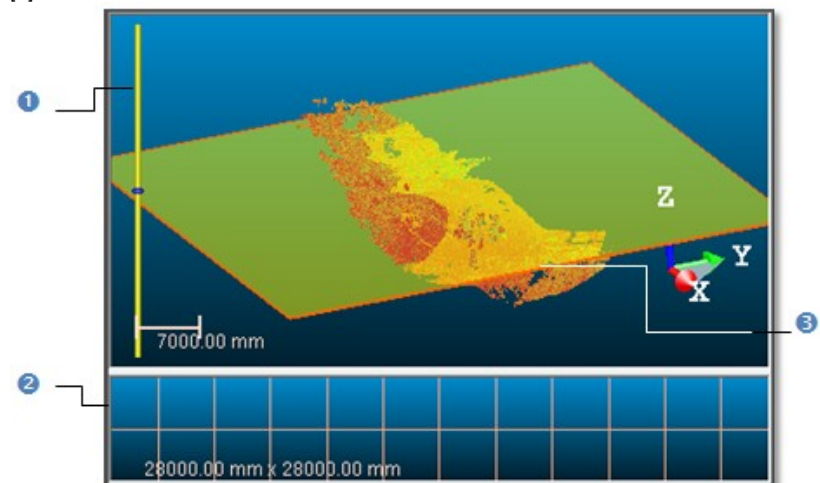
3 - The point cloud
4 - A point cloud slice

Note: If the selected object is a mesh, the cutting result will be a polyline. Otherwise the result will be a cloud slice.

Previewing a Multiple Slice

If the **Multiple Slice** option has been chosen, the **Preview** button switches from inactive to active and the planar view becomes empty of contents (see [A]). Clicking **Preview** will display the cutting results in the **3D View** and will remove the defined plane representation from it. The active slice, the one in pink in the **3D View**, is shown in the planar view (see [B]). The **Control** tools (see **Step 3** of the **Cutting Plane** dialog) become active. In the planar view, there is by default a 2D Grid superposed on the displayed sliced cloud; you can choose to change the grid size or to hide it by using the corresponding items in the pop-up menu. Note that the **View Manager** toolbar appears at the bottom of the **3D View**. You can use the icons in this toolbar to change the configuration of the two sub-views.

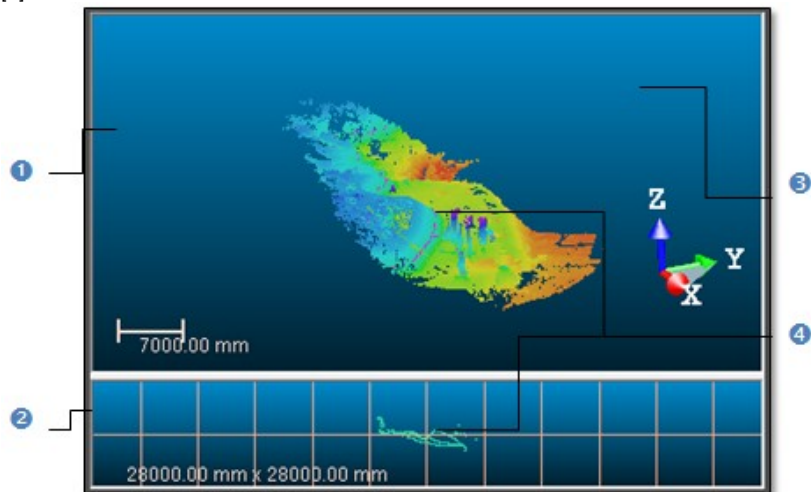
[A]



1 - 3D view
2 - Planar view

3 - Point Cloud

[B]



1 - The 3D View
2 - The Planar view

3 - A set of point cloud slices
4 - The Active point cloud slice

Note: If the selected object is a mesh, the cutting results will be polylines. Otherwise the results will be cloud slices.

Building Polylines

To Build a Polyline from a Single Slice:

1. Click **2D-EasyLine**. The **2D-EasyLine** dialog opens.
2. Build a polyline from the slice.
3. Click **Apply**. The **2D-EasyLine** dialog closes.

To Build Polylines from a Multiple Slice:

1. To select a slice, do one of the following:
 - Pick a slice in the **3D View**.
 - Use the **Control** tools as follows:
 - Click **Display Next Slice** to view the one after the active slice.



1 - Display First Slice
2 - Display Last Slice

3 - Display Next Slice
4 - Display Previous Slice

- Click **Display Previous Slice** to view the one before the active slice.
 - Click **Display First Slice** to view the first slice.
 - Click **Display Last Slice** to view the last slice.
 - Key in a number and press **Enter**.
2. Click on the **2D-EasyLine** button. The **2D-EasyLine** dialog opens.
 3. Build a polyline from the slice.
 4. Click **Apply**. The **2D-EasyLine** dialog closes.

Notes:

- Instead of clicking **Display Next Slice** or (**Display Previous Slice**), you can also use the **Up** (or **Down**) key on your keyboard.
- You can multi-select cloud slices (or polylines) in the **3D View** using the **Ctrl + A** shortcut keys, open the **2D-EasyLine™ Tool** and build polylines based on the selected cloud slices or polylines.
- The **2D-EasyLine Tool** is not available in **RealWorks Viewer**.

Saving the Cutting Result(s)

If you are satisfied with the cutting result(s), you can create it (or them) in the database. When selecting **Single Slice**, only an object will be created. When selecting **Multiple Slice**, a folder* will be created in which each slice result (including sliced cloud and the polyline if it exists) will be created as an object. You can create as many cutting planes as you need without leaving the **Cutting Plane Tool**.

To Save the Cutting Result(s):

1. Click **Create**.
2. Click **Close**.

Notes:


- **Close** can also be selected from the pop-up menu.
- Leaving the **Cutting Plane Tool** without saving the result(s) will make appeared a warning message.
- (*) The folder default name is **Cross-Cut-Interval -XX-Th YY**. **XX** is the **Interval** value and **YY** the **Thickness** value.

Contouring Tool

The purpose of the **Contouring Tool** is to create iso-contours from 2.5 point cloud(s) or mesh(es) along the **Z Axis** (or **Elevation Axis**) of the active coordinate frame. The output of this tool will be a set of contours, each of which is represented by a polyline lying on the plane situated at the corresponding elevation.

Opening the Tool

To Open the Tool:

1. Select an object (point cloud or mesh) from the **Project Tree**.
2. In the **OfficeSurvey** menu, select **Contouring Tool** . The **Contouring** dialog opens.

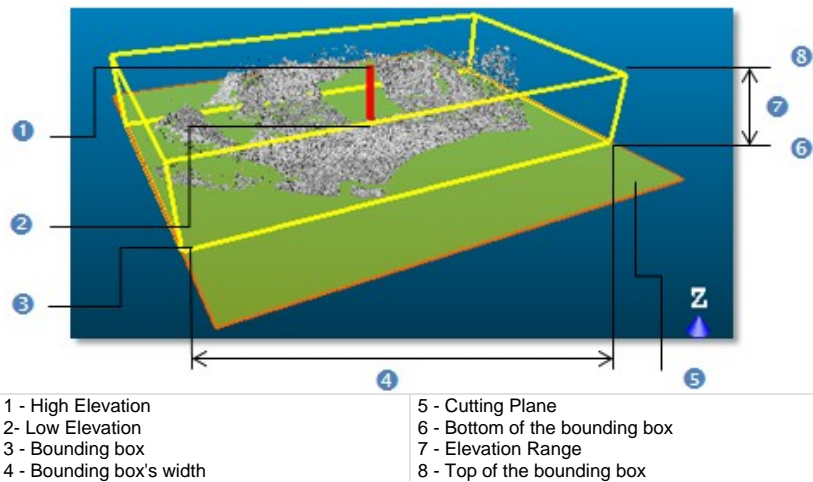
This dialog opens as the third tab of the **WorkSpace** window and is composed of several parts. The first part contains two sub-tools (**Segmentation** and **Sampling**)*. If the input is a point cloud, you can clean it by removing parasite points (or reduce its size by simplifying it). The way that point cloud is rendered changes. Its **Rendering** swaps to **White Color**. If the input is a mesh, both sub-tools are grayed-out and its **Rendering** remains unchanged. The second part enables to define an elevation range. The third part enables to set a tolerance for contour decimation. The fourth part is to define principal contours and the last part lets you to display and save the contour creation results.

Notes:

- You can leave the **Contouring Tool** by pressing **Esc** or by right-clicking anywhere in the **3D View** to display the pop-up menu and select **Close**.
- (*) The results issued from the use of both sub-tools cannot be saved. The **Create** command is deactivated.

Defining an Elevation Range

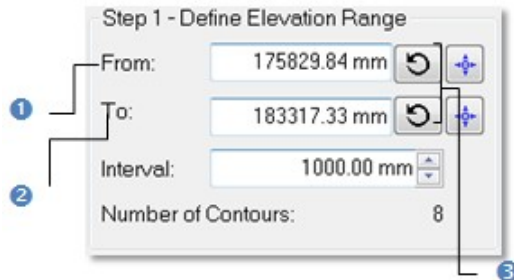
A default elevation range is setup so that the **High Elevation** and **Low Elevation** correspond to the top and bottom of the bounding box that highlights the selected object. This elevation range is represented by a graduated vertical bar. A **Cutting Plane** perpendicular to the **Z (or Elevation) Axis** of the active coordinate frame runs across the **Low Elevation** of the default **Elevation Range** and its size (only width) is given by the bounding box size (only width).



Setting Low and High Elevation Values

To Set the Low and High Elevation Values:

1. Enter a value in the **From** field and press **Enter**.




1 - Low Elevation
2 - High Elevation

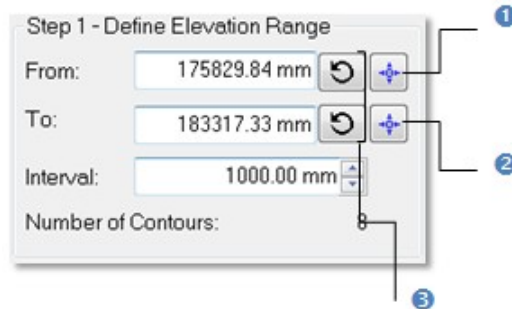
3 - Icons for resetting the initial value

2. Enter a value in the **Top** field and press **Enter**.
3. If required, get back the initial **Low** (or **High**) **Elevation** value by clicking the **Reload Initial Low Elevation** (or **Reload Initial High Elevation**) icon.

Picking Low and High Elevations

To Pick Low and High Elevations:

1. Click the **Pick Low Elevation**  icon. The **Picking Parameters** toolbar appears. The **Cutting Plane** and the graduated vertical bar are removed from the **3D View**.
2. Pick a point on displayed object in the **3D View**. It's up to the user to pick a point (freely or with constraint).



1 - Pick Low Elevation
2 - Pick High Elevation

3 - Reload Initial Elevation

3. Repeat the two upper steps for the **High Elevation**.
4. If required, get back the initial **Low** (or **High**) **Elevation** value by clicking the **Reload Initial Low Elevation** (or **Reload Initial High Elevation**) icon.

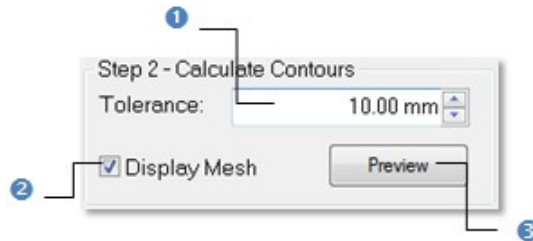
Interval

To Define an Interval Value:

1. Enter a value in the **Interval** field and press **Enter**.
2. Or use the **Up**  and **Down**  buttons to select a value.

Calculating Contours

This step enables to set a value for the **Tolerance** parameter, preview the contouring result and if required hide the input data.



1 - Contour decimation tolerance
2 - The Hide / Display option

3 - Preview the contouring results

Tolerance

The **Tolerance** value is used to decimate contours. The polyline of each contour will be decimated in such a way that the vertices of the original polyline will be inside the defined tolerance range.

To Define the Tolerance Parameter:

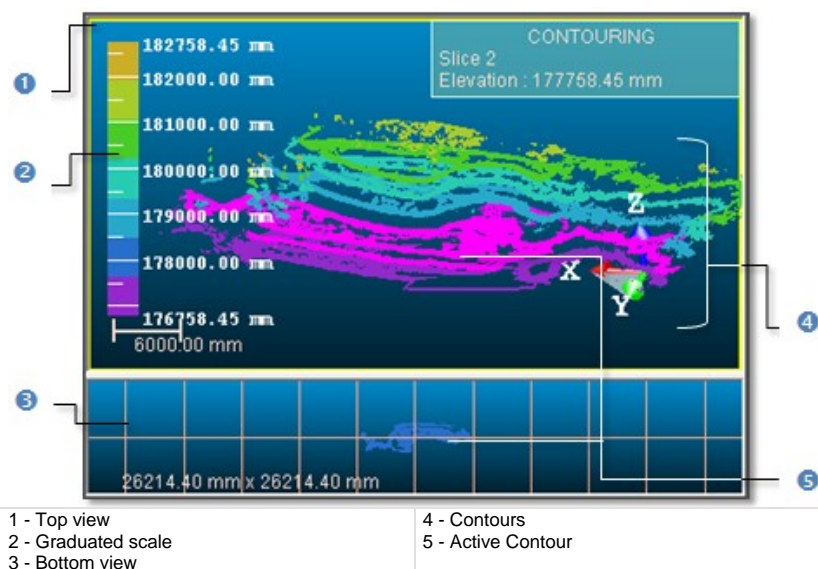
1. Enter a value in the **Tolerance** field.
2. Or use the **Up**  and **Down**  buttons to select a value.

Previewing

After defining an elevation range and the decimation tolerance, you can use the **Preview** button to visualize the contouring result. At this moment, the **3D View** will be split into two sub-windows: one for visualizing the data with the contours superposed in 3D and another for displaying each contour in a planar view.

To Preview Contours:

- Click **Preview**. The results are shown in two sub-windows.



In the top window, each contour is displayed with a color. A graduated scale at the left side gives the altitude information for a given color. An information box at the right top corner displays the active (selected) contour's properties: Order and Elevation. In bottom view, there is by default a **2D Grid** superposed (if not hidden previously) on the displayed contour. You can choose to change the **2D Grid**'s size or to hide it by using the corresponding items from the pop-up menu. An information box at the right top corner displays in addition to the properties listed above the **Fitted Polyline**'s size. A **View Manager** toolbar appears at the bottom of the **3D View**. You can use the icons in this toolbar to change the configuration of the 2D sub-views.

Hiding/Displaying Input Data

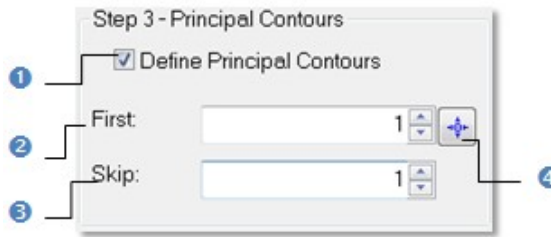
If the input data (required to open the **Contouring Tool**) is a point cloud, you can remove its representation from the **3D View** by clearing the **Display Cloud** option. If the input data is a mesh, this option will become **Display Mesh**. You can clear the option to hide the mesh representation.

Defining Principal Contours

You can choose and assign some of the contours as principal contours. The remained contours are then considered as intermediate contours.

To Define Principal Contours:

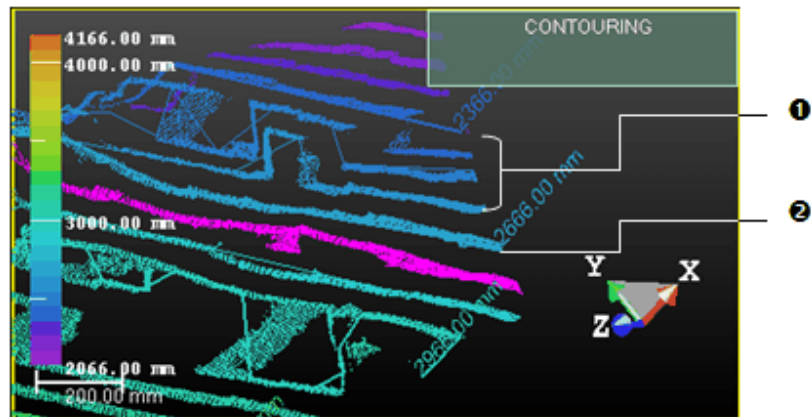
1. Check the **Define Principal Contours** option. The **First** and **Skip** fields become enabled as well as the **Pick Principal Contour** icon. A label appears next to each contour.



1 - The Define Principal Contours option
2 - Define the first principal contour by entering a number

3 - Define the number of contours to skip
4 - Pick Principal Contour

2. Enter a value in the **First** field and press **Enter**.
3. Or click the **Pick Principal Contour** icon. The **Picking Parameters** toolbar appears.
4. Go to the top view and pick a contour - using the constraint or not.



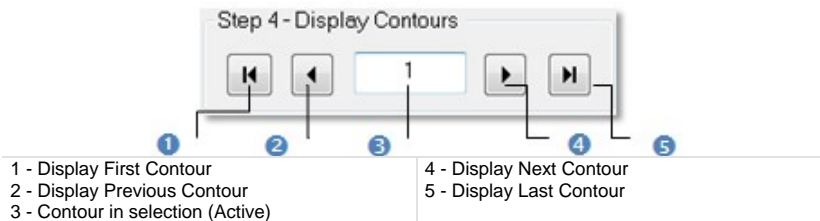
1 - Intermediate contours have no label

2 - Each principal contour is shown with a label

5. Enter a value in the **Skip** field and press **Enter**.

Displaying Contours

The first contour is the active one. It is displayed in the bottom view and appeared in pink in the top view.



If the active contour is other than the first contour, you can use the **Up** and **Down** keys of your keyboard (or the **Display Previous Contour** and **Display Next Contour** buttons in the **Step 4** of the **Contouring** dialog) to display the next and the previous contour in the bottom view. Be sure to first select the top view to be able to use the **Up** and **Down** keys. It should have a yellow frame. Clicking the **Display First Contour** and **Display Last Contour** buttons will set the first and last contour as active (selected). You can key in a contour's order in **Step 4** to select it. Do not forget to validate by pressing the **Enter** key.

Tip: You can visualize several contours in the bottom view. Please, select the ones you need from the top view by combining the use of **Ctrl** key with left-clicking; or multi-select all using the **Ctrl + A** shortcut keys.

Creating Contours

Once you are satisfied with the contouring results, you can use the **Create** button to create them in the database. A folder named "Cross-Map" is created and put under the current active folder, in which all contours are put. The **Interval** parameter is appended to the folder name. Each contour is named by combining a default name string "Cross-Map" with the elevation information and of **Fitted Polyline** type.

You can create as many contours as required without leaving the **Contouring Tool**. If you decide to leave this tool without creating any contours, a message appears and prompts you to confirm, undo or cancel the action you are going to perform.

To Create Contours:

1. Click **Create**.
2. Click **Close**.

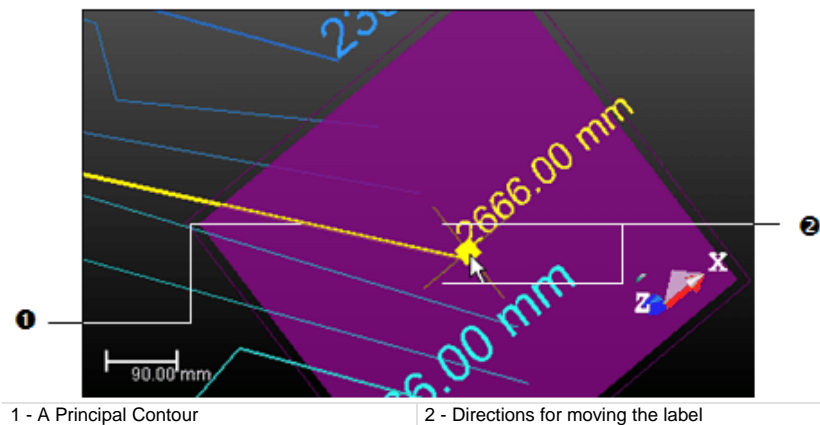
Note: You can leave the **Contouring Tool** by pressing **Esc** or by right-clicking anywhere in any window and select **Close** from the pop-up menu.

Manipulating Labels

Each principal contour has a label which contains the length information in text. You can move that label to any location in the **3D View**. Note that you should first create the contours in the database and leave the **Contouring Tool** to be able to manipulate the labels.

To Manipulate a Label:

1. Select a **Principal Control** from the **3D View** by picking it.
2. From the **OfficeSurvey** menu, select **Show Manipulators**. A white square appears beside the selected principal contour's label.
3. Pick on the white square. It becomes yellow.
4. Drag and drop the square from its current position to a new one. The label will move consequently.



1 - A Principal Contour

2 - Directions for moving the label

5. From the **OfficeSurvey** menu, select again **Show Manipulators** to leave this tool.

Note: Instead of selecting **Show Manipulators** from the **OfficeSurvey** menu, you can also click its corresponding icon in the **Tools** toolbar.

2D-EasyLine Tool

This tool allows you to create polylines from point cloud's slice(s) that result(s) from the use of the **Cutting Plane Tool** or from planar polylines. The resulting polylines can contain only segments or a combination of segments and circular arcs. The **2D-EasyLine Tool** can be used as a standalone tool or as a sub-tool inside the **Cutting Plane Tool**.

Opening the Tool


The behavior of the **2D-EasyLine Tool** depends upon the input data.

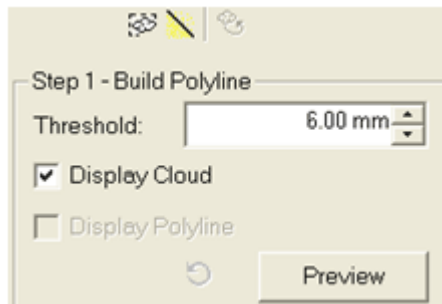
If the input data is a pure point cloud slice, **Step 1** of the **2D-EasyLine** dialog looks as shown in [A]. You can use either the automatic procedure or the manual procedure (**Step 2** in the **2D-EasyLine** dialog) for modeling polylines.

[A] When a Point Cloud Slice is Selected:

1. In the **Cutting Plane** dialog, select a point cloud slice.
2. Click on the **2D-EasyLine** button.

Or


3. Select a point cloud slice from the **Models Tree**.
4. From the **OfficeSurvey** menu, select **2D-EasyLine Tool** .

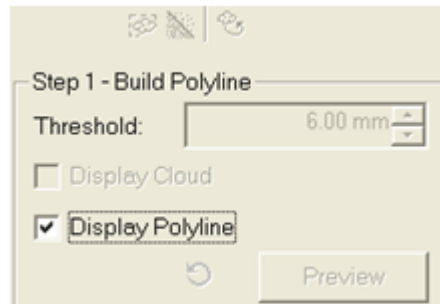


- Both the number of segments and the number of arcs are equal to zero.
- The sub-tools (**Sampling** and **Segmentation**) as well as the **Display Cloud** option and the **Preview** button are enabled.

If the input data is a planar polyline (without points inside), **Step 1** of the **2D-EasyLine** dialog looks as shown in [B]. You can only edit the planar polyline (**Step 2** in the **2D-EasyLine** dialog).

[B] When a Planar Polyline is Selected:


1. Select a planar polyline (with no points inside) from the **Models Tree**.
2. From the **OfficeSurvey** menu, select **2D-EasyLine Tool** . The **2D-EasyLine** dialog opens as shown below.

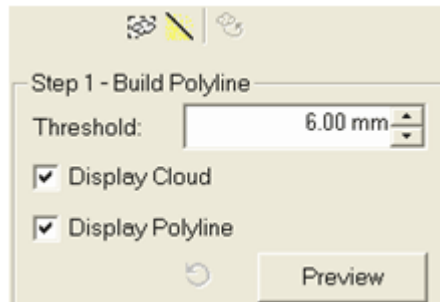


- The number of segments and the number of arcs inside the planar polyline are shown in text in **Step 2**.
- The sub-tools (**Sampling** and **Segmentation**) as well as the polyline computation parameter (**Threshold**) and the **Preview** button are all dimmed.

If the input data is a planar and fitted polyline (with points inside), **Step 1** of the **2D-EasyLine** dialog looks as shown in [C]. You can choose between the two procedures for modeling new polylines or use the editing tools to modify the planar and fitted polyline.

[C] When a Planar and Fitted Polyline is Selected:

1. Select a planar and fitted polyline (with points inside) from the **Models Tree**.
2. From the **OfficeSurvey** menu, select **2D-EasyLine Tool** . The **2D-EasyLine** dialog opens as shown below.





- The number of segments and the number of arcs inside the planar polyline are shown in text in **Step 2**.
- The sub-tools (**Sampling** and **Segmentation**) as well as the polyline computation parameter (**Threshold**), the **Preview** button and the two displayed options (**Display Cloud** and **Displayed Polyline**) are all enabled.

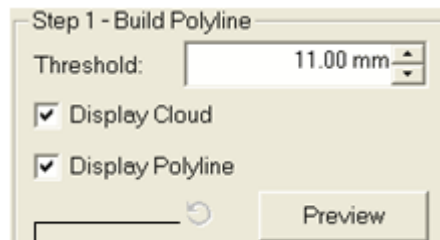
The selection required to open the tool - planar polyline (fitted or not) or point cloud slice - is displayed in a planar view (2D) with the 2D grid in superimposition. This means that the planar polyline (fitted or not) or the point cloud slice is locked in 2D; you can navigate through it (like performing a zoom, panning or rotation). You can use the **View Manager** toolbar to show the 3D sub-view and the planar view at the same time, or to switch between them. In the planar view, you can use the pop-up menu to modify the size of the 2D grid or to hide it.

Modeling Automatically Polylines

The automatic modeling procedure uses an algorithm which approximates points of the selected cloud slice (or fitted polyline) with segments. The **Threshold** parameter will be used so that points of the selected cloud (or fitted polyline) which are inside the **Threshold** will be taken into account for the automatic modeling procedure. You can change the modeling parameter and use **Preview** where several attempts are required*.

To Model Automatically Polylines:

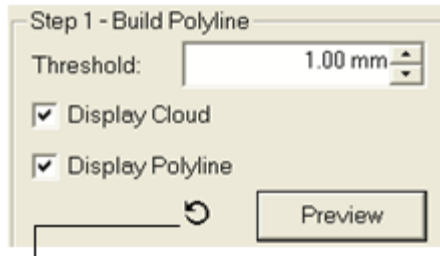
1. Enter a value in the **Threshold** field and press **Enter**.
2. Or select a value using the **Up**  (or **Down** ) button.
3. Click on the **Preview** button.
 - If the input data is a pure slice cloud, new polylines are modeled according to the value set in **Threshold** and are displayed in the **3D View**. You have choice between the two display options (**Display Cloud** and **Display Polyline**) and the **Reload Initial Cloud - Polyline** is grayed out.



The Reload Initial Cloud - Polyline is dimmed

Note: The other editing tools in **Step 2** in the **2D-EasyLine** dialog become enabled.

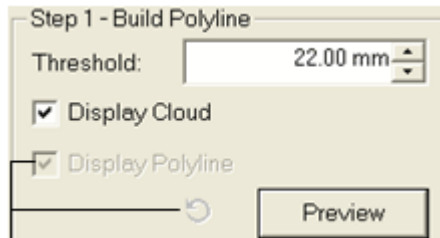
- If the input data is a fitted polyline, new polylines are modeled according to the value set in **Threshold** and are displayed in the **3D View**. You have choice between the two display options (**Display Cloud** and **Display Polyline**) and the **Reload Initial Cloud - Polyline** becomes enabled.



The Reload Initial Cloud - Polyline is enabled

Note: **Reload Initial Cloud - Polyline** cancels the computed polyline(s) and reloads the initial polyline.

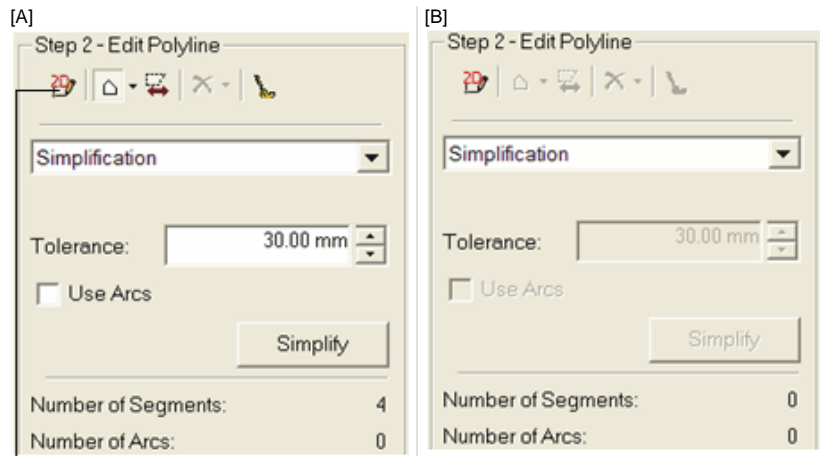
- For both, the number of segments and the number of arcs will be updated according to the **Threshold** value. (*) If the **Threshold** value is too high, new polylines cannot be built. The number of segments and the number of arcs fall to zero. The **Step 1** becomes as shown below and only points are displayed in the **3D View**.



Both the Reload Initial Cloud - Polyline and the Display Polyline are dimmed


Modeling Manually Polylines

You can use the **Polyline Drawing Tool** to model by hand polylines. You can model polylines with only segments or a combination of segments and circular arcs. If the input is a planar polyline (fitted or not), the **Step 2** looks as shown in [A]. If the input is a pure slice cloud, it is as shown in [B].



Polyline Drawing Tool

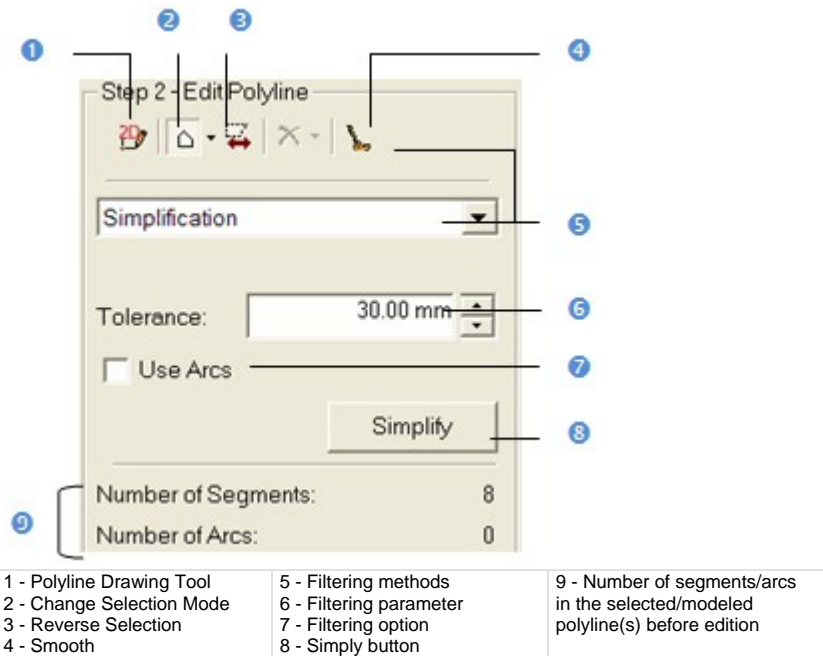
To Model Manually Polylines:

1. Click **Polyline Drawing Tool** . The **Drawing Tool** and **Picking Parameters** (in 2D constraint mode) toolbars appear.
2. Pick a series of points (on displayed points or not) to draw a polyline.
 - After modeling by hand a polyline, the number of segments and the number of arcs that are inside are shown in text.
 - If the input is a planar polyline (fitted or not), **Reload Initial Cloud - Polyline** becomes enabled. Click on it to reload the initial polyline.
 - If the input is a pure slice cloud, the other editing tools become enabled.

Note: The **Create** button in the **Drawing Tool** toolbar is dimmed. To validate the polyline, choose **Close Tool** from the toolbar or from the pop-up menu.

Editing Polylines

You can edit the modeled polyline. The edition can be done manually or thanks to filters. If you want to use the by-hand method; choose the **Polyline Drawing Tool** to move or add vertices, delete a segment, etc. (see the **Polyline Drawing Tool**) or use the **Edit Polyline** tools to define which parts of the modeled (or selected) polyline you want to keep. If you want to use the by-filter method, any selection is required and you can apply filters like simplification, smoothing, filling holes, etc.



Selecting Items

We describe hereafter the different selection modes that you can use for editing polyline(s). There are four modes in all. Before selecting items from the displayed polyline(s), only two modes can be used: **Standard Selection** and **Multi Selection**. Once a first selection is made, the two other modes become enabled: **Partial Deselection** and **Partial Reselection**.

Items in the polyline(s) are mainly arcs and segments. The information box at the top right-corner of the **3D View** displays in text the number of arcs and the number of segments in the selection. The **2D-EasyLine** dialog displays the total number of arcs and segments in the polyline(s). The numbers in the information box will be updated automatically each time you add or subtract items from the polyline(s).

To Select Items:

1. Select item(s).
2. Add new selected item(s) to previous one(s).

Once a first selection is made, the two other selection modes - **Partial Deselection** and **Partial Reselection** - in **Change Selection Mode** become active as well as **Change Deletion Mode**.

3. Subtract new selected item(s) from previous one(s).
4. Intersect new selected item(s) with previous one(s).

Clicking **Reverse Selection** will set unselected items as selected and those are selected as unselected. If any polygonal fence has been drawn, clicking **Reverse Selection** will then select the whole polyline in the **3D View**.

Notes:


- The polygonal fence should contain at least one item (segment or arc) in its entirety so that this item can be selected.
- You can undo a selection by using the **Undo** command.
- Selecting **New Fence** from the pop-menu (or pressing **Esc**) will undo the polygonal fence in progress.
- Selecting **Clear Selection** from the pop-menu will clear the polygonal fence from the polyline(s).

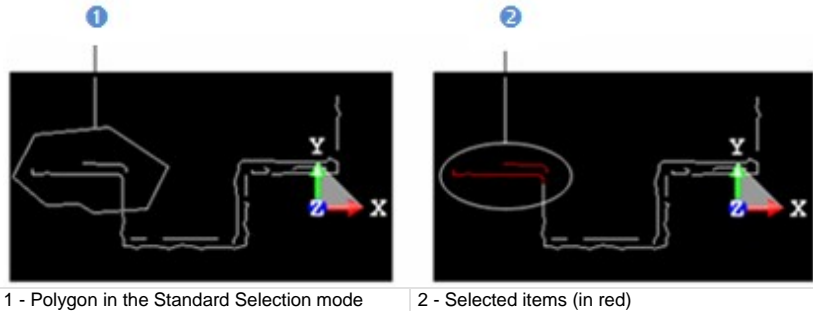
Tips:

- Instead of double-clicking to close the polygonal fence, you can also right click anywhere in the **3D View** window and select **End Fence** from the pop-up menu.
- You can select **Clear Selection** from the pop-up menu to cancel the selection.

Standard Selection Mode


To Select:

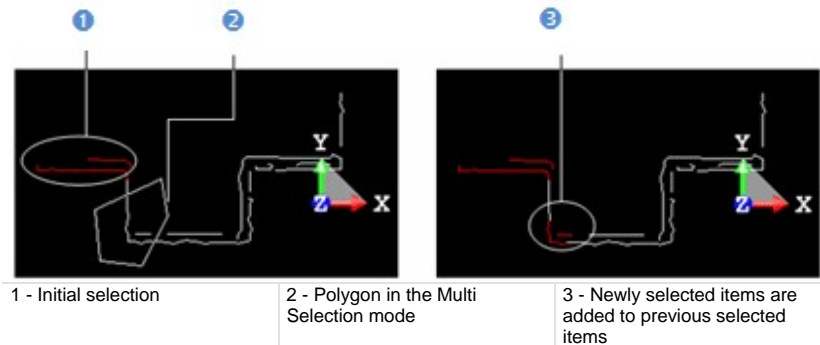
1. Click on the **Change Selection Mode** pull down arrow.
2. Choose **Standard Selection Mode** .
3. Draw a polygonal fence.



Multi-Selection Mode

To Multi-Select:


1. Click on the **Change Selection Mode** pull down arrow.
2. Select **Multi-Selection Mode** .
3. Draw a series of polygonal fence.

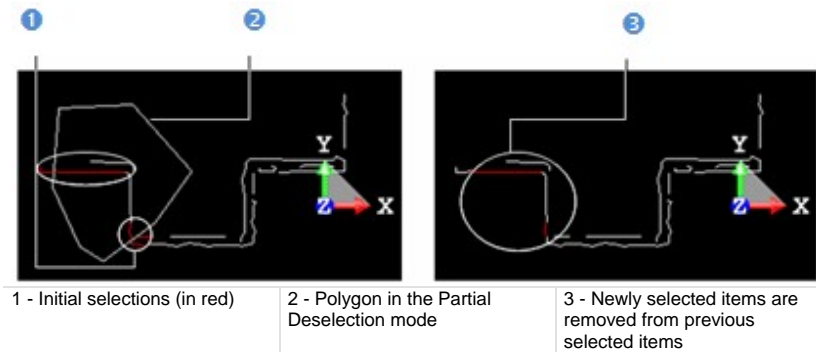


Tip: In the **Standard Selection Mode**, hold the **Ctrl** (or **Shift**) key pressed and pick a point to define the first vertex of a polygonal fence.

Partial Deselection Mode

To Partial Deselect:


1. Click on the **Change Selection Mode** pull down arrow.
2. Select **Partial Deselection Mode** .
3. Draw a polygonal fence.
 - If the polygonal fence contains some of the previously selected items. These items are deselected and the others remain selected.
 - If the polygonal fence contains any of the previously selected items. No subtraction will be performed.

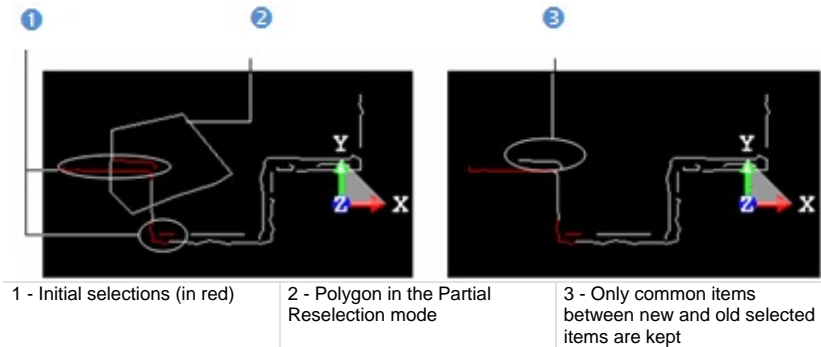


Tip: In the **Standard Selection Mode**, hold the **Alt** key pressed and pick a point to define the first vertex of a polygonal fence.

Partial Reselection Mode

To Partial Reselect:

1. Click on the **Change Selection Mode** pull down arrow.
2. Select **Partial Reselection Mode** .
3. Draw a polygonal fence.
 - If the polygonal fence contains some of the previously selected items; then common items remain selected and the others are unselected.
 - If the polygonal fence contains any of the previously selected items. No intersection will be performed.




Tip: In the **Standard Selection Mode**, hold the **Ctrl + Alt** keys pressed and pick a point to define the first vertex of a polygonal fence.

Deleting Items

You can now continue editing the selected/modeled polyline(s) using the available filters. Filters can be separated into two categories. The first category contains filters for which you do not need to set parameters. These filters are: **Delete Selection**, **Delete Selection Filling Holes** and **Smooth**.

Delete Selection


To Delete Items:

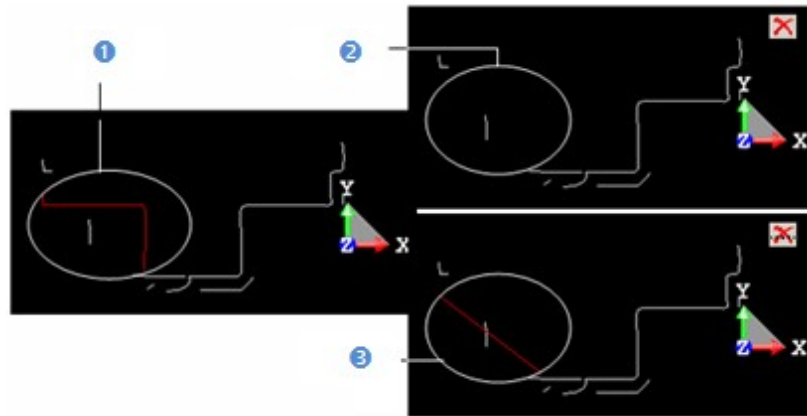
1. Perform a selection as described previously.
2. Click on the **Change Deletion Mode** pull down arrow.
3. Select **Delete Selection**  from the drop down list.
 - Segments and arcs inside the selection are deleted.
 - If the input is a slice cloud, **Reload Initial Cloud - Polyline** remains dimmed.
 - If the input is a polyline (fitted or not), **Reload Initial Cloud - Polyline** becomes enabled. Click on it to reload the initial polyline.

Note: You can also right-click anywhere in the **3D View** window to display the pop-up menu and select **Delete Selection**.

Delete Selection Filling Holes

To Delete Items and Prevent from Hole Creation:

1. Perform a selection as described previously.
2. Click on the **Change Deletion Mode** pull down arrow.
3. Select **Delete Selection Filling Holes**  from the drop down list.
 - Segments and arcs inside the selection are deleted and the extremities are connected together.
 - If the input is a slice cloud, **Reload Initial Cloud - Polyline** remains dimmed.
 - If the input is a polyline (fitted or not), **Reload Initial Cloud - Polyline** becomes enabled. Click on it to reload the initial polyline.



1 - Selected items (in red)

2 - Selected items are deleted

3 - Selected items are deleted and extremities are connected

Tip: You can also right-click anywhere in the **3D View** to display the pop-up menu and select **Delete Selection Filling Holes**.

Applying Filters



The second category contains filters for which parameters and options should be set. These filters are **Simplification** and **Fill Line Breaks**. The purpose of the **Simplification** filter is to simplify the selected/modeled polyline(s) by segments. The **Tolerance** parameter will be used in this filter so that the original points or the polyline vertices will be inside this tolerance with respect to the final approximated polyline(s). You can choose the **Use Arcs** option; the filter will use both segments and arcs to approximate the original polyline(s). The purpose of the **Fill Lines Breaks** filter is to fill gaps on the selected/modeled polyline(s) with segments. The **Gap** parameter will be used in this filter so that gaps whose size is smaller than this parameter will be filled by segments. Note that the default unit of measurement is set to millimeters; you do not need to enter “mm” after the value. You can change the default unit of measurement in **Preferences**.

Notes:

- No selection is required for both the **Simplification** filter and the **Fill Line Breaks** filter.
- If a selection has been done, both the **Simplification** filter and the **Fill Line Breaks** filter are applied to the selection.



Simplification

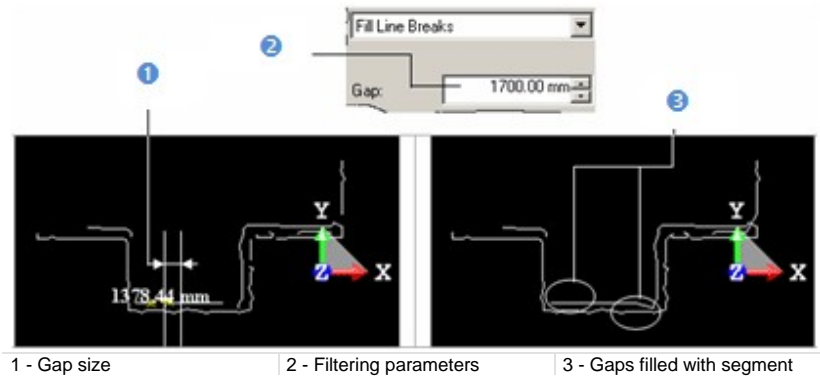
To Simplify the Modeled Polyline:

1. Drop down the selection list and select **Simplification**.
2. Enter a value in the **Tolerance** field and press **Enter**.
3. Or select a value using the **Up**  (or **Down** ) button.
4. Select the **Use Arcs** option if needed.
5. Click on the **Simplify** button.
 - Segments and arcs displayed in the **3D View** are simplified.
 - If the input is a slice cloud, **Reload Initial Cloud - Polyline** remains dimmed.
 - If the input is a polyline (fitted or not), **Reload Initial Cloud - Polyline** becomes enabled. Click on it to reload the initial polyline.

Fill Line Breaks

To Fill Line Breaks:

1. Drop down the selection list and select **Fill Line Breaks**.
2. Enter a value in the **Gap** field and press **Enter**.
3. Or select a value using the **Up**  (or **Down** ) button.
4. Click on the **Fill** button.
 - Line breaks are filled with segment lines.
 - If the input is a slice cloud, **Reload Initial Cloud - Polyline** remains dimmed.
 - If the input is a polyline (fitted or not), **Reload Initial Cloud - Polyline** becomes enabled. Click on it to reload the initial polyline.



Saving Results

After checking the modeled results, you can use the **Apply** button to create them in the **RealWorks**. Each modeled polyline will be created as a polyline. Note that if the original is also a polyline, then it will be replaced by the new one.

To Save the Results:

1. Click **Apply**.
2. Click **Close**.

Note: **Close** can also be selected from the pop-up menu.


Mesh Creation Tool

The purpose of this tool is to create a triangulated mesh from the selected point cloud(s) which, in this case, must have no geometry. Such a mesh can be used for further editing, texture mapping, and ortho-projection image creation or as input for the **Cutting Plane Tool**. You can also export it to other software in DXF (or DGN) format.

The **Mesh Creation Tool** uses a 2D triangulation method applying a projection of the 3D points onto a 2D surface. There are several ways to define this projection surface. Note that you can select several point clouds to use this tool. In this case, the tool will create a mesh for each selected point cloud.

Opening the Tool

To Open the Tool:

1. Select a point cloud from the **Project Tree**.
2. From the **OfficeSurvey** menu, select **Mesh Creation Tool** . The **Mesh Creation** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window and is composed of four parts. The first one enables to edit a point cloud previously selected and is composed of the **Sampling** and **Segmentation** tools. The selected point cloud is called **Working Cloud** and its total number of points is displayed in this dialog. The second part of this dialog allows you to select a projection mode. The third and last parts enable previewing and building meshes.

Frequently, the selected cloud contains many points, you may need to decimate them before doing the triangulation. You may also decide to generate a mesh on just a part of the selected cloud. To do this, you can use the **Segmentation** and the **Sampling** sub-tools.

Note: **Sampling** and **Segmentation** can be selected either from the pop-up menu or from the dialog.

Selecting a Projection Mode

A projection surface can be a 2D/3D plane or a 3D cylinder. There are three ways for determining a projection plane. The first way (called **Plane-based Projection**) consists of defining a 3D plane as in the **Cutting Plane Tool**. The second way (called **Screen-View-Based Projection**) consists of using the current camera position. In that case, the projection plane (in 2D) will be the screen plane. The third way (called **Station-Based Projection**) consists of using the scanning position linked to each station. In that case, the projection plane (in 2D) is the scanning grid surface, which, depending on the scanner, will be a plane or a spherical or cylindrical surface. And there is only one way to define a projection cylinder.

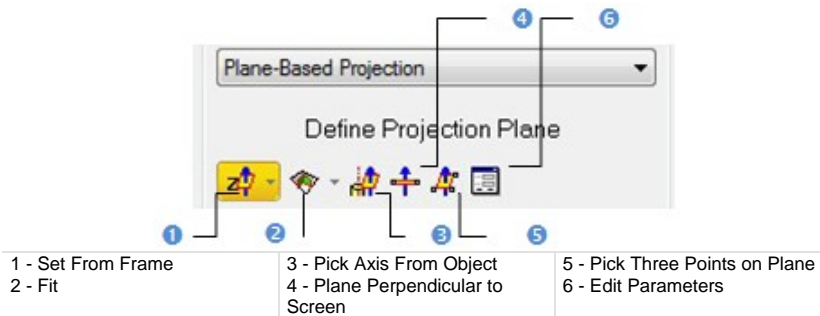
To Select a Projection Mode:

1. Drop down the **Select Projection Mode** list in the **Mesh Creation** dialog box.
2. Select a projection mode from the drop-down list.
3. Do any of the following:
 - Define a projection plane,
 - Define a projection cylinder,
 - Define a 2D projection based on the current view,
 - Define a 2D projection based on the scanning direction,
 - Define no projection plane.

Note: There is another method for computing meshes from the selected point cloud. This method (called **No Projection**) is based to any projection surface. Satisfactory results may be obtained where the selected point cloud is relatively free of spikes and peaks on its surface.

Plane-Based Projection

After selecting **Plane-Based Projection**, a projection plane perpendicular to the **Y Axis*** of the active coordinate frame appears in the **3D View**. You can change its direction as you're used to do in the **Cutting Plane Tool** or by manual-editing. In all cases, the bounding box that highlights the selection (point cloud) delimits the height of the projection plane and you cannot exceed it.



To Define a Projection Plane:

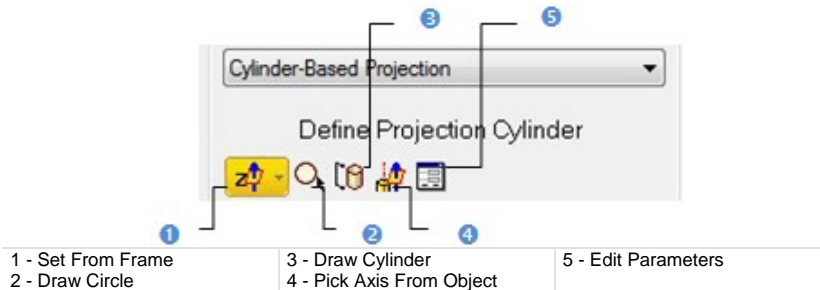
1. Select a frame's axis (1).
2. Or fit an extracted set of points with a plane (1).
3. Or find a perpendicular view plane from an extracted set of points (1).
4. Or pick an object's axis (1).
5. Or pick a plane perpendicular to the screen (1).
6. Or pick three points (1)(2).
7. Or edit the project plane's parameters.
 - a) Click **Edit Parameters**. The **3D Plane Editing** dialog opens.
 - b) Click on the pull down arrow and choose between **Normal + Point** and **Point + Point**.
 - c) If **Normal + Point** has been chosen, enter a direction in the **Normal** field and give a position in the **Point** field.
 - d) If **Point + Point** has been chosen, enter a position in the **Point (From)** and **Point (To)** fields.
 - e) Click **OK**. The **3D Plane Editing** dialog closes.

Notes:

- For more information related to (1), see **Step 2** of the **Cutting Plane Tool**. When selecting (2), the **Picking Parameters** toolbar appears, it's up to you to do a free picking or a constrained picking.
- (*) In the **X, Y, Z Coordinate System**.

Cylinder-Based Projection

After selecting **Cylinder-Based Projection**, a projection cylinder with an axis parallel to the **Y Axis*** of the active coordinate frame appears in the **3D View**. You can change the projection cylinder's direction according to the two other axes (**X*** and **Z***). If the selection (point cloud) that you performed in **Step 1** contains an entity, you can pick on it so that its axis becomes the new axis of the projection cylinder. You can also pick points on your selection to define a projection cylinder or edit one manually. In all cases, the bounding box that highlights the selection (point cloud) delimits the height of the projection cylinder and you cannot exceed it.









Note: (*) In the **X, Y, Z Coordinate System**.

Selecting a Frame Axis


This method consists of selecting an axis from the active frame as **Normal** direction. The initial plane will be moved so that its **Normal** will be parallel to the selected axis. Its position in the 3D scene will be kept and its **Offset** (altitude) will be reset.

To Select a Frame Axis:

1. Click on the **Set from Frame** pull down arrow.
2. Choose among **X Axis** , **Y Axis**  and **Z Axis**  (in the **X, Y and Z Coordinate System**).
3. Or choose among **North Axis** , **East Axis**  and **Elevation Axis**  (in the **North, East and Elevation Coordinate System**).


Picking an Axis From Object

To Pick an Axis From an Object:

1. Click **Pick Axis from Object** . The initial projection disappears from the **3D View**.
2. Click on the **Models** tab.
3. Right-click on the selection to display the pop-up menu.
4. Select **Display Geometry**.
5. Click one point.

Drawing a Circle

To Draw a Circle:

1. Click **Draw Circle** . The **Picking Parameters** toolbar appears.
2. Pick two free points or two constrained points.

Note: Picking can be done anywhere - on the selection (point cloud or mesh) or not. These two points determine the projection cylinder's diameter and its direction is perpendicular to the screen view.

Drawing a Cylinder


To Draw a Cylinder:

1. Click **Draw Cylinder** . The **Picking Parameters** toolbar appears.
2. Pick three free points or three constrained points.

Note: Picking should be on the selection (point cloud or mesh) for the two first points and anywhere the third point (on selection or not). The first and second picked points give the projection cylinder's direction and the second and third picked points determine its diameter.

Editing Parameters

To Edit Parameters:

1. Click **Edit Parameters** . The **Cylinder Editing** dialog opens.
2. Choose between **2 Points + Radius** and **Point + Direction + Radius**.
3. If **2 Points + Radius** has been selected:
 - a) Enter a point's position in the **Point1** field.
 - b) Enter another point's position in the **Point2** field.
 - c) Enter a value in the **Radius** fields.
4. If **Point + Direction + Radius** has been selected:
 - a) Enter a point position the **Point** field.
 - b) Define a direction in the **Direction** field.
 - c) Enter a value in **Radius** field.
5. Click **OK**.

Screen View-Based Projection

This method uses the current viewing direction to define a 2D projection.

Station-Based projection

This method uses the scanning direction to define a 2D projection.

No Projection

By choosing this method, the user can compute a mesh on more complex geometries.

Previewing Meshes

Before previewing a mesh, the **Number of Vertices** and **Number of Triangles** in the dialog are both equal to zero.

To Preview a Mesh:

1. Check the **Display Edges** option. This will display the edges of triangles for easier verification of the result.
2. If needed, uncheck the **Display Points** option. The input representation is removed from the **3D View**.
3. If needed, check the **Remove Discontinuities** option. This will remove the triangles around surface discontinuities (spikes, peaks etc.).
4. Click on the **Preview Meshes** button. The triangulation procedure will be performed.
 - On completion, the triangular mesh will be displayed in the **3D View**.
 - The **Number of Vertices** and **Number of Triangles** in the final mesh are shown in the dialog.
 - You can cancel the mesh and compute a new one. The **Number of Vertices** and **Number of Triangles** will then be updated automatically.

Notes:

- The **Remove Discontinuities** option is not available in the **No Projection** method.
- The **Display Edges**, **Display Points** and **Remove Discontinuities** options can be checked either before or after previewing the meshes.

Tip: **Preview Meshes** can also be selected from the pop-up menu.

Creating Meshes

By clicking on the **Create** button, the previewed mesh will be created in the **RealWorks** database. If you select several point clouds, the corresponding meshes will be created in association with each other. You cannot create a mesh without performing a preview. **RealWorks** will forbid you to do so by inhibiting the **Create** button.

To Create a Mesh:

- Click **Create**. The **Mesh Creation** dialog closes on its own.
 - If the input is a **Cloud**, a **Fitted Mesh** named **ObjectX** is created based on the **Cloud** in the **Models** tree.
 - If the input is the **Project Cloud**, the result, a separate object containing the created mesh and points of the **Project Cloud**, is named **ObjectX**.


Tip: **Create** and **Close** can also be selected from the pop-up menu.

Mesh Editing Tool

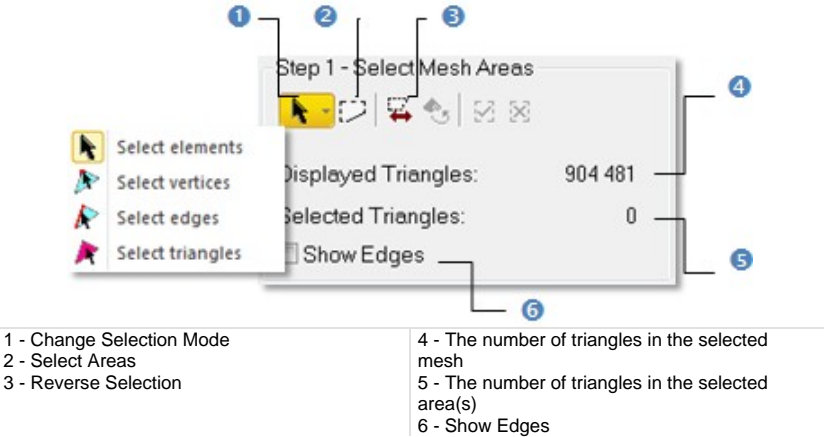
We have explained in the previous section how to compute a mesh from a point cloud using the **Mesh Creation Tool**. Here, we are going to describe a set of tools which can be used in complement of the previous one in order to improve the quality of the mesh computation. The **Mesh Editing Tool** allows you to edit a mesh: you can delete its vertices, edges or triangles, smooth or refine it, remove noisy peaks, extract a part from it in order to create a new mesh or to texture map or invert normal.

Opening the Tool

To Open the Tool:

1. Select a mesh from the **Models Tree**.
2. In the **OfficeSurvey** menu, select **Mesh Editing Tool** . The **Mesh Editing** dialog opens.


This dialog displays two numbers: "Displayed Triangles" and "Selected Triangles". "Displayed Triangles" is the number of triangles inside the Mesh displayed in the **3D View**. "Selected Triangles" is the number of triangles once a selection has been performed on the displayed mesh (in red in the **3D View**). Before any selection, only the **Change Selection Mode**, **Select Areas** and **Reverse Selection** icons are available, the number of "Displayed Triangles" matches the number of triangle inside the input (required to open the **Mesh Editing Tool**) and the number of "Selected Triangles" is equal to zero.



Tip: You can also click on the **Mesh Editing Tool** icon in the **Tools** toolbar.



Note: Clicking on the **Reverse Selection** icon without performing any selection will select the whole mesh displayed in the **3D View**.

Selecting an Element



You can select each element that composes a mesh. Please, use the **Select Elements**  icon to pick an element, no matter the element could be. Or use the **Select Vertices**, **Select Edges** and **Select Triangles** icons to respectively pick a vertex, an edge and a triangle.

Picking an Element

To Pick an Element:

1. Check the **Show Edges** option (if required).
2. Drop down the **Change Selection Mode** list.
3. Choose **Select Elements** .
4. Pick an element from the selected mesh. The picked element becomes yellow (or red).
5. Click **Reverse Selection** . All non-selected elements are selected and appear in red.



Notes:

- The **Keep Selected**  and **Keep Unselected**  icons remain dimmed (for either a vertex or an edge) and become enabled (for a triangle).
- To add a new element to the previous selection, first press **Ctrl** and then pick. Otherwise, the previous selection will be cancelled.
- The number of "Selected Triangles" remains unchanged (for either a vertex or an edge) and is updated (for a triangle).


Tip: The **Select Element** icon can also be selected from the pop-up menu.

Picking a Vertex

To Pick a Vertex:



1. Check the **Show Edges** option (if required).
2. Drop down the **Change Selection Mode** list.
3. Choose **Select Vertices** . A vertex symbol appears next to the cursor.
4. Pick a vertex from the selected mesh. The picked vertex becomes yellow.
5. Click **Reverse Selection** . All non-selected items are selected and appear in red.

Notes:



- The **Keep Selected**  and **Keep Unselected**  icons remain dimmed.
- To add a new vertex to the previous selection, first press **Ctrl** and then pick. Otherwise, the previous selection will be cancelled.
- The number of "Selected Triangles" remains unchanged.

Picking an Edge

To Pick an Edge:



1. Check the **Show Edges** option (if required).
2. Drop down the **Change Selection Mode** list.
3. Choose **Select Edges** . An edge symbol appears next to cursor.
4. Pick an edge from the selected mesh. The picked edge becomes yellow.
5. Click **Reverse Selection** . All non-selected items are selected.

Notes:



- The **Keep Selected**  and **Keep Unselected**  icons remain dimmed.
- To add a new edge to the previous selection, first press **Ctrl** and then pick. Otherwise, the previous selection will be cancelled.
- The number of "Selected Triangles" remains unchanged.

Picking a Triangle

To Pick a Triangle:

1. Check the **Show Edges** option (if required).
2. Drop down the **Change Selection Mode** list.
3. Choose **Select Triangles** . A triangle symbol appears next to the cursor.
4. Pick a triangle from the selected mesh. The picked triangle becomes red.
5. Click **Reverse Selection** . All non-selected items are selected and appear in red.







Notes:

- The **Keep Selected**  and **Keep Unselected**  icons become enabled.
- To add a new triangle to the previous selection, first press **Ctrl** and then pick. Otherwise, the previous selection will be cancelled.
- The number of "Selected Triangles" is updated according to the number of triangles that has been selected.

Fencing an Area

You can select by fencing an area on the selected mesh as with the **Segmentation** of a **Point Cloud**.

To Fence an Area:

1. Click on the **Select Areas**  icon.
2. Pick several points to draw a polygonal fence.
3. Double-click to close the polygonal fence. The **Keep Selected**  and **Keep Unselected**  icons become enabled. The "Selected Triangles" number is updated to match the number of triangles inside the polygonal fence.
4. Click **Keep Selected** . Triangles inside the fence are kept. The "Displayed Triangles" number is updated to match the "Selected Triangles" number which becomes then zero.
5. Or Click **Keep Unselected** . Triangles outside the fence are kept. The "Displayed Triangles" number is updated to match the opposite "Selected Triangles" number which becomes then zero.
6. Click **Reversion Selection**  to reverse the selection.

Tips:

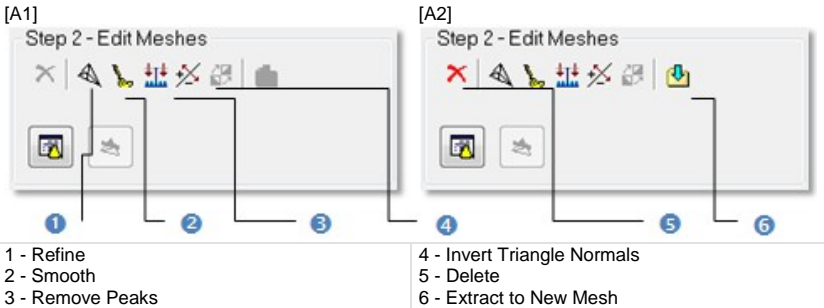
- You can press **Space Bar** to close the fence instead of double-clicking.
- You can click left + **Ctrl** to multi-select or click left + **Shift** to remove from the selection.

Note: Picking can be done out of the displayed mesh. The **Keep Selected** (or **Keep Unselected**) command is similar to the **In** (or **Out**) operation in the **Segmentation Tool**.

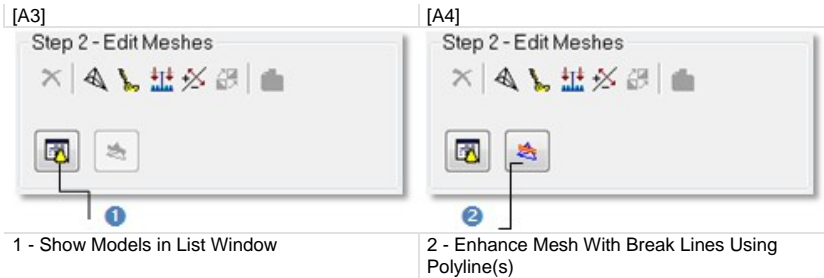
The **Reload All** command becomes active after choosing **Keep Selected** or **Keep Unselected**. You can then reload all triangles of the selected mesh.

Editing a Mesh

There are several tools for editing the selection previously displayed. You can delete, refine, smooth, remove peaks, extract to new mesh, invert normal on triangles or flip selected edges. Note that the **Refine**, **Smooth**, **Remove Peaks** and **Invert Triangle Normal** features do not require a selection on the mesh (see [A1]). Each of them (when selected) will be applied to the whole mesh. After selecting a triangle or a set of triangles, vertices and edges, **Delete** and **Extract to New Mesh** become active in addition to the four tools named above (see [A2]). Each of them (when selected) will be applied to the selection done in **Step 1** (vertices, edges, triangles or an area of the mesh).



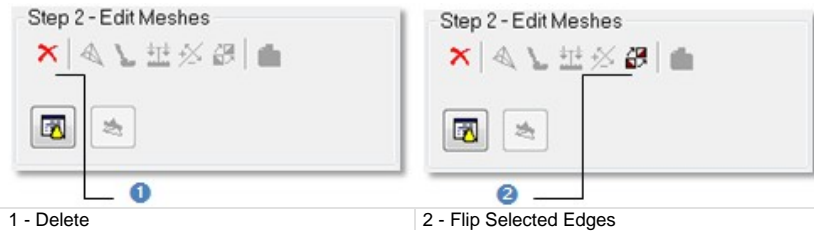
The **Show Models in List Window** lists polylines in the project (see [A3]). After selecting one, the **Enhance Mesh With Break Lines Using Polyline(s)** icon becomes enabled (see [A4]).



After selecting a vertex (or an edge), **Step 2** of the **Mesh Editing** dialog becomes as below in [B1] (or [B2]).

[B1]

[B2]



Smoothing a Mesh

The **Smooth** feature enables to apply a median filtering to the vertices of the selected triangles.

To Smooth a Mesh:

1. In the **Mesh Editing** dialog, click **Smooth**.
2. Or select the command from the pop-up menu.

Refining a Mesh

The **Refine** feature consists of swapping or splitting edges.

To Refine a Mesh:

1. In the **Mesh Editing** dialog, click **Refine**.
2. Or select the command from the pop-up menu.

Removing Peaks from a Mesh

The **Remove Peaks** feature enables to remove certain noisy peaks in the displayed mesh.

To Remove Peaks From a Mesh:

1. In the **Mesh Editing** dialog, click **Remove Peaks**.
2. Or select the command from the pop-up menu.

Reversing Triangles

To Reverse Triangles:

1. In the **Mesh Editing** dialog, click **Invert Triangle Normal**.
2. Or select the command from the pop-up menu.

Note: The **Invert Triangle Normal** will be applied to the whole mesh in display whatever the selection you made.

Deleting an Element from a Mesh

To Delete an Item from a Mesh:

1. In the **Mesh Editing** dialog, click **Delete**.
2. Or select the command from the pop-up menu.
 - Deleting a vertex will delete all triangles of the displayed mesh having that vertex in common.
 - Deleting an edge will delete all triangles of the displayed mesh having that edge in common.
 - Deleting a triangle will only delete that triangle.
 - In all cases, the "Displayed Triangles" number in the **Mesh Editing** dialog is then updated.

Tip: Instead of selecting **Delete**, you can use the related short-cut key **Del**.

Extracting to a New Mesh

The **Extract to New Mesh** feature enables to create a new mesh from the selection done in **Step 1**. By performing this operation, you can segment a mesh into different sub-meshes. This feature can be applied to a single triangle (or a set of triangles).

To Extract to a New Mesh:

1. In the **Mesh Editing** dialog, click **Extract To New Mesh**.
2. Or select the command from the pop-up menu.

When you extract the selection to a new mesh, the "Display Triangles" number in the **Mesh Editing** dialog is updated.

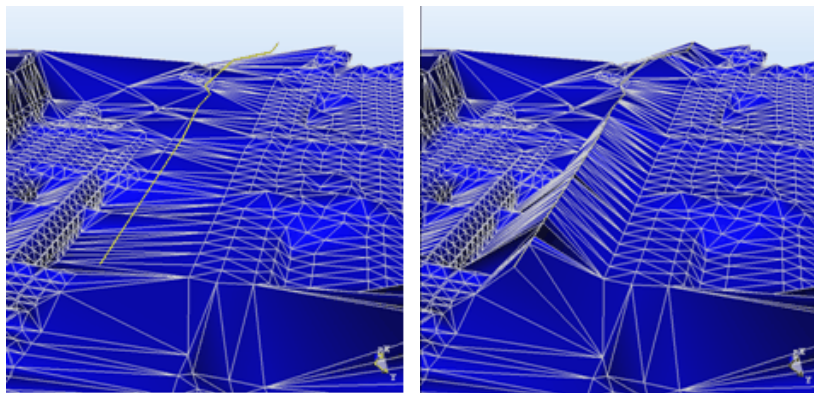
Tip: Instead of selecting the **Extract to New Mesh** icon, you can use the related short-cut key **P**.

Enhancing With Break Lines Using Polyline(s)

The **Enhance Mesh With Break Lines Using Polyline(s)** feature does not require a selection; it will be applied to the mesh displayed in the **3D View**. This allows you to integrate a polyline into a mesh.

To Enhance a Mesh with Break Lines using Polyline(s):

1. Click the **Show Models in List View** icon. Polyline(s) are listed in the **List** window and none is displayed in the **3D View**.
2. Select the appropriate polyline from the **List** window. The **Enhance Mesh With Break Lines Using Polyline(s)** button becomes enabled.
3. Toggle the selected Polyline's **On/Off** icon **On** (if required). It is displayed in the **3D View**.
4. Click the **Enhance Mesh With Break Lines Using Polyline(s)** icon.



The new mesh contains new edges that correspond to the **Polyline**. All the vertices of the previous mesh are preserved during this operation.

Note: Several polylines may be selected at the same time.

Tip: The **Enhance Mesh With Break Lines Using Polyline(s)** icon can also be selected from the pop-up menu.

Flipping Edges

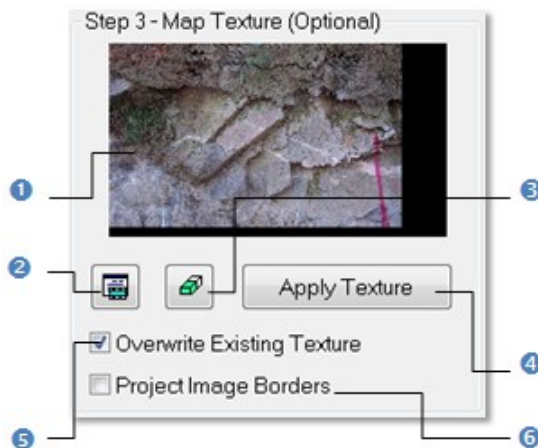
When an edge is shared by two triangles, you can use the **Flip Selected Edges** tool for swapping it so that it is still shared by these two triangles but from the two other vertices.

To Flip an Edge:

1. Select an edge (or a set of edges) from the displayed mesh.
2. Click on the **Flip Selected Edges** icon.
3. Or select the command from the pop-up menu.

Mapping With a Texture

This step (in option) consists of using a matched image to texture map the selection done on the displayed mesh. If any selection has been performed, the texture mapping will be applied to the entire mesh. If the selection (or the entire mesh) has already been textured, you can choose to overwrite or remove to the existing texture. Texture mapping can be done by recomputing (splitting) the edges to fit the image boundaries.



1 - The selected image shown as a thumbnail
 2 - Show Images in List View
 3 - Remove Existing Texture

4 - Apply Texture
 5 - Overwrite Existing Texture
 6 - Project Image Borders

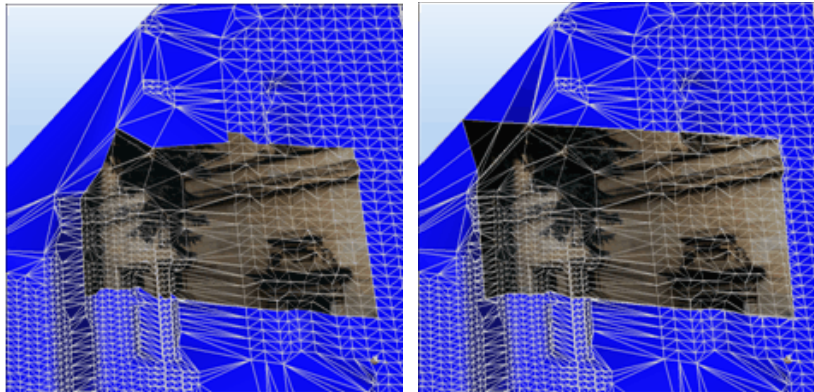
Notes:

- Unmatched images cannot be used for texture mapping. This is why selecting one will not show it as a thumbnail in **Step 3** and the **Apply Texture** button is dimmed.
- Several matched images may be selected at the same time and applied as textures.

Applying a New Texture

To Apply a Texture:

1. Select an area (or the entirety) of the mesh for applying the texture.
2. Click the **Show Images In List View** icon. Images are listed in the **List** window and none is displayed.
3. Select a matched image from the **List** window. It is shown as a thumbnail in **Step 3**.
4. Toggle the selected image's **On/Off** icon **On**. It is displayed in an independent window in the **3D View**. The displayed mesh is aligned with the camera's point of view.
5. Click **Apply Texture**.
 - If the **Project Image Borders** option is not checked, only the selected triangles that lie entirely inside the image will be textured.
 - If the **Project Image Borders** option is checked, the selected triangles that lie entirely inside the image will be textured in the same way, but the triangles that intersect the selected image boundaries will also be split. In this way, the whole image is used for texturing the selected triangles. Note that the shape of the mesh does not change during this operation.



Removing an Existing Texture

To Remove an Existing Texture:

1. Select an area (or the entirety) of the mesh where a texture removal is required.
2. Click on the **Remove Existing Texture** icon.

Overwriting an Existing Texture

To Overwrite an Existing Texture:

1. Select an area (or the entirety) of the mesh where a texture overwriting is required.
2. Click the **Show Images in List View** icon. Matched images are listed in the **List** window and none is displayed.
3. Select the appropriate image from the **List** window. It is shown as a thumbnail in **Step 3**.
4. Toggle the selected image's **On/Off** icon **On**. It is displayed in an independent window in the **3D View**. The displayed mesh is aligned with the camera's point of view.
5. Check the **Overwrite Existing Texture** option.
6. Click **Apply Texture**.

Applying Operations

Till now, all operations applied to the selected mesh(es) are just temporarily stored. To make them permanent, you have to apply the operations. You can then quit the tool.

To Apply the Operation:

1. Click **Apply**.
2. Click **Close**.

Tip: **Close** can also be selected from the pop-up menu.

Note: Leaving the **Mesh Editing Tool** without applying all changes in the database will make appear a message.

Ortho-Projection Tool


Conventional and perspective photographs taken by any 2D camera show distortions caused by the camera angle and the topography itself. These phenomena can be noticed particularly on aerial photographs. Non-uniform scale on this kind of photographs prevents from direct measurement, like on a map. These disadvantages can be cancelled by ortho-rectification. This means that such photographs are computer-deformed.

The **Ortho-Projection Tool** allows you to create ortho-images from the selected point cloud or textured mesh. You can either export the ortho-images to CAD software, such as AutoCAD® or MicroStation® for further processing or drafting operations, or you can perform 2D measurements directly within **RealWorks**. The basic principle behind this tool is to choose a **Projection Plane** on which the ortho-image will be created, and choose the right information that you want to store in this image, and then create it. All the metric information will be stored in this image, i.e. measurements made on this photo will be accurate.

Opening the Tool

If a point cloud has been selected, you can work it to delimit an area for the Ortho-Projection calculations, to render it cleaner without parasite points or to simplify it. Use the two following sub-tools (**Sampling** and **Segmentation**) for doing these operations.

To Open the Tool:

1. Select a point cloud (or mesh) from the **Project Tree**.
2. From the **OfficeSurvey** menu, select **Ortho-Projection Tool** . The **Ortho-Projection Tool** dialog opens.

This dialog opens as the third tab of the **Workspace** window and is composed of four parts. Each part corresponds to one step in the ortho-projection computation. The first step (called **Define Projection Plane**) is to define, orientate and check a **Projection Plane**. The second part (called **Define Zone of Interest**) is to draw a **Zone of Interest** on the previous **Projection Plane**. The third part (called **Define Image Resolution**) is to specify parameters and rendering to apply. The fourth part is to preview and create ortho-images.

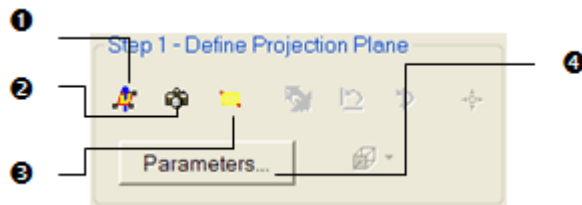
Tip: You can also click on the **Ortho-Projection Tool** icon in the **Tools** toolbar.

Note: The two sub-tools (**Segmentation** and **Sampling**) are not available if the input is a mesh.

Defining a Projection Plane

You can define a **Projection Plane** by three points on an object or by using the current viewing/camera position. The **Projection Plane** that results from these two methods are not similar in terms of size. In the first method, the three picked points delineate the size of the **Projection Plane**. In the second method, the bounding box that highlights the input (point cloud or **mesh**) delineates the size of the **Projection Plane**.

A **Projection Plane** is characterized by a projection direction called **View**, two orientations **Right** and **Up** (also called **X*** and **Y*** directions) which define an orthogonal frame, a position which is its position in the 3D scene and two dimensions that correspond to its length and width. The **View** direction is the **Normal** of that **Projection Plane**. You can also define a **Projection Plane** by getting the parameters of an existing ortho-image.



1 - Define Projection Plane by Picking 3 Points
2 - Define Projection Plane by Screen View


3 - Define Corners of Zone of Interest
4 - Define Projection Plane from Existing Ortho-Image

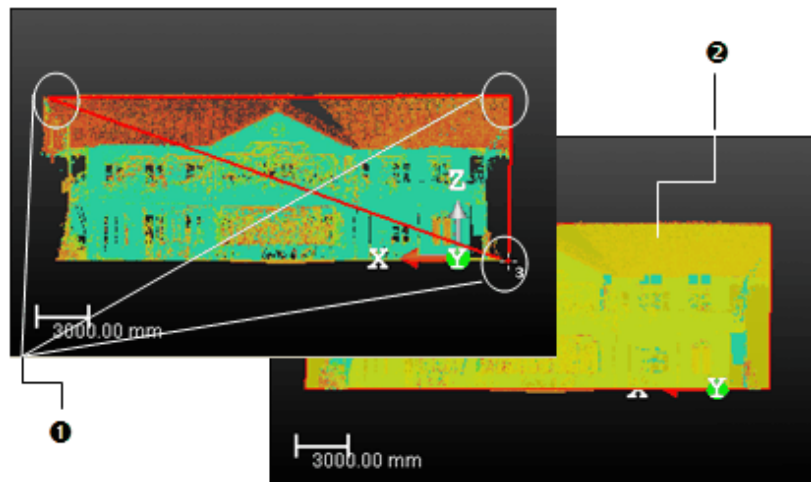
Note: (*) In the **X, Y, Z Coordinate Frame**.

Picking Three Points

You can pick three points to define a **Projection Plane**. The accuracy of such **Projection Plane**'s orientation will be influenced by the points you pick. It is recommended to pick these points in such way that they are distributed across the area on which you want to calculate the ortho-projection.

To Pick Three Points:

1. Click on the **Define Projection Plane by Picking 3 Points**  icon. The **Picking Parameters** toolbar appears.
2. Pick three points (free or constrained) on displayed objects.



1 - Three picked points

2 - The defined projection plane (in yellow)

A number located beside the mouse's pointer guides you in picking points. This number starts from **One** and ends by **Three**. When two points are picked, they are linked by a red segment. When three points are reached, they are linked two-by-two by a red segment and form in that way a triangle. A **Projection Plane** is then computed. No need of defining its **Normal**; it will be automatically calculated. You can cancel this plane whenever you want and start a new one. To do it, start again the previous procedure.

Notes:

- You can also select **Cancel Picking** (or **Close**) from the pop-menu.
- You can **Pan**, **Rotate**, **Zoom** (in or out) and **Zoom** with constant ratio in the **3D View** while defining a **Projection Plane**.

- In the picking mode, pressing **Esc** cancels the selection of points in progress. Out of the picking mode, pressing **Esc** closes the **Ortho-Projection Tool**.

Using the Current Camera View

In certain applications, you may need to visually choose a projection plane. The **Define Projection Plane by Screen View** tool allows this. You first need to rotate the 3D scene to find the right viewing direction, and use the current screen plane as the **Projection Plane**.


To Use the Current Camera View:

1. Turn the scene to find the right viewing direction.
2. Click the **Define Projection Plane By Screen View**  icon.

Setting Corners

Another method for defining a **Projection Plane** consists of setting its corner values. This operation has no influence upon the **View** direction and the **Right** and **Up** orientations (**X** and **Y** directions).

To Set Corners:

1. Click the **Define Corners of Zone of Interest**  icon. The **Vertical Rectangle Corner Coordinates** dialog opens.
2. Enter a point position in the **Top Left Corner** field.
3. Enter a point position in the **Bottom Right Corner** field.
4. Click **OK**.

Tip: You can also select the **Set Projection Plane Corners** icon from the pop-up menu.

Using Existing Parameters

Sometimes, you may need to use the parameters of an existing ortho-image to calculate your own ortho-image. You can use the method below to recover the **Projection Plane** from that of an existing ortho-image by getting its directions (**View**, **X*** and **Y***), its position in the 3D scene and its Resolution and Size. An option allows you to preview that ortho-image in **3D View**.

To Use Existing Parameters:

1. Click **Parameters**. The **Set Parameters of Projection Plane** dialog opens.

1 - An existing ortho-image in your project or the current Projection Plane

2 - Parameters of the existing ortho-image
3 - Resolution and Size are not yet defined

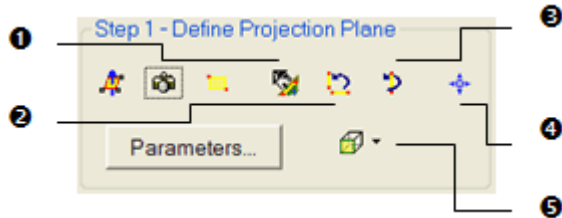
2. Click on the **Select Source Plane** pull-down arrow.
3. Select an existing ortho-image from the drop-down list
4. Modify the parameters (if necessary).
5. Check the **Preview in 3D Window** option (if required).
6. Click **OK**. The **Set Parameters of Projection Plane** dialog closes.

Notes:

- You can use the **C** short-cut (or select **Set Camera Parameters (C)** from the pop-up menu) to set or edit parameters.
- (*) In the **X, Y, Z Coordinate Frame**.

Modifying a Projection Plane

After defining a **Projection Plane**, the other icons in the **Step 1** panel become active.




- 1 - Define Horizontal by Picking Two Points
- 2 - Rotate Counterclockwise 90°
- 3 - Rotate 90° around Vertical Axis

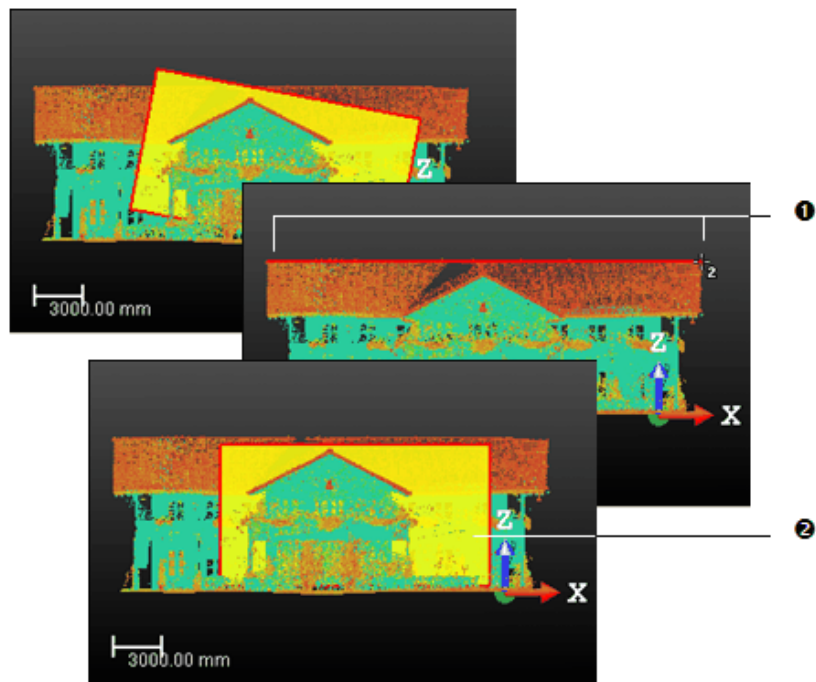
- 4 - Define Projection Plane Position by Picking Point
- 5 - Buttons for selecting a canonical view

Defining the Horizontal

The **Define Horizontal By Picking 2 Points** feature enables to align the horizontal orientation of a **Projection Plane** to the displayed data by picking two points. It means that for a given **Projection Plane**, you keep unchanged its **Normal** (**View** direction) and you adjust its **Right** and **Up** orientations (horizontal and vertical).

To Set the Horizontal of a Projection Plane:

1. Click **Define Horizontal by Picking 2 Points** . The **Picking Parameters** toolbar appears.
2. Pick a free or constrained point on the selected object to start the first point of the X direction.
3. Pick then the second point (free or constrained) to end this X direction.



1 - The first and second picked points

2 - The projection plane is aligned to the defined horizontal


After clicking **Define Horizontal By Picking 2 Points**, a **2D Grid** appears upon the 3D scene. You can hide it or change its size. The mouse cursor shape changes. The arrow becomes a pointer. A number beside this pointer guides you in selecting points. It starts from **One** that corresponds to the first point of the X direction and ends by **Two**. Once the two points reached, a red segment links them and the new **Projection Plane** will be then generated. No need of defining a **Normal** direction. It keeps the former one. You can cancel it whenever you want and start a new one. To do it, start again the previous procedure.

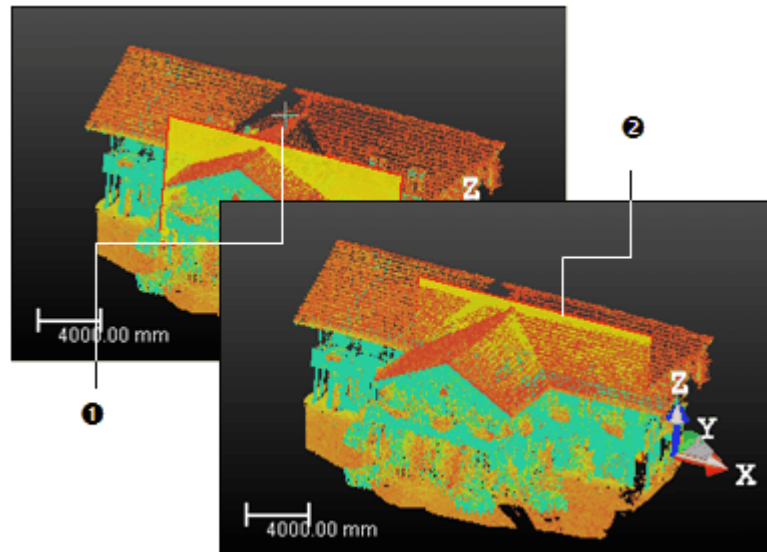
Note: In the picking mode, pressing **Esc** or selecting **Cancel Picking (Esc)** from the pop-up menu cancels the selection of points in progress and closes **Define Horizontal by Picking 2 Points**.

Setting a Position

You can also modify the position of a **Projection Plane** by picking a point on the displayed scene (point cloud or mesh). This is important if you want to use the elevation information for calculating the ortho-projection.

To Set a Position:

1. Click the **Define Projection Plane Position by Picking Point**  icon. The **Picking Parameters** (in 3D constraint mode) toolbar opens.
2. Pick a point (free or constrained) anywhere on the selected object.



1 - A point picked on the displayed cloud

2 - The projection plane moves from its current position to its new position

Notes:

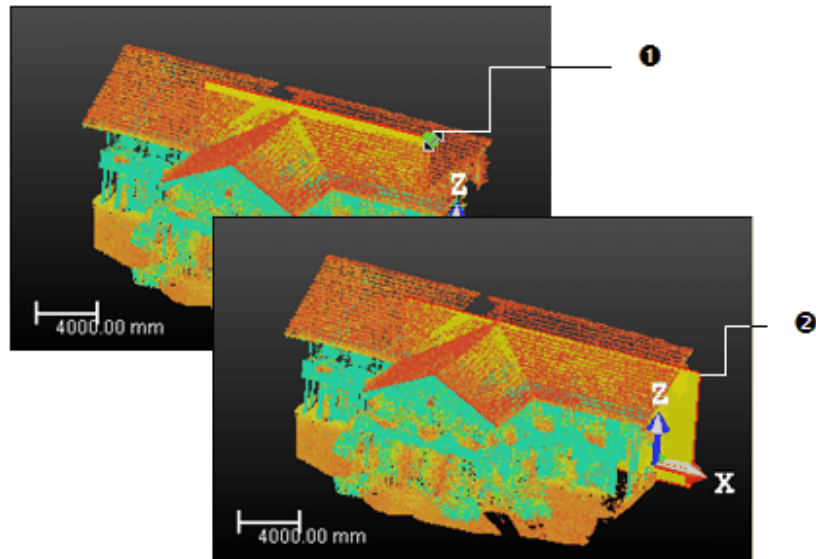
- Modifying a **Projection Plane**'s position will not modify its orientation.
- Press **Esc** (or select **Cancel Picking (Esc)** from the pop-up menu) to close the **Define Projection Plane Position by Picking Point** tool.

Changing Dimensions

You can resize the previous **Projection Plane**. This operation has not an influence upon the **View** direction and the **Right** and **Up** orientations (**X*** and **Y*** directions). The resized **Projection Plane** keeps the same parameters than before except the dimensions. You can do this either by dragging & dropping a corner for example in the **3D View** or by entering coordinates in a dialog.

To Change Dimensions:

1. Place the mouse cursor upon any handle of a **Projection Plane**. A green square appears.



1 - The handle before the drag & drop operation | 2 - The handle during the drag & drop operation

2. If a corner handle is selected, drag it to increase or reduce the **Projection Plane** size. During this operation, the green square becomes yellow.
3. If a middle handle is selected, drag it to increase or reduce the **Projection Plane** width (or length). During this operation, the green square becomes yellow.

Note: (*) In the **X, Y, Z Coordinate Frame**.

Checking a Projection Plane

In the example of a house, in order to have a complete facade drawing, you need to calculate ortho-images for both the front and side views. You can rotate the **Projection Plane** or the scene to calculate a series of ortho-images in different orientations. You can use a canonical view to control the projection definition, rotate left the scene (**Half Pi** rotation) or rotate the scene around the screen vertical axis. Six canonical views are available: **Front View**, **Back View**, **Left View**, **Right View**, **Top View** and **Bottom View**. You can edit and validate the current **Projection Plane**'s parameters. This enables an accurate definition and control of the **Projection Plane**'s vectors and depth.

Viewing from a Projection Plane' Side

The user can view the displayed scene from each of the **Projection Plane**' side.

To View from a Projection Plane' Side:

1. Click on the canonical view pull down arrow.
2. Choose among **Front View**, **Back View**, **Left View**, **Right View**, **Top View** and **Bottom View**.


Notes:

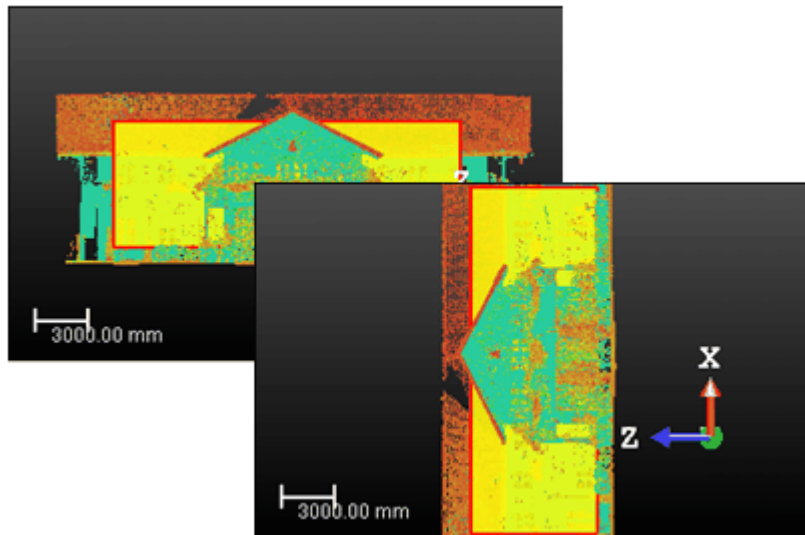
- You need to first define a **Projection Plane**. Otherwise, all views are dimmed.
- You can swap from a view to another not by clicking on the button as are used to do in the **View Alignment** toolbar but by clicking on the pull down arrow. Because clicking on that button will always bring you to the **Front View** of the **Projection Plane** which is the view required to compute an ortho-image.

Rotating Counterclockwise 90°

The **Rotate Counterclockwise 90°** feature rotates left both the displayed scene and the **Projection Plane**.

To Rotate Left Both the Scene and the Projection Plane:


1. Set the **Projection Plane** to see its front view (by choosing **Front View**).
2. Click the **Rotate Counterclockwise 90°**  icon. The displayed scene and the **Projection Plane** are rotated left.
3. Click again the **Rotate Counterclockwise 90°** icon. The displayed scene and the **Projection Plane** are rotated left again, and so on.

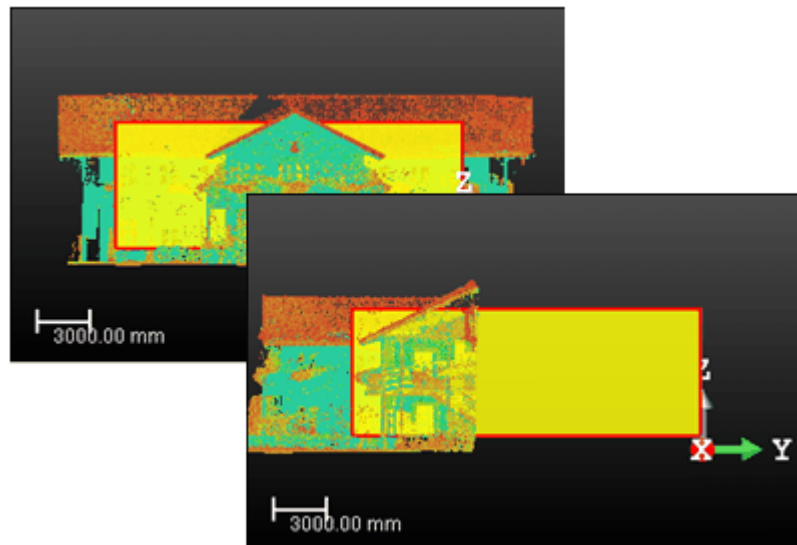


Both the Projection Plane and the 3D scene are rotated left

Rotating 90° Around Vertical Axis

To Rotate the Projection Plane 90° Around the Vertical Axis:

1. Bring the **Projection Plane** in **Front** view.
2. Define the region on which you want to calculate the ortho-image and save the result.
3. Click **Rotate 90° Around Vertical Axis**  to turn the active projection plane to the side view.
4. Modify the region definition and calculate the second ortho-image and save the result.



The projection plane remains fixed while the 3D scene rotates 90° around its vertical axis

Editing Parameters

To Edit Parameters:

1. Click **Parameters**. The **Set Parameters of Projection Plane** dialog opens.
2. Check the **Preview in 3D View Window** option (if required).
3. Enter new values in the **View Direction** field.
4. Enter new values in the **X Direction** field.
5. Enter new values in the **Y Direction** field.
6. Enter a new value in the **Point Position** field.
7. Click **OK**.

Tip: You can also use the short-cut key **C** to edit parameters (or select **Set Camera Parameters** from the pop-up menu).

Defining a Zone of Interest

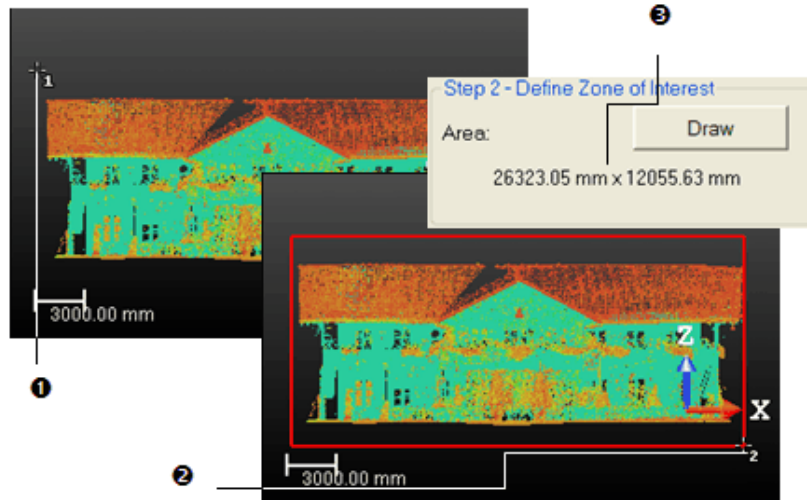
You need to define a region (called **Zone of Interest**) on the previous **Projection Plane** from which the ortho-projection will be computed. For this, you have to draw a rectangular frame. In that drawing mode, you are locked in the **Projection Plane**'s plane with a 2D-grid in superimposition and the **Projection Plane** is hidden.

Before clicking on the **Draw** button, the **Area** in the **Step 2** panel is "Undefined". After clicking on the **Draw** button, the entire **Projection Plane** is set as a **Zone of Interest** and its dimensions appears in the **Area** field. After drawing a rectangular frame, the **Area** values are updated to match the drawn frame' size.

Drawing

To Define a Zone of Interest:

1. Click **Draw**. The cursor will take the shapes as shown below.
2. Draw a rectangular frame by picking two points.



- | | |
|---|--|
| 1 - The cursor shape of the first point to pick | 2 - The cursor shape of the second point to pick |
| | 3 - The Zone of Interest size |

A **2D Grid** appears upon the locked scene. You can hide it or change its size. The mouse cursor shape changes. The arrow becomes a pointer with a number beside; this guides you in selecting points. This number starts from **One** (first corner of a rectangular frame) and ends by **Two** (opposite corner). You can cancel the rectangular frame whenever you want and start a new one.

Notes:

- Press **Esc** (or select **Cancel Rectangle (Esc)** from the pop-up menu) to undo the selection of points in progress (once the first has been picked).
- Movements are constrained to the **Projection Plane's** plane and are restricted to **Pan** and **Zoom**. **Rotate** is not allowed.

Resizing

To Resize a Zone of Interest:

1. Pick in the previous rectangular frame.
2. Move the cursor over a handle in green.
3. Drag-and-drop the handle to resize the rectangular frame.

Note: The **Area** values are updated to match the resized frame's dimensions.

Setting a Resolution

You can define an ortho-image's resolution by giving the PPI (Pixels Per Inch), specifying the number of pixels in the **X*** and **Y*** directions or giving the pixel size. The pixel size is equal to 1 (if the unit of measurements is Inch) or to 25.4 (if the unit of measurement is millimeters) divided by the image resolution (in PPI). The image size (in pixels) is obtained by dividing the interest zone size by the pixel size.

To Set a Resolution:

1. Click **Resolution**. The **Set Image Resolution of Ortho-Image** dialog opens.
2. Do one of the following:
 - Enter a value (in PPI) in the **Image Resolution (PPI)** field.
 - Enter two values (in pixels) in the **Image Size (WxH)** field.
 - Enter a value (in the current unit of measurement) in the **Pixel Size** field.
3. Click **OK**.

Tip: You can use the short-cut key **R** or select **Set Image Resolution (R)** from the pop-up menu to open the **Set Image Resolution of Ortho-Image** dialog.

Note: (*) In the **X, Y, Z Coordinate Frame**.

Rendering

You can choose an option to render the computed ortho-images(s). There are five options for a point cloud input: **Using Elevation** (the elevation value is calculated for each point based on its distance to the defined area. The calculated elevation value will be discretized in 8 bits to create the ortho-projection images), **Using Normal Shading** (the normal of each point will be used in shading calculation to create the ortho-projection images), **Using Grey Scaled Intensity** (the (grey) intensity value of each point will be used to create the ortho-projection images), **Using Color Scaled Intensity** (the (color) intensity value of each point will be used to create the ortho-projection images) and **Using True Color** (the color associated with each point will be used to create the ortho-projection images). There is only one option for a mesh input: **Textured Meshes**.

To Choose a Rendering Option:

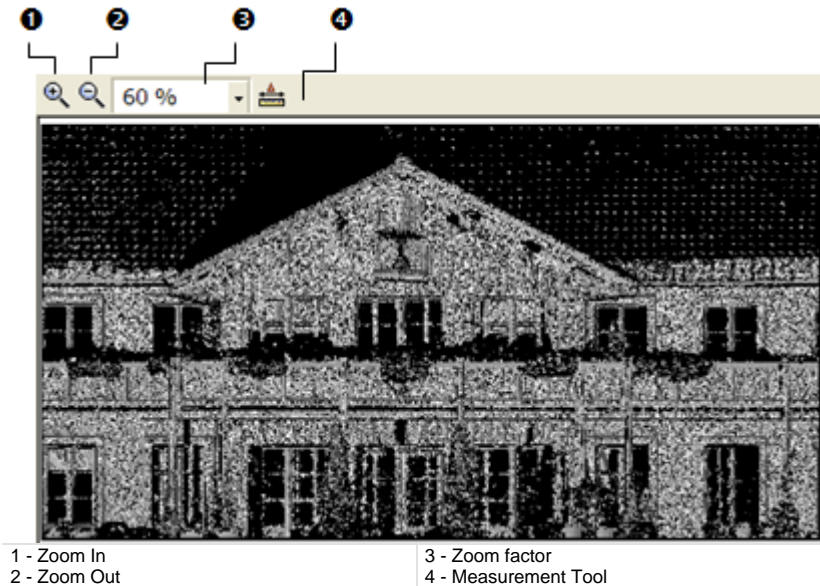
1. Click on pull down arrow below the **Density**.
2. Select a rendering option from the drop down list.

Previewing an Ortho-Image

You can now use the **Preview** button to calculate the ortho-image. When completed, the calculated image will be shown in the **2D Image** viewer. You can change the current parameters and perform a new preview. You can do this as many times as required until you have the right result.

To Preview an Ortho-Image:

- Click **Preview**. The computed image is shown in a **2D Image** viewer.



Before previewing the ortho-image, clicking **Close** leaves the **Ortho-Projection Tool** and the position of the ortho-image is lost. After previewing the ortho-image, clicking **Close** opens a dialog which prompts you to abort or continue the operation. In this **2D Image** viewer, you can zoom-in (or zoom-out). You can zoom in three ways. The first one is to magnify (or reduce) an area of the ortho-image using **Zoom In** and **Zoom Out**. The second way is to magnify (or reduce) the ortho-image using the mouse wheel (if existed). The third is to select a zoom factor from the drop-down list.

If the ortho-image is bigger than the **2D Image** viewer can show, you can pan it on left-click in any direction. You can also make a 2D-distance measurement. Because the metric information is stored in the calculated ortho-image, the measurement is accurate. You can create the measurement in the database by using the corresponding command in the pop-up menu. Note that once it is created, the measurement will be shown in the **3D View**.

Tips:

- You can select **Preview Ortho-Image** from the pop-up menu.
- The color of the background on an **Ortho-Image** is the same as the one in the **3D View**. You can change it in **Preferences\Viewer**. This change will be then applied whatever the **Rendering** option you choose.

Printing an Ortho-Image

You can print the preview of an ortho-image.

To Print an Ortho-Image:



1. Right-click in the 2D Image view.
2. Select **Print** from the pop-up menu. The **Print Setup** dialog opens.
3. Define a **Printer**.
4. Choose a **Paper' Size** and **Source**.
5. Choose an **Orientation** between **Portrait** and **Landscape**.
6. Add some comments in the **Legend** panel.
7. Set an **Appearance** between **As Is** and **Reverse Colors**.
8. Click **OK**.

Tip: You can also select **Print** from the menu bar or clicking its icon in the **Main** toolbar.

Splitting an Ortho-Image

When an ortho-image is too wide (or long), you can split it into a set of ortho-images of smaller size. You can split an ortho-image along a direction (**Width** (or **Height**)) or along two directions (**Width + Height**).

To Split an Ortho-Image:

1. Enter a number in the **W** (or **H**) field.
2. Or use the **Up**  (or **Down** ) button to select a number in the **W** (or **H**) field.

Note: Each split ortho-image is named as follows: ImageX_Line Index_Column index.

Creating an Ortho-Image

You can save the result in the database. An ortho-image whose name is "Image" is created and put under the current active group of the **Images Tree**. You can create several ortho-images without leaving the tool.

To Create an Ortho-Image:

1. Click **Create**.
2. Click **Close**.

Tip: You can also select **Create Ortho-Image** (or **Close**) from the pop-up menu. You should first hide the **2D Image** viewer.


Note: You do not need to first preview an ortho-image to create it in the database.

Multi-Ortho-Projection Tool

This tool allows you to create multi-ortho-projection images from the selected point cloud(s) or mesh(es). You can either export the ortho-projection images to CAD software, such as AutoCAD® or MicroStation® for further processing or drafting operations, or you can perform 2D measurements directly within **RealWorks**. The basic principle behind this tool is to use a polyline to create multi-projection planes on which the ortho-projection images will be created, and choose the right information that you want to use in the ortho-projection, and then create the images either one-by-one or all at once. All the metric information will be stored in this image, i.e. measurements made on this photo will be accurate.

Opening the Tool

To Open the Tool:

1. Select a point cloud (or mesh) from the **Project Tree**.
2. In the **OfficeSurvey** menu, select **Multi-Ortho-Projection Tool** . The **Multi-Ortho-Projection** dialog opens.

If a point cloud has been selected, you can work on it to delimit an area for the multi-ortho-projection computation, to render it cleaner without parasite points or to simplify it (see the **Sampling** and **Segmentation Tools**). If a mesh has been selected, the two upper sub-tools are unavailable. The results of applying these sub-tools will not save into the database; they are used only for calculation purposes.

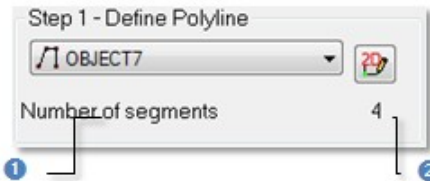
Tip: You can click on the **Multi-Ortho-Projection Tool** icon in the **Tools** toolbar.

Defining a Polyline

A polyline can be either a line of continuous segments which can be closed (or not). It can also be composed of one or several non-continuous segments but it cannot be formed by arcs.

Selecting

If there is a polyline within the project, it is displayed in the **Define Polyline** field. The "Number of Segments" is not then equal to zero. The selected point cloud (or mesh) and the **Projection Planes** obtained by extruding each segment of this selected polyline along the Z* direction are displayed in the **3D View**.



1 - Field for selecting an existing Polyline

2 - Number of segments in the selected (or drawn) Polyline

To Select a Polyline:

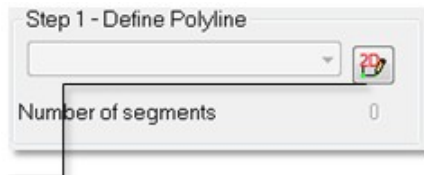
1. Click on the **Define Polyline** pull down arrow.
2. Select a polyline from the drop down list.

Notes:

- A polyline which is composed of arcs will not appear in the selection list.
- (*) In the **X, Y, Z Coordinate System**.


Drawing

If no polyline is available. The **Define Polyline** combo box is grayed out and the "Number of Segments" is equal to zero. You have to create at least one in the database. The selected point cloud (or mesh) is shown with a 2D Grid in superimpose (if not hidden previously) in the **3D View**. The scene is constrained in the **XY*** plane of the active coordinate frame, brought to the **Top** view and movements while picking points are restricted to **Rotate** (around the **Z*** axis), **Zoom** (along the **Z*** axis), and **Pan** (in the **XY*** plane).



Create New 2D Polyline

To Draw a Polyline:

1. Click the **Create New 2D Polyline**  icon. The **Drawing Tool** and **Picking Parameters** (in 2D constraint mode) toolbars appear in the **3D View**. The mouse cursor shape changes to a pencil.
2. Draw a polyline by picking several points.
3. Click **End Line**. The last picked point ends the line.
4. Click **Create**. The drawn Polyline appears in the **Define Polyline** field and its "Number of Segments" is updated.

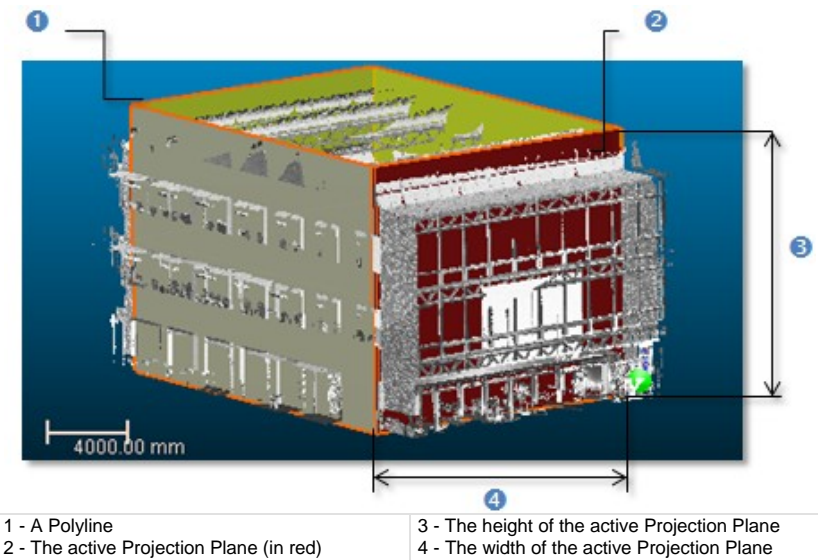
Tip: You can select each of the **Drawing Tool** features from the pop-up menu.

Notes:

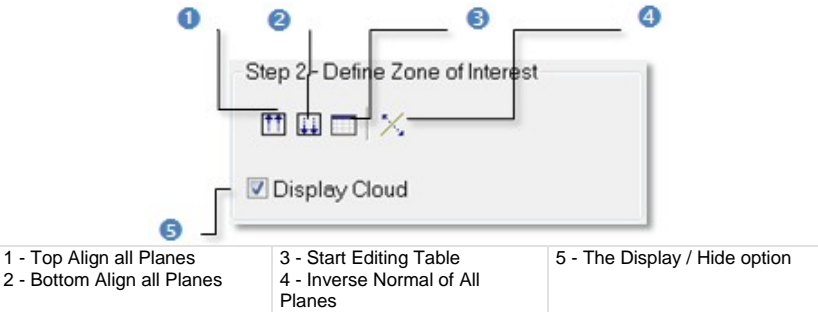
- (*) In the X, Y, Z Coordinate System.
- In the **Drawing Tool** toolbar, the **Change Mode to Arc**, **Draw Circle by Defining the Center and Radius** and **Draw Circle by Defining the Diameter** icons are enabled and let the user to draw such kind of polyline but any will be into account.

Defining a Zone of Interest

After defining a polyline, a series of **Projection Planes** runs along that polyline. For a given **Projection Plane**, its width corresponds to the length of a segment and its height to the height of the bounding box that highlights the input (point cloud or mesh) along the **Z** axis of the active coordinate frame.



You can modify all the **Projection Plane** heights at once by changing the **Maximal Altitude** (or **Minimal Altitude**) value or each **Projection Plane** height individually. The **Maximal** (or **Minimal**) **Altitude** default value will be the **Maximal** (or **Minimal**) **Altitude** value of the active **Projection Plane**. The maximal value must be higher than the minimal value.



To Define a Zone of Interest:

1. Top align all **Projection Planes**.
2. Or bottom align all **Projection Planes**.
3. Or Edit all **Projection Planes**.
4. Or invert the Normal of all **Projection Planes**.

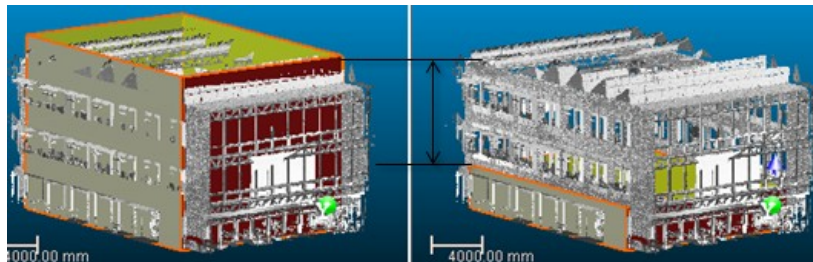
Notes:

- Instead of clicking a button in the **Multi-Ortho-Projection** dialog, you can also select its equivalent from the pop-up menu.
- You undo the change you have done and redo it again by selecting **Undo** and **Redo**.
- You can also select a **Projection Plane** in the **3D View** by picking it. Enlarge (or reduce) the selected **Projection Plane**'s height by dragging and dropping the two horizontal edges.

Top Aligning all Planes

To Top Align all Planes:


1. Click **Top Align all Planes** . The **Maximal Altitude** dialog opens.
2. Enter a new value in the **Maximal Altitude** field.
3. Or use the **Up** and **Down** buttons* to select a value.
4. Click **OK**. All **Projection Planes** are aligned to the top.

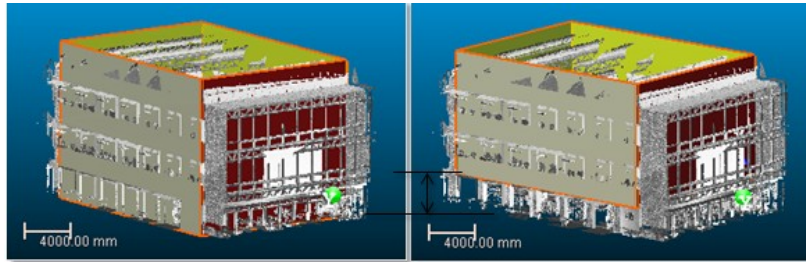


Note: (*) The value will be incremented (or decremented) of 5 millimeters. If the unit of measurement in use is too big, the user won't see the increment (or decrement).

Bottom Align all Planes

To Bottom Align all Planes:


1. Click **Bottom Align all Planes** . The **Minimum Altitude** dialog opens.
2. Enter a new value in the **Minimum Altitude** field.
3. Or use the **Up** and **Down** buttons* to select a value.
4. Click **OK**. All **Projection Planes** are aligned to the bottom.

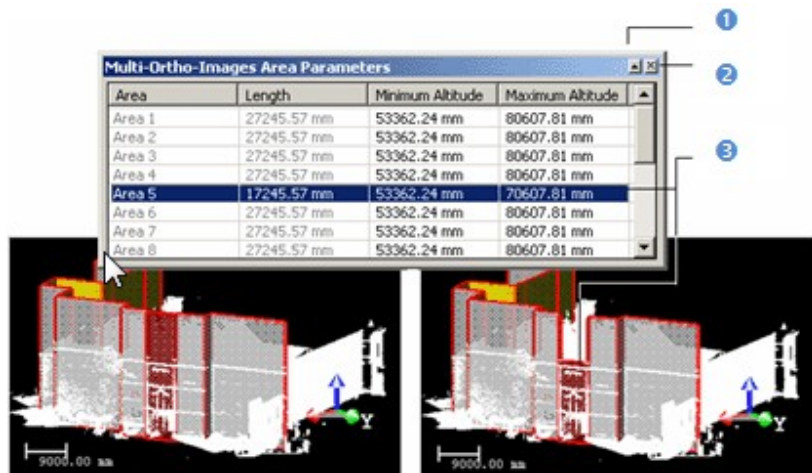


Note: The value will be incremented (or decremented) of 5 millimeters. If the unit of measurement in use is too big, the user won't see the increment (or decrement).

Editing Planes

To Edit Planes:

1. Click **Start Editing Table** . The **Multi-Ortho-Image Area Parameters** dialog opens.
 - Each area in the **Area** column corresponds to a unique **Projection Plane**.
 - The length of an area in the **Length** column is the gap between its **Minimum Altitude** and its **Maximum Altitude** along the Z direction.
2. Select an area* from the dialog by picking a line.
3. Click on a value in either the **Minimal Altitude** or **Maximum Altitude**. The value becomes editable.
4. Enter a new value and press **Enter**. The **Projection Plane** which corresponds to the selected area has its **Minimal Altitude** (or **Maximum Altitude**) changed as well as its height.
5. Click on the **Close** button at the top right corner of the dialog box.



1 - The Auto-hide button
2 - The Close button

3 - Only the Maximum Altitude of the selected Projection Plane is modified

The **Auto Hide** feature allows displaying more information using less screen space by hiding (or showing) the **Multi-Ortho-Image Area Parameters** dialog. When you press the **Auto Hide** button, the **Multi-Ortho-Image Area Parameters** dialog will hide away. Move the mouse pointer over the **Multi-Ortho-Image Area Parameters** title bar, it will slide out and will become visible. If you want the **Multi-Ortho-Image Area Parameters** dialog to stay visible after it has slid out, just press the **Auto Hide** button.

Tip: (*) You may see the **Projection Plane** corresponding to the selected area appears in **Step 4 - Preview Single Image** as well as its **Size**, **Minimum Altitude** and **Maximum Value**.

Hiding/Displaying the Input

You can also display/hide the input: point cloud or mesh.

To Hide/Display the Input:

1. Un-check to hide the input from the **3D View**.
2. And check to display it again.

Defining Image Parameters

You should define the parameters used for generating the ortho-images.

To Define the Image Parameters:

1. Define a **Resolution**.
2. Choose a **Density** to apply.
3. Define the **Depth** parameter.

Resolution

The user should define a **Resolution** to apply to the active **Projection Plane**. A **Resolution** is by default square.

To Set a Resolution:

1. Enter a value in the **Resolution** field.
2. Use the  and **Down**  to select a value.

Density

The **Density** expressed in terms of **PPI** (Pixels Per Inch) is automatically computed according the current value of the **Resolution**.

Rendering

You can choose an option to render the computed ortho-images(s). There are five options for a point cloud input: **Using Elevation** (the elevation value is calculated for each point based on its distance to the defined area. The calculated elevation value will be discretized in 8 bits to create the ortho-projection images), **Using Normal Shading** (the normal of each point will be used in shading calculation to create the ortho-projection images), **Using Grey Scaled Intensity** (the (grey) intensity value of each point will be used to create the ortho-projection images), **Using Color Scaled Intensity** (the (color) intensity value of each point will be used to create the ortho-projection images) and **Using True Color** (the color associated with each point will be used to create the ortho-projection images). There is only one option for a mesh input: **Textured Meshes**.

To Choose a Rendering Option:

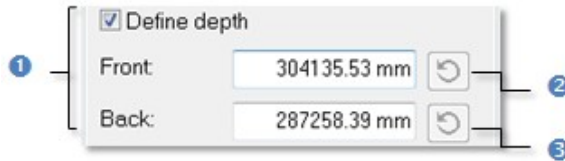
1. Click on pull down arrow below the **Density**.
2. Select a rendering option from the drop down list.

Depth

You also should select the **Define Depth** option in order to define the distances **Front** and **Back** all rectangular areas.

To Define the Depth Parameter:

1. Check the **Define Depth** option. The **Front** and **Back** fields become editable.



1 - Distance Front and Back all Projection Planes

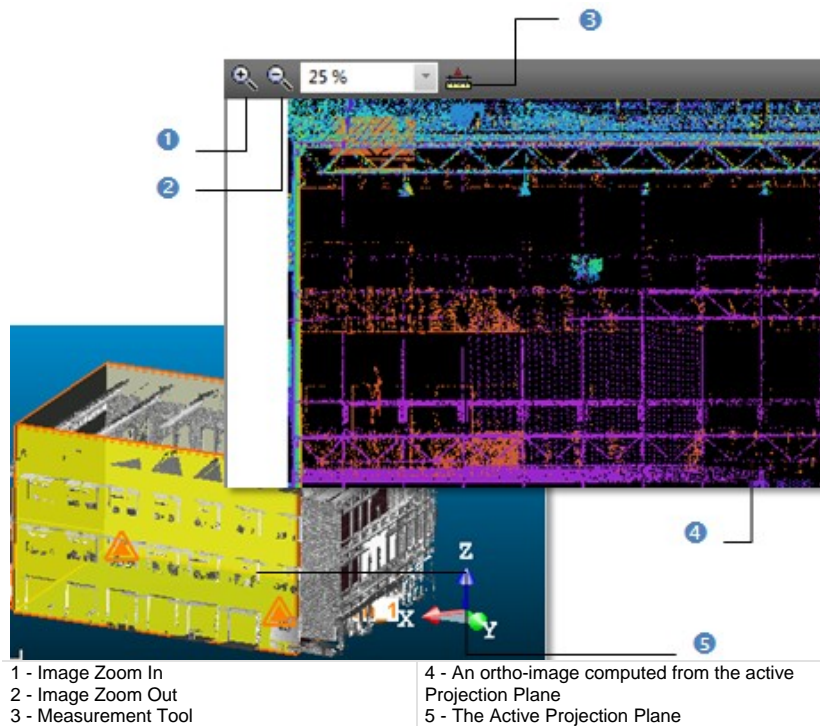
2 - Reload Front Depth
3 - Reload Back Depth

2. Enter a value in the **Front** field. The **Reload Front Depth** icon becomes enabled.
3. Enter a value in the **Back** field. The **Reload Back Depth** icon becomes enabled.
4. If required, load the initial value by pressing **Reload Front Depth**.
5. If required, load the initial value by pressing **Reload Back Depth**.

Previewing a Single Ortho-Image

To Preview a Single Ortho-Image:

1. Select a **Projection Plane**.
2. Click **Preview**. The computed ortho-image is shown in a 2D Image view and the **Preview** button becomes inactive.



In this view, you can zoom or make a 2D distance measurement. You can zoom according three ways. The first way is to magnify (or reduce) an area of the ortho-projection image using **Zoom In** and **Zoom Out**. The second way is to magnify (or reduce) the whole ortho-projection image using the mouse's wheel (if existed). The last way is to select a rate from the drop-down list.

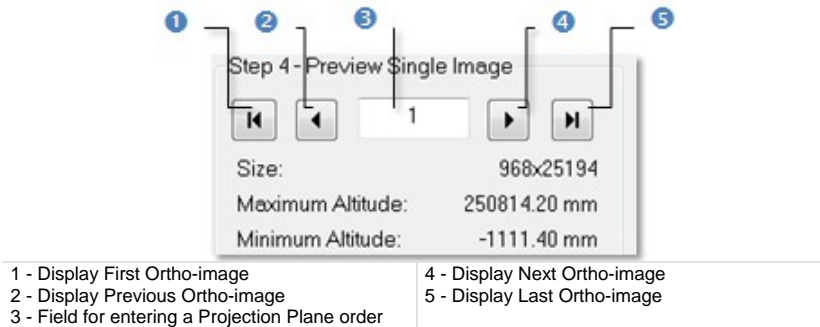
3. Select another **Projection Plane** (if required).
4. Click again **Preview**.

Note: You cannot go down under 10% when zooming out.

Tip: The color of the background on an **Ortho-Image** is the same as the one in the **3D View**. You can change it in **Preferences\Viewer**. This change will be then applied whatever the Rendering option you choose.

Selecting a Projection Plane

An active **Projection Plane** is the one which appears in the **Preview Single Image** field and is shown in red in the **3D View**.



To Select a Projection Plane:

- Click **Display Previous** (or **Next**) **Ortho-image** to set the previous (or next) **Projection Plane** as active.
- Click **Display First** (or **Last**) **Ortho-image** to set the first (or last) **Projection Plane** as active.
- Key in a **Projection Plane**'s order in the **Preview Single Image** field to select it. Do not need to validate by pressing the **Enter** key.

Tips:

- Use the **Home** (or **End**) button of your keyboard instead of **Display First** (or **Last**) **Ortho-image**.
- Use the **Up** (or **Down**) arrow of the keyboard instead of **Display Previous** (or **Next**) **Ortho-image**.

Notes:

- The active (current) **Projection Plane**'s parameters like its **Size**, **Maximum Altitude** and **Minimum Altitude** are displayed in **Step 4** of the **Multi-Ortho-Projection** dialog.
- The **Display Previous Ortho-image**, **Display Next Ortho-image**, **Display First Ortho-image** and **Display Last Ortho-image** buttons are dimmed if there is only one segment in the defined polyline.

Viewing from a Projection Plane' Side

The user can view the displayed scene from each of the active **Projection Plane'** side.

To View from a Projection Plane' Side:

1. Select a **Projection Plane**.
2. Click on the canonical view pull down arrow.
3. Choose among **Front View**, **Back View**, **Left View**, **Right View**, **Top View** and **Bottom View**.

Printing an Ortho-Image

You can print the preview of an ortho-image.

To Print an Ortho-Image:

1. Right-click in the 2D Image view.
2. Select **Print** from the pop-up menu. The **Print Setup** dialog opens.
3. Define a **Printer**.
4. Choose a **Paper' Size** and **Source**.
5. Choose an **Orientation** between **Portrait** and **Landscape**.
6. Add some comments in the **Legend** panel.
7. Set an **Appearance** between **As Is** and **Reverse Colors**.
8. Click **OK**.

Tip: You can also select **Print** from the menu bar or clicking its icon in the **Main** toolbar.

Creating Ortho-Images

You can create an ortho-image at a time or all at once in the database. Each ortho-image is named as follows: Cross-Object-Polyline's name-Segment's order. You can export each of them as a TIFF format file or via the DXF format to AutoCAD®. Note that you can create several series of ortho-images without quitting the tool.

A Single Ortho-Image

A single ortho-image is created and put under the current active group of the **Images Tree**.

To Create a Single Ortho-Image:

1. First preview a single ortho-image.
2. Click **Create**.
3. Click **Close**.

All Ortho-Images

A set of ortho-images is created and put in a folder under the current active group of the **Images Tree**. This folder is named as follows: Multi-Ortho-Polyline's name

To Create an Ortho-Image:


1. Click **Create All**.
2. Click **Close**.

Image Rectification Tool

This tool allows the creation of images rectified from perspective distortion by projecting station images onto a given 3D plane.

Opening the Tool

To Open the Tool:

- From the **OfficeSurvey** menu, select **Image Rectification Tool** . The **Image Rectification** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window. The navigation mode swaps to **Station-Based**.

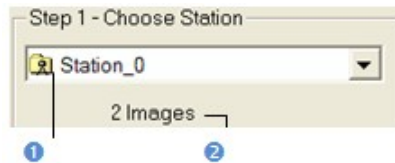
Tip: You can also click on the **Image Rectification Tool** icon in the **Tools** toolbar.

Choosing a Station

You need to select a station with at least an image inside.

To Choose a Station:

1. Click on the pull-down arrow.



1 - Station(s) under the current project

2 - Number of images under the selected station

2. Choose a station with images from the drop-down list.
3. Or use the navigation buttons (**Go to First Station**, **Go to Previous Station**, **Go to Next Station** and **Go to Last Station**) in the 3D View.
4. Or click inside the station number's field and select a station from the drop-down list.



1 - Go to First Station

2 - Go to Previous Station

3 - The current station

4 - Go to Next Station

5 - Go to Last Station

Defining a Projection Plane

There are two methods for defining a projection plane. The first one is by picking points and the second one is using the parameters of an already created rectified-image.

Picking Points

In the **Station-Based** mode, there are five tools for defining a projection plane; all are detailed below. In the **Examiner** (or **Walkthrough**) mode, only two are available (**Define Plane by Picking Three 3D Points** and **Define Vertical Plane by Picking Two 3D Points**); the others are disabled. You can swap from the **Station-Based** mode to the **Examiner** (or **Walkthrough**) mode and vice versa; Switching to the **Examiner** mode is typically useful for checking a plane that has been defined in the **Station-Based** mode.



1 - Define Vertical Plane by Picking Two Screen Points and One 3D Point	4 - Define Plane by Picking Three 3D Points
2 - Define Horizontal Plane by Picking Two Screen Points and One 3D Point	5 - Define Vertical Plane by Picking Two 3D Points
3 - Define Plane by Picking Three Screen Points and One 3D Point	6 - Define Plane Position by Picking on 3D Point

To Pick Points:

1. Define a 3D plane in the **Examiner/Walkthrough/Station-Based** mode.
2. Or define a 3D plane in the **Station-Based** mode.


Note: You can define a projection plane even if the selected station does not contain any image.

Defining a 3D plane in the Examiner/Walkthrough/Station-Based Mode

To Define a Plane by Picking Three 3D Points:

1. Click **Define Plane by Picking Three 3D Points** . The **Picking Parameters** toolbar appears in 3D constraint mode.
2. Pick three points (free or constrained) in the **3D View**.


To Define a Vertical Plane by Picking Two 3D Points:

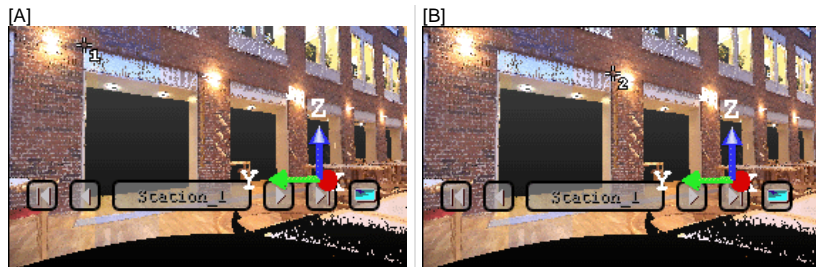
1. Click **Define Vertical Plane by Picking Two 3D Points** . The **Picking Parameters** toolbar appears in 3D constraint mode.
2. Pick two points (free or constrained) in the **3D View**.

Defining a 3D Plane in the Station-Based Navigation Mode

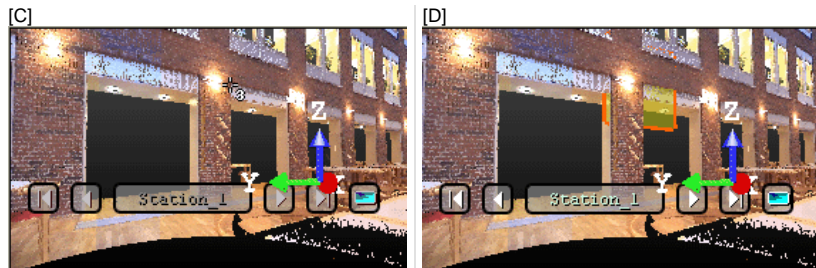
There are ten methods available in the **Station-Based** mode for defining a 3D plane. Three are specific to that mode: two are based on two screen points and a 3D point and one on three screen points and a 3D point. The other methods are already described in the **Examiner** (or **Walkthrough**) mode.

To Define a Vertical Plane by Picking two Screen Points and one 3D Point:


1. Click on the  button. The cursor becomes as shown in [A].
2. Pick a point anywhere on the screen. The cursor becomes as shown in [B].



3. Pick another point anywhere in the **3D View** (on the displayed point cloud or not). The cursor becomes as shown in [C] and the **Picking Parameters** toolbar appears in 3D constraint mode.
4. Pick the last point anywhere in the **3D View** (only on the displayed point cloud). A vertical plane appears with the third picked point as center.

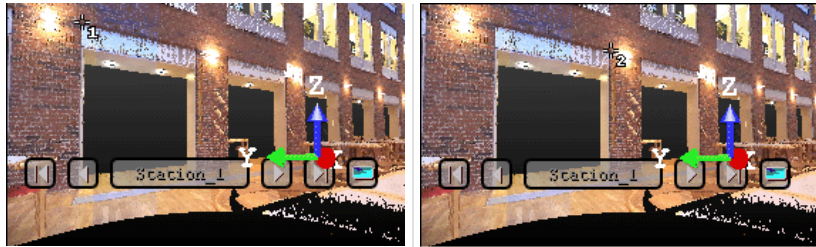


To Define a Horizontal Plane by Picking Two Screen Points and one 3D Point:

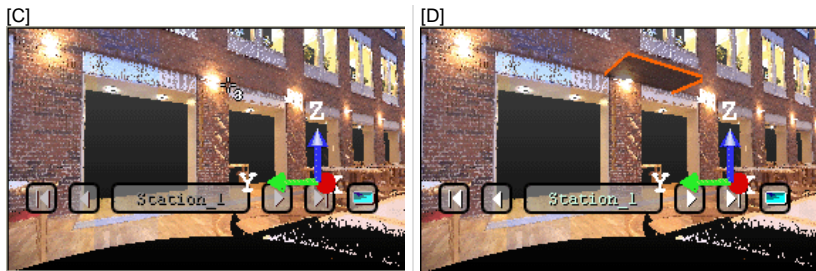
1. Click on the  button. The cursor becomes as shown in [A].
2. Pick a point anywhere on the screen. The cursor becomes as shown in [B].

[A]


[B]

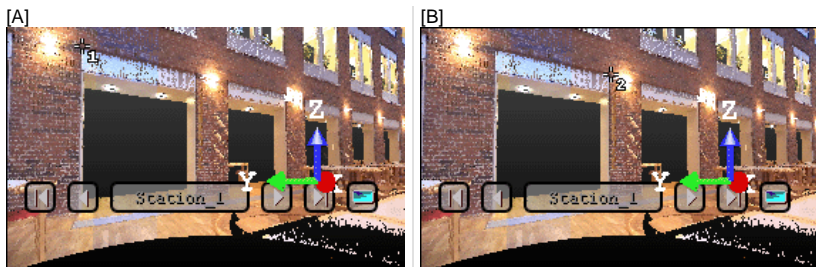


3. Pick another point on the screen. These two points will define the orientation of the first axis of the horizontal plane's frame. The cursor becomes as shown in [C] and the **Picking Parameters** toolbar appears in 3D constraint mode.
4. Pick the last point in 3D (on a cloud point, a measured point or a geometry). This point defines the height of the vertical plane. A horizontal plane appears with the third picked point as center.

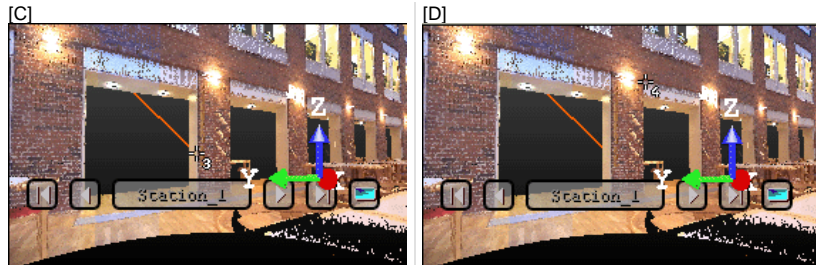


To Define a Plane by Picking Three Screen Points and one 3D Point (this Enables to Define a Plane with any Orientation):

1. Click on the  button. The cursor becomes as shown in [A].
2. Pick a point anywhere on the screen. The cursor becomes as shown in [B].






3. Pick another point on the screen so that the two points represent a horizontal segment in the 3D space. These two points define the orientation of a horizontal segment drawn on the final plane. The cursor becomes as shown in [C].
4. Pick another on the screen so that the previous point and this new one represent the steepest slope direction of the final plane. The cursor becomes as shown in [D] and the **Picking Parameters** toolbar appears in 3D constraint mode.



5. Pick the last point in the **3D View** (only on the displayed point cloud). The three first picked points - which are not collinear (not lying on the same line) - draw a 3D plane; the fourth picked point is its center.



Tip: ,  and  can be selected from the pop-up menu.

Loading Existing Parameters

If there are some rectified images in your project, the **From** button in the **Step 2 - Define Projection Plane** is active. You can select a rectified image and use its parameters for computing a new one.

To Load an Existing Rectified Image Parameters:

1. Click on the **From** button. The **Copy From Existing Rectified Image** dialog opens. All rectified images inside the project are listed.



2. Select a rectified image from the **Project Tree**. The **OK** button becomes active.
3. Click **OK**. The **Copy From Existing Rectified Image** dialog closes.

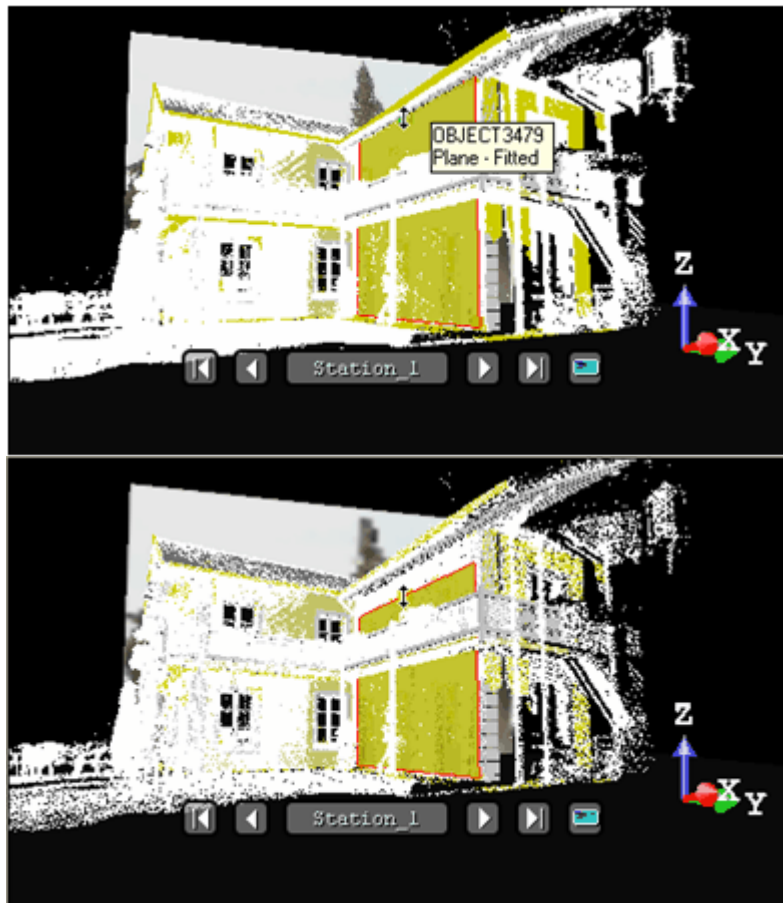
The rectified image parameters appear in **Step 3** and **Step 4** of the **Image Rectification** dialog. Its projection plane is shown in the **3D View** window.

Modifying a Projection Plane's Size

You can resize the previous projection plane. The resized projection plane keeps the same parameters as before except the dimensions. You can do this by dragging & dropping a corner in the **3D View** window.

To Modify a Projection Plane's Size:

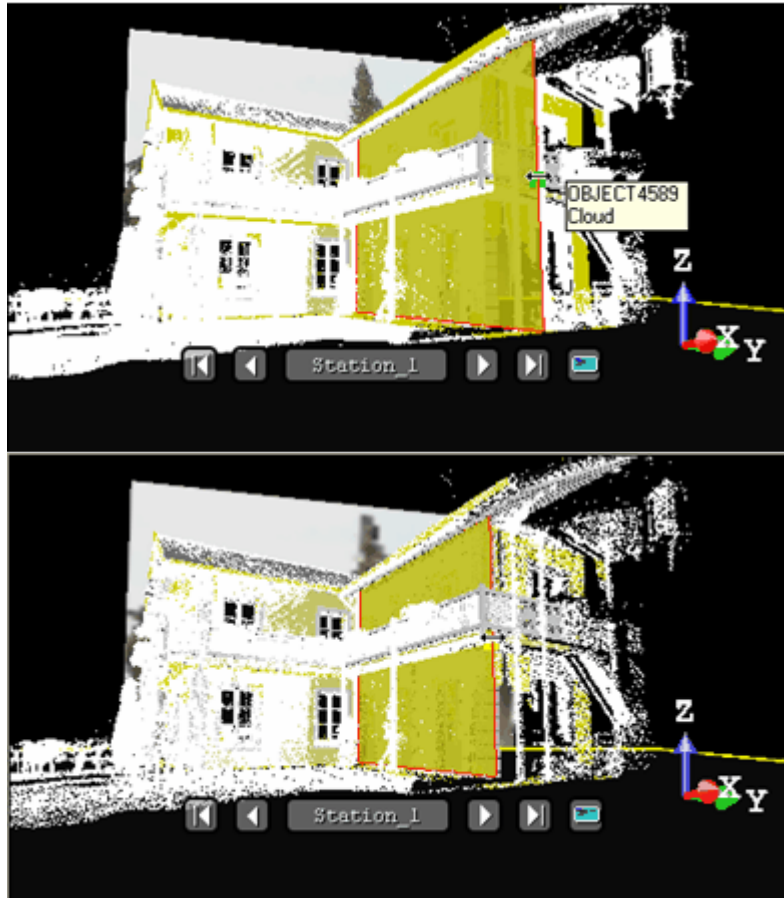
1. Place the mouse cursor upon any handle of a projection plane. A green square appears.
2. If a corner handle is selected, drag it to increase (or reduce) the projection plane size. During this operation, the green square becomes yellow.



1 - Handle before the drag & drop operation

2 - Handle during the drag & drop operation

3. If a middle handle is selected, drag it to increase (or reduce) the projection plane width (or length). During this operation, the green square becomes yellow.




1 - Handle before the drag & drop operation

2 - Handle during the drag & drop operation

Modifying a Projection Plane's Position

To Modify the Position of the Projection Plane:

1. Click **Define Plane Position by Picking One 3D Point** . The **Picking Parameters** toolbar appears in 3D constraint mode.
2. Pick a point (free or constrained) on the displayed point cloud in the **3D View**.

Defining a Zone of Interest

The objective of this step is to define a region (called **Zone of Interest**) on the previous projection plane onto which the station images will be projected. For this, you have to draw a rectangular frame. In the drawing mode, the 3D scene is locked. Before drawing a zone of interest, the entire projection plane is set as zone of interest and its size is shown in text in **Step 3** of the **Image Rectification** dialog.

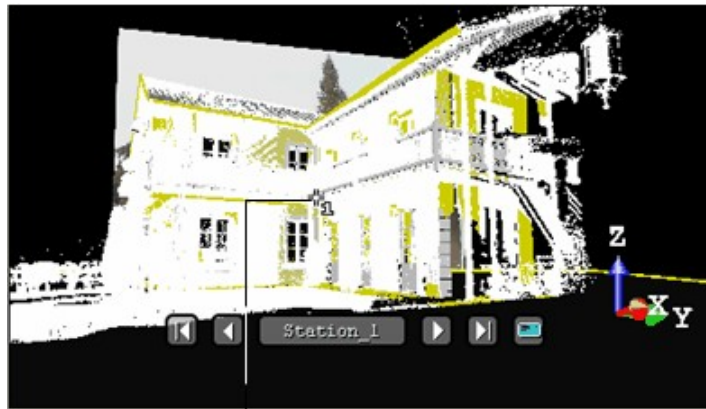
Notes:

- The size of the zone of interest is in the current unit of measurement (the one set in **Preferences**).
- If no projection plane has been defined in **Step 1**, the grayed-out **Area Undefined** message appears in the dialog and the **Draw** button remains dimmed.

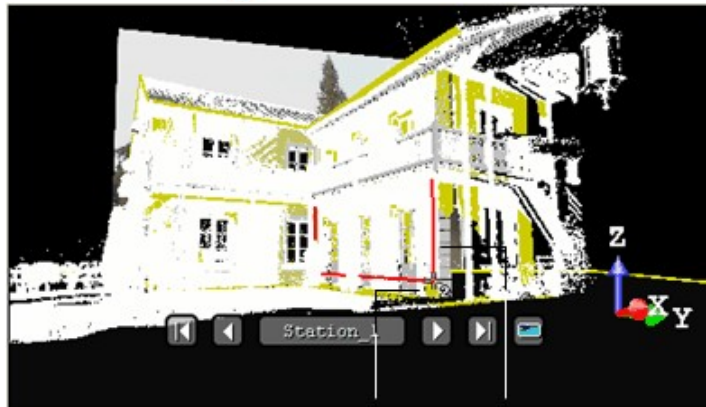
Drawing

To Draw a Zone of Interest:

1. Click **Draw**. The cursor will take the shapes as shown below and the projection plane is hidden.
2. Draw a rectangular frame by picking two points.



1



2

3

1 - Cursor shape of the first point to be picked

2 - Cursor shape of the second point to be picked

3 - Size of the interest zone

The mouse cursor shape changes. The arrow becomes a pointer to indicate that you are in the picking mode. A number beside this pointer guides you step-by-step in the point's selection. It starts from **One** that corresponds to the first point of the X* direction and ends by **Two**. Once the two points are reached, the interest zone is generated. You can cancel it whenever you want and start a new one.

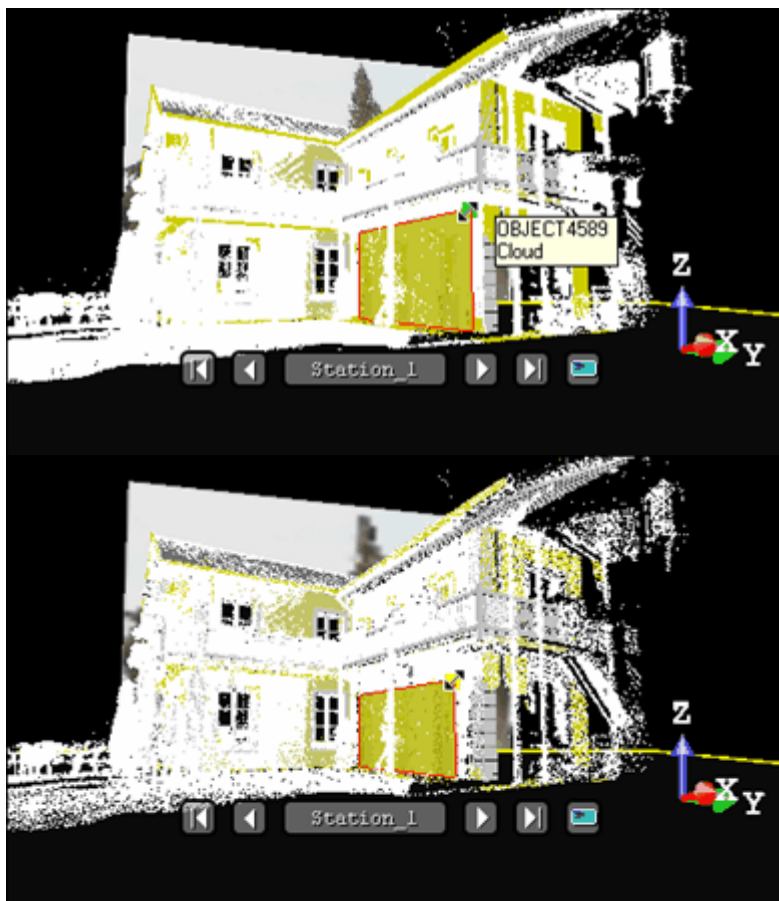
Notes:

- Pressing **Esc** cancels the selection of points in progress and leaves the drawing mode.
- (*) In the **X, Y, Z Coordinate System**.

Modifying

To Modify a Zone of Interest:

1. Move the cursor over a handle in green.



2. Drag-and-drop the handle to resize the rectangular frame.

Setting an Image Resolution

You can define the rectified image's resolution by giving the PPI (Pixels Per Inch), specifying the number of pixels in the X and Y directions or giving the pixel size. The pixel size is equal to 1 (if the unit of measurements is Inch) or to 25.4 (if the unit of measurement is millimeters) divided by the image resolution (in PPI). The image size (in pixels) is obtained by dividing the interest zone size by the pixel size.

To Set an Image Resolution:

1. Enter a value in the **Pixel Size** field. The image size (WxH) is updated according the new value.
2. Or click **Advanced**. The **Set Image Resolution** dialog opens.
3. Do one of the following:
 - Enter a value (in PPI) in the **Image Resolution (PPI)** field.
 - Enter two values (in pixels) in the **Image Size (WxH)** field.
 - Enter a value (in the current unit of measurement) in the **Pixel Size** field.
4. Click **OK**. The **Set Image Resolution** dialog closes.

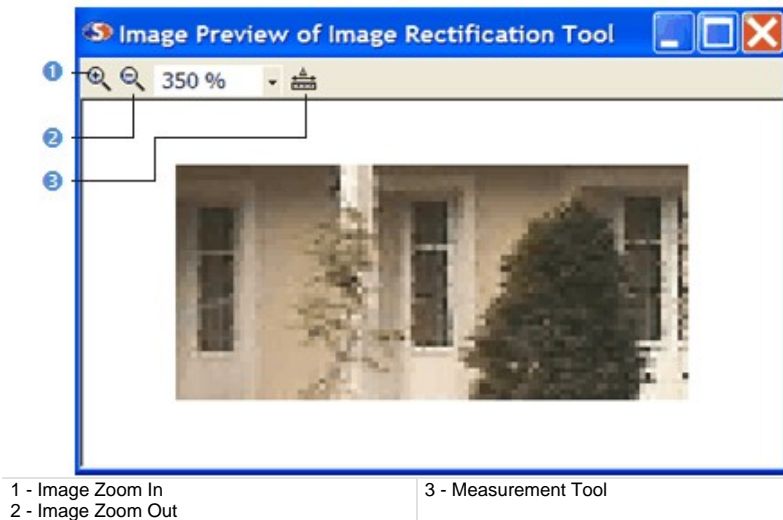
Tip: You can use the short-cut key **R** (or select **Set Image Resolution (R)** from the pop-up menu) to open the **Set Image Resolution** dialog.

Previewing a Rectified Image

You need to have at least one image inside the selected station to be able to preview the rectified image. Otherwise, the **Preview** button remains dimmed even if a projection plane and a zone of interest have been defined.

To Preview a Rectified Image:

- Click **Preview**. The computed image is shown in a **2D Image** viewer called **Image Preview of Image Rectification Tool**.



1 - Image Zoom In
2 - Image Zoom Out

3 - Measurement Tool

Before previewing the rectified-image, clicking **Close** leaves the **Image Rectification Tool** and the position of the rectified-image is lost. After previewing the rectified-image, clicking **Close** opens a dialog which prompts you to abort or continue the operation. In this **2D Image** viewer, you can zoom-in (or zoom-out). You can zoom in three ways. The first one is to magnify (or reduce) an area of the rectified-image using **Image Zoom In** and **Image Zoom Out**. The second way is to magnify (or reduce) the rectified-image using the mouse wheel (if existed). The third is to select a zoom factor from the drop-down list. If the rectified-image is bigger than the **2D Image** viewer can show, you can pan it on left-click in any direction.

Tip: You can select **Preview Rectified Image** from the pop-up menu.

You can also make a 2D-distance measurement. As the metric information is stored in the calculated rectified-image, the measurement is accurate.

To Make a Measurement:

1. Click **Measurement Tool**.
2. Pick two points on the rectified-image. The measurement distance is shown in text in the **2D Image** viewer.
3. Select **Create Measurement** from the pop-up menu.
4. Select **Close Tool** from the pop-up menu.

Tip: You can also use the **Esc** key instead of selecting **Close Tool** from the pop-up menu.

Note: Pressing **Esc** without saving the measured distance will cancel that distance.

To Print the Rectified-Image:



1. Right-click in the rectified-image.
2. Select **Print** from the pop-up menu.

Tip: You can also select **Print** from the **File** menu (or click on the **Print** button in the **Main** toolbar).

Splitting a Rectified Image

For rectified images of large size, you can split them into rectified images of small size. Splitting can be done along a rectified image's width, along its height or in both directions.

To Split a Rectified Image:

1. Enter a number in **W** (or **H**) field.
2. Or use the **Up**  (or **Down** ) button to select a number in the **W** (or **H**) field.

Creating a Rectified Image

You can now create the rectified image in the database. The created image will be put under the current active group of the **Images Tree**. Note that the newly created image has “ortho-image” as type. You can create several rectified images without quitting the tool.

To Create a Rectified Image:

1. Click **Create**.
2. Click **Close**.

Tips:

- You can also right-click in the **3D View** and select **Create Rectified Image** from the pop-up menu.
- Once the zone of interest has been defined (3D rectangle), defining a new plane will merely modify the position and orientation of the zone of interest. In particular, the size of the rectangle is kept. This can prove useful when producing a series of rectified images of a facade (to keep the height and vertical position of the rectangle).


Note: You do not need to first preview a rectified image to create it in the database.

Polyline Drawing Tool

This tool allows you to create quickly a polyline drawing by successively picking on the displayed object(s) or not. The final result is a polyline which you can export as a 3D polyline in DXF (or DGN) format, or as a 2D polyline in DXF format for coplanar polyline. You can use the **Polyline Drawing Tool** in both the **3D View** and the planar view (as used in the **2D-EasyLine™ Tool**). In the latter case, the created polyline is a planar one. You can use this to carry out inspections.

Opening the Tool

To Open the Tool:

1. Select and display an object (point cloud or geometry).
2. From the OfficeSurvey menu, select **Polyline Drawing Tool** .
3. Or if you are in a main tool like e.g. the **2D-EasyLine™ Tool**, click on the **Polyline Drawing Tool** button.

Tip: You can also click the **Polyline Drawing Tool** icon in the **Tools** toolbar.

Note: No selection is necessary to access the tool. Anything that is displayed in the **3D View**, whether selected or not, can be picked.

Defining a 3D Plane

You can use the **3D Plane Tool** to define a plane and draw polylines on it. The **Drawing Tool** toolbar looks as shown in [A] and [B] when you are respectively in the **Examiner/Walkthrough** and **Station-Based** mode.

[A]



[B]



1 - Start 3D Plane Tool

2 - Shown/Hide Plane

3 - Lock in 2D

To Draw in a 3D Plane:

1. Click the **Start 3D Plane Tool** icon. The **3D Plane Tool** toolbar and a 3D plane both appear.
 - In the **Examiner** (or **Walkthrough**) mode:
 - a) Define and validate a 3D plane. The **Show/Hide Plane** icon becomes enabled and the **Lock in 2D** icon is set by default. The scene is locked on the defined 3D plane with a 2D grid superimposed (if not hidden previously) and the **Picking Parameters** toolbar appears in the 2D constraint mode.
 - b) Click the **Show/Hide Plane** icon to display the defined plane.
 - In the **Station-Based** mode:
 - a) Define and validate a 3D Plane. The **Show/Hide Plane** and **Lock in 2D** icons both become respectively enabled and set by default.
 - b) Click the **Show/Hide Plane** icon to hide the defined plane (if required).
 - c) Use the **Manipulator** to move the 3D plane.
2. Draw a polyline in that plane.

Tip: You can also select the **Start 3D Plane Tool** icon from the pop-up menu.

Notes:

- In the **Polyline Drawing Tool**, you can swap from a navigation mode (**Examiner/Walkthrough/Station-Based**) to another as often as required.
- In the **Station-Based** mode, the **Lock in 2D** icon remains unavailable before defining a 3D plane.
- If a 3D plane has been selected as input of the **Polyline Drawing Tool**, the 3D scene will be locked on that plane with a 2D grid superimposed (if not hidden previously).

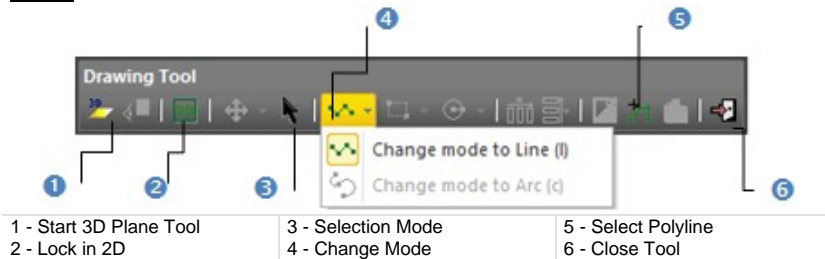
Drawing Polylines

The basic tool to draw a polyline is the action of picking which can be free or constrained. The **Polyline Drawing Tool**, split into three drawing modes (**Polyline**, **Rectangle** and **Circle**), behaves differently according to the input data that you select for using it. In the **Chain** mode, if the input data is an object like pure 3D point cloud or geometry, you use this tool in the **3D View** and you can only draw segments (that's why **Change Mode to Arc** is dimmed). If the input data is an object like a slice of point cloud (see the **Cutting Plane Tool**) or a fitted **Polyline** (see the **Contouring** or **Profile/Cross-Sections Tools**), you use this tool locked in the planar view and you can draw only segments or a combination of segments and circular arcs. In the **Rectangle** and **Circle** modes, the input data can only be of 2D type.

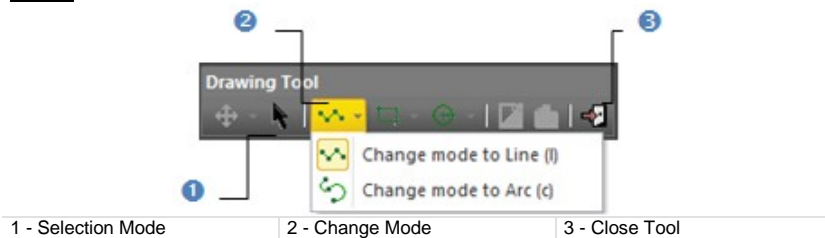
Chains of Segments/Arcs

If you are in 3D, the **Drawing Tool** toolbar opens as shown below and the **Picking Parameters** toolbar (in the 3D constraint mode) appears. Note that the **Change Mode to Arc**, **Draw Circle** and **Draw Rectangle** icons are unavailable. If you are in 2D, the **Drawing Tool** toolbar looks as shown here below and the **Picking Parameters** toolbar in the 2D constraint mode (H/V or Angle/Distance) appears. The **Change Mode to Arc**, **Draw Circle** and **Draw Rectangle** icons are available.

In 3D:



In 2D:

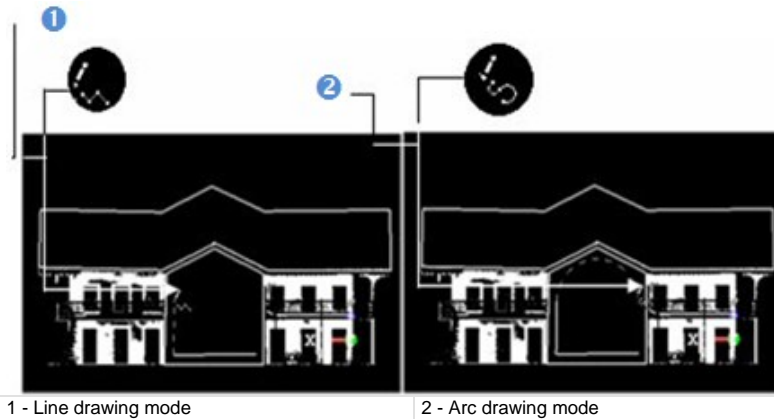


To Draw in the Chain Mode:

1. Pick a point to start the first node of a polyline.
2. Navigate through the 3D scene and pick another point. A segment links these two points.
3. Click on the **Change Mode** pull-down arrow.
4. Choose **Change Mode to Arc (C)** from the drop-down list.
5. Navigate through the 3D scene and pick another point. The newly picked point is linked to the previous picked point by an arc.
6. Click on the **Change Mode** pull-down arrow.
7. Choose **Change Mode to Line (l)** from the drop-down list.
8. Navigate through the 3D scene and pick another point. The newly picked point is linked to the previous picked point by a segment.

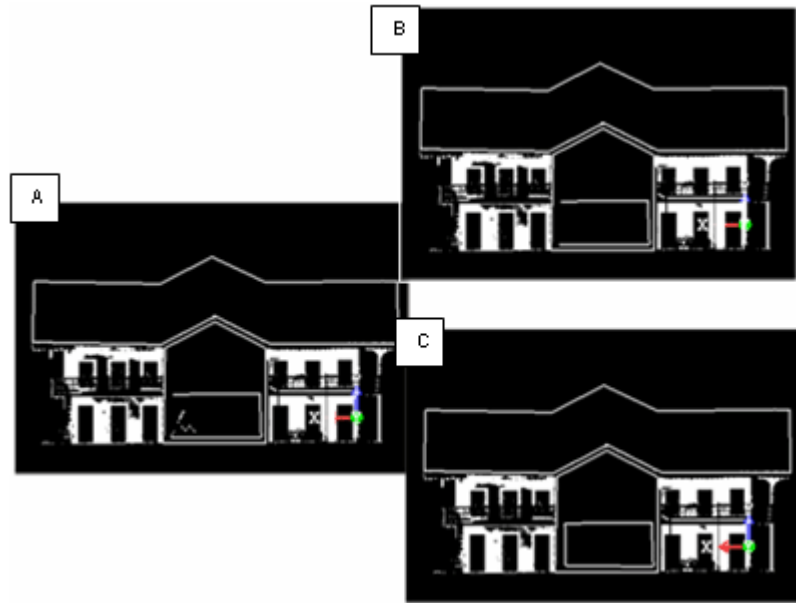
Notes:

- The steps from 3 to 7 are only available in 2D and pickings can be done anywhere in the planar view, on cloud point/geometry or not.
- In 3D, pickings should be done on displayed object(s) and can be constrained on a cloud point using the **Ctrl** key with the left-click. This means that you cannot pick anywhere except on the point.



Tip: You can switch from the line drawing mode to the arc drawing mode and conversely as often as you wish just by pressing respectively the **L** and **C** keys on your keyboard, by clicking on the **Change Mode** button in the **Drawing Tool** toolbar or by selecting its related command from the pop-up menu.

9. Continue in picking in order to define the other nodes of the polyline.
10. Right-click anywhere in the **3D View** to display the pop-up menu.
11. Select **End Line** to terminate the polyline. The start node is not linked to the last selected node.
12. Select **Close Line** to end and close the polyline. The start node is linked to the last selected node.



A - The first selected node

B - End Line: The start node is not linked to the last selected node

C - Close Line: The start node is linked to the last selected node

Notes:

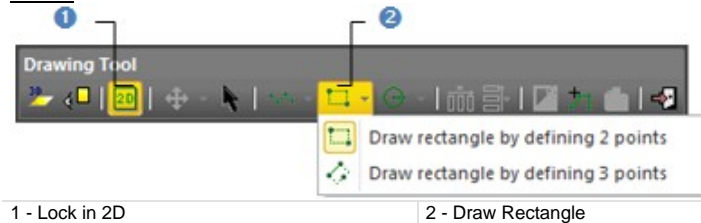
- Pressing **Esc** while you are picking points will end and validate (but not create) the polyline in progress.
- Each time you validate a polyline by double-clicking or by using the **End Line** and **Close Line** commands; you can continue to draw other polylines. These polylines will not be connected.

Tip: You can double-click to end drawing. In this case, the drawn polyline is always an open one.

Rectangles

To draw a polyline in the **Rectangle** mode when the input is of 3D data type, first click the **Lock in 2D** icon to lock the **3D View** in 2D (or after defining a 3D plane using the **3D Plane Tool**, the **Draw Rectangle** icon becomes enabled). The scene is then locked in 2D with a 2D grid superimposed (if not hidden previously) and the picking mode switches from 3D constraint to 2D constraint. Clicking again the **Lock in 2D** icon frees the scene from 2D lock and 2D constraint picking mode. If the input is of 2D data type; you do not need to lock the scene in 2D because it is (by definition) locked in 2D plane.

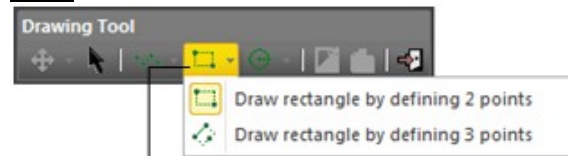
In 3D:



1 - Lock in 2D

2 - Draw Rectangle

In 2D:




Draw Rectangle

To Draw a Rectangle:

1. Click on the **Draw Rectangle** pull-down arrow.
2. Choose **Draw Rectangle by Defining 2 Points** from the drop down list.
3. Pick a point. A node ■ appears. This sets up the first corner of a rectangle.
4. Move your cursor to setup the opposite corner. The node disappears and a rectangle in dotted appears. Its shape changes as long as you move the cursor.
5. Pick a point. A rectangle is then drawn.

Or

6. Click on the **Draw Rectangle** pull-down arrow.
7. Choose **Draw Rectangle by Defining 3 Points** from the drop down list.
8. Pick a point. A node ■ appears. This sets up the first end of a rectangle' side.

9. Move your cursor to setup the opposite end. The node disappears and a segment in dotted appears. Its shape changes as long as you move the cursor.
10. Pick a point. The segment in dotted becomes continuous and another node  appears.
11. Move your cursor to setup the opposite end. The node disappears. Three other sides in dotted and perpendicular to the first side appear. Their length changes as long as you move the cursor.
12. Pick a point. A rectangle is then drawn.

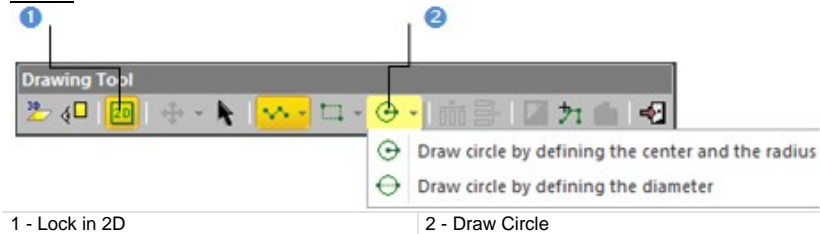
Note: You can switch from the 2-point drawing mode to the 3-point drawing mode and conversely as often as you wish just by clicking on the **Draw Rectangle** icon.

Tip: You can also select **Lock in 2D**, **Draw Rectangle by Defining 2 Points** and **Draw Rectangle by Defining 3 Points** from the pop-up menu.

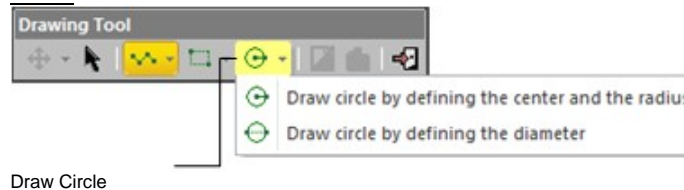
Circles

To draw a polyline in the **Circle** mode when the input is of 3D data type, first click the **Lock in 2D** icon to lock the **3D View** in 2D (or after defining a 3D plane using the **3D Plane Tool**, the **Draw Circle** icon becomes enabled). The scene is locked in 2D with a 2D grid superimposed (if has not been hidden previously) and the picking switches from 3D constraint mode to 2D constraint mode. Clicking again the **Lock in 2D** icon frees the scene from 2D lock and 2D constraint picking mode. If the input is of 2D data type; you do not need to lock the scene in 2D because it is by definition locked in 2D plane.

In 3D:



In 2D:



To Draw a Circle:

1. Click on the **Draw Circle** pull-down arrow.
2. Choose **Draw Circle by Defining the Center and the Radius** from the drop-down list.
3. Pick a point. It will be the center of a circle to come.
4. Navigate through the 3D scene and pick another point. These two points will form the radius of a circle.

Or

5. Click on the **Draw Circle** pull-down arrow.
6. Choose **Draw Circle by Defining the Diameter**.
7. Pick a point to start the first point of a circle's diameter.
8. Navigate through the 3D scene and pick another point to set the second point of the diameter.



Note: You can switch from the center-and-radius drawing mode to the diameter drawing mode and conversely as often as you wish just by clicking on the **Draw Circle** button.

Tip: You can also select **Lock in 2D**, **Draw Circle by Defining the Center and the Radius** and **Draw Circle by Defining the Diameter** from the pop-up menu.

Selecting Polylines

When the input contains already created polylines (or after drawing and creating some), you can set one of them as selected by picking. Note that this is only available in 3D (or 3D locked in 2D).

To Select a Polyline:





1. Click **Select Polyline** . The cursor becomes as .
2. Pick a polyline. A polyline (in green) appears over the picked polyline.
 - If a 2D polyline has been picked, the scene is locked in a 2D plane that contains the picked polyline (the **Lock in 2D** in the **Drawing Polyline** toolbar is default set and dimmed) with a 2D grid superimposed (if not hidden previously). The **Picking Parameters** toolbar appears in the 2D constraint mode.
 - If a 3D polyline has been picked, the scene remains unlocked and the **Picking Parameters** toolbar remains in the 3D constraint mode.

Notes:

- Press **Esc** to leave the picking mode.
- If required, hide all objects displayed in the **3D View**; this can help you in picking a polyline.
- If you are in 3D locked in 2D, after choosing **Select Polyline** and before picking a polyline, the 3D scene is free from the 2D lock and after picking a polyline the 3D scene comes back to the 2D lock state.

Tip: You can also check **Select Polyline** from the pop-up menu.

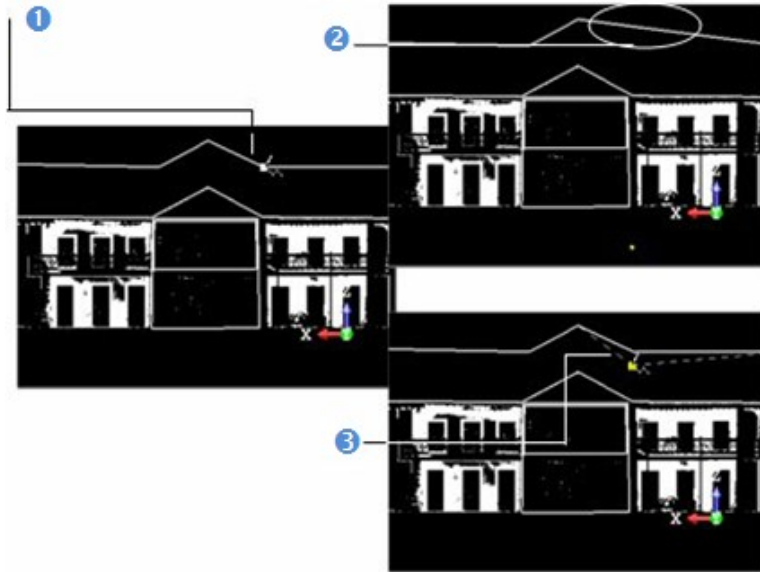
Editing Polylines

Before creating a polyline, you can delete the whole of it or modify it by deleting, moving and inserting nodes or by continuing it. When you place the cursor over a segment of a polyline, you may see the following symbols:  (Nodes),  (Middle Nodes) and  (Middle Nodes to Insert). When you place the cursor over an arc of a polyline; only  (Nodes) are available.

Deleting Nodes

To Delete a Node:

1. Place the cursor over a node. A solid square appears upon the node.
2. Right-click to display the pop-up menu and select **Delete Node**.



1 - The selected node on the polyline

2 - The selected node is removed from the polyline

3 - The selected node is moved

Notes:

- Deleting the **Start** (or **End**) node of a chain of segments will remove the **First** (or **Last**) segment from that chain. Deleting a **Conjunction** node will delete the segment on both side of that node.
- Deleting a node at the end of a lonely segment won't delete that segment.
- You cannot delete a node of a lonely arc.

Moving Nodes

To Move a Node:

1. Place the cursor over a node. A solid square appears upon the node.
2. Drag the node to a position. The green square turns to yellow. If the node belongs to a segment; that segment becomes dotted. If it belongs to an arc, the arc shape does not change.
3. Drop the node to that position. Note that in the case of drawing in the **3D View**, the new position should be on displayed objects.

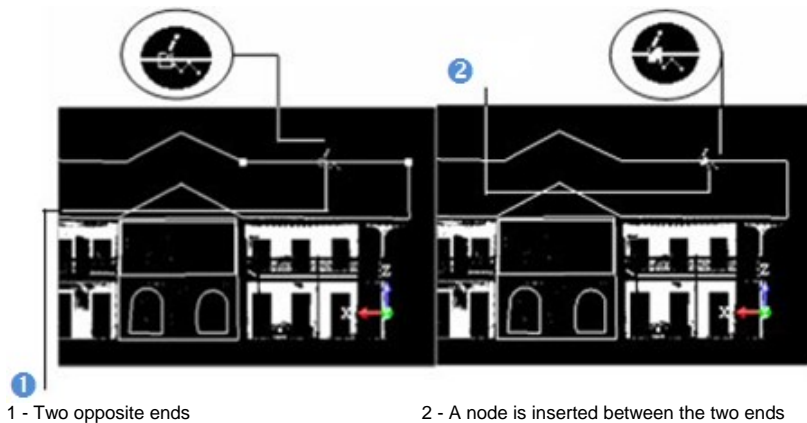
Notes:

- Moving a node at the end of a segment (or arc) will only move that node.
- Moving a node at the middle of a segment will move the whole segment.
- Picking a point anywhere on a segment except on the end and middle nodes or on an arc except on the end nodes will transform that point to a node.

Inserting Nodes

To Insert a Node:

1. Place the cursor anywhere on a segment (except at the end/middle nodes) or on an arc (except at the end nodes). A hollow square appears upon the segment at the cursor position.
2. Right-click to display the pop-up menu and select **Insert Middle Node**. A new **Middle Node** is inserted not at the picking position but at the middle of the segment (or arc).



Continuing Polylines

To Continue a Polyline:

1. Place the cursor over the end (or start) node of a polyline or anywhere over the last segment of a polyline.
2. Right-click to display the pop-up menu and select **Continue**. A dotted line appears between the cursor and the selected node if end (or start) node has been chosen or between the cursor and the last node if the last segment has been chosen.
3. Left-click anywhere in the **3D View** to continue the polyline.



1 - Selected node on the polyline

2 - A segment is added after the selected vertex

Note: You cannot continue in drawing a closed polyline.

Deleting Polylines

In 3D (or 2D), after drawing a polyline, the **Create**, **Delete Polyline** and **Delete All** icons become enabled. Note that the **Delete Polyline** icon is not present on the toolbar but can only be reached from the pop-up menu.

In 3D:



In 2D:



1 - Delete All

2 - Create

A Single Polyline

If there is a lonely polyline that has been drawn, this polyline will be deleted. If there are several polylines, the last polyline will be deleted.

To Delete a Lonely Polyline:

1. Right-click anywhere in the **3D View**.
2. Select **Delete Polyline** from the pop-up menu.

Note: You can also delete a polyline while drawing it (or after validating it).

Tip: To delete a polyline that is already created, please use the **Undo** button. It is not necessary to close the **Polyline Drawing Tool** for that.

All Polylines

To Delete all Polylines:

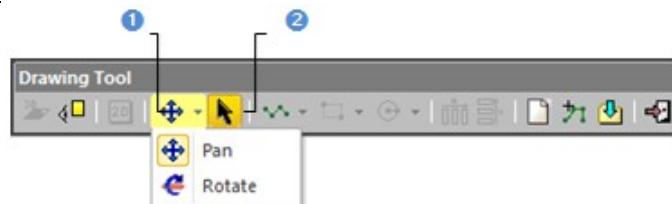
- Click **Delete All**.

Note: You can also delete a polyline that is set as selected (using the **Select Polyline** command).

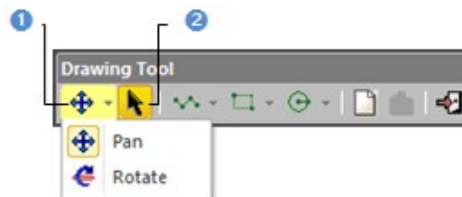
Moving Polylines

After drawing a polyline (or setting an already created one as selected), you can use the displacement mode to **Pan** or **Rotate** the polyline within the displayed scene. The displacement mode which comes first is the one chosen during the last use of that tool.

In 3D:



In 2D:



1 - Change Move Mode

2 - Selection Mode

Tip: You can choose the **Selection Mode** from the pop-up menu.

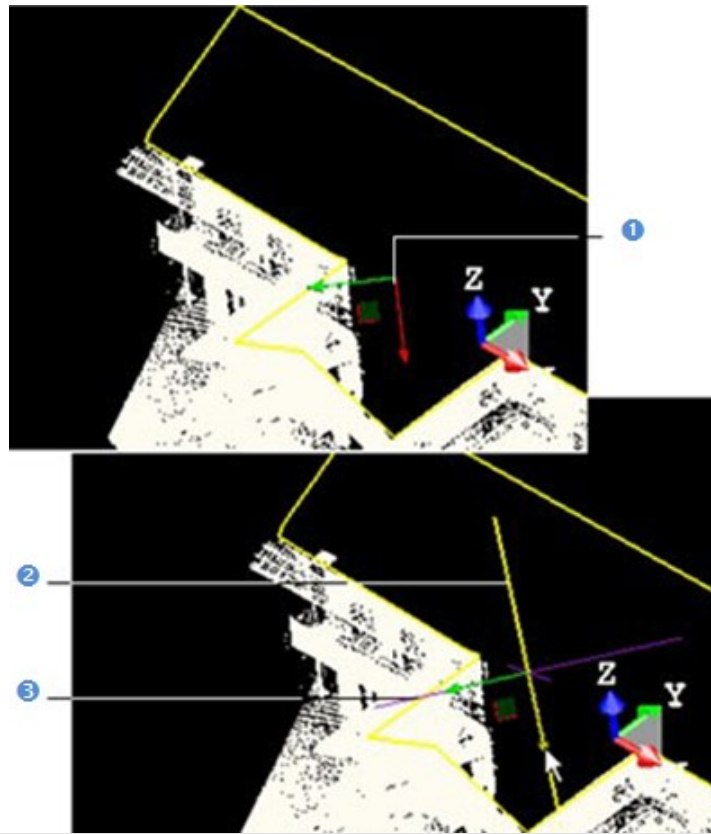
Note: Moving an already created polyline (which is set as selected) will not displace it but only the selection (the green polyline which appears over the created polyline).

Pan

After choosing **Pan**, a **Manipulator** in a plane parallel to the screen view appears. If there is a polyline selected, (or when there are several polylines drawn), the **Manipulator** has as origin the selected (or last drawn) polyline's center. You can move the selected (or last drawn) polyline along a direction at once or anywhere in the **Manipulator**'s plane.

To Pan:

1. Click on the **Change Move Mode** pull-down arrow.
2. Choose **Pan** from drop-down list. A **Manipulator** appears. It is composed of two **Axis-Handles** and one **Plane-Handle**.
3. If you are in 3D, rotate lightly so that the plane - inside which the **Manipulator** is - is unparallel to the screen view.
4. Pick an **Axis-Handle** to select it; it turns to yellow. The direction along which you can move the selected polyline is highlighted in yellow. Those (two in all) for which you cannot move the selected polyline are in mauve. Move the selected polyline along that direction.



1 - Manipulator (with three handles)

2 - Allowed direction (in yellow)

3 - Forbidden directions (in mauve)

5. Or pick on the **Plane-Handle**. The two **Axis-Handles** remain with their own color and the two directions along which you can move the selected polyline are highlighted in yellow. The forbidden direction - normal to the plane - is in mauve. Move the selected polyline in that plane.
6. If you are in 2D, you can only move the selected polyline along a direction. You may only see one forbidden direction in mauve. If you move the selected polyline in a plane, you may not see any forbidden direction.

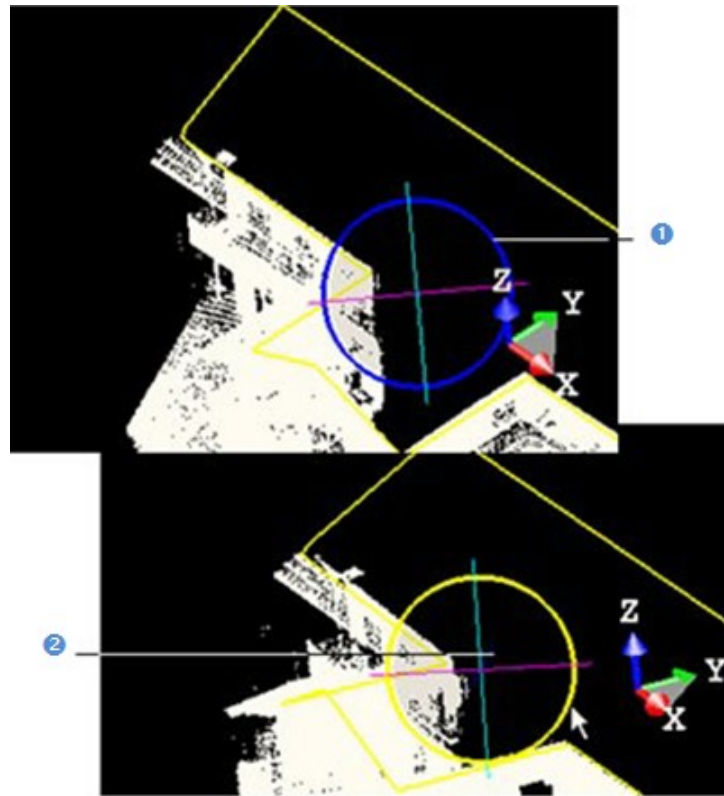
Tip: You can also select first **Change Move Mode** from the pop-up menu and then **Pan**.

Rotate

After choosing **Rotate**, a **Manipulator** of ring shape (in deep blue) with two extended diagonals (in light blue and mauve) appears in a plane parallel to the screen view. If there is a polyline selected (or when there are several polylines drawn), the **Manipulator** has as origin the selected (or last drawn) polyline's center. You can rotate the selected (or last drawn) polyline around an axis perpendicular to the ring's plane.

To Rotate a Polyline:

1. Click on the **Change Move Mode** pull-down arrow.
2. Choose **Rotate** from the drop-down list. A **Manipulator** appears. It is of ring shape with two extended and perpendicular diagonals.
3. If you are in 3D, rotate lightly so that the plane - inside which the two extended diagonals are - is unparallelled to the screen view. You may see the direction - around which you can rotate the selected polyline - in dashes.
4. Pick the ring; it turns to yellow. The direction around which you can rotate the selected polyline tilts to deep blue. Rotate the selected polyline around that direction.



1 - Ring Manipulator

2 - Axe of rotation (in deep blue)

5. If you are in 2D, do the same procedures than step 4. You may not see any forbidden direction.

Tip: You can also right-click to display the pop-up menu and select first **Change Move Mode** and then **Rotation**.

Auto-Duplicating Polylines

In the **Station-Based** mode, you can duplicate a polyline in two directions (**Horizontal** or **Vertical**). The polyline needs to be a 2D polyline or a 3D coplanar polyline (all nodes in the same plane).



1 - Auto-Duplicate Horizontally (Image-Based)

2 - Auto-Duplicate Vertically (Image-Based)

Tip: You need to have matched images within your station(s).

Horizontally

To Auto-Duplicate Horizontally a Polyline:


1. Set a polyline as selected.
2. Or draw a new 3D coplanar polyline.



3. Click **Auto-Duplicate Horizontally (Image-Based)** to duplicate the selected (or drawn) polyline horizontally.



4. Click **Auto-Duplicated Vertically (Image-Based)** to duplicate the selected (or drawn) polyline vertically.

Tip: You can also select a part of an already duplicated polyline using the **Selection Mode**  and duplicate it horizontally.

Vertically

To Auto-Duplicate Vertically a Polyline:


1. Set a polyline as selected.
2. Or draw a new 3D coplanar polyline.



3. Click **Auto-Duplicate Horizontally (Image-Based)** to duplicate the selected (or drawn) polyline horizontally.




4. Click **Auto-Duplicated Vertically (Image-Based)** to duplicate the selected (or drawn) polyline vertically.

Tip: You can also select a part of an already duplicated polyline using the **Selection Mode**  and duplicate it vertically.


Manually Duplicating Polylines

You can manually duplicate a polyline in any navigation mode (**Examiner**, **Walkthrough** or **Station-Based**). The duplication direction is the one given by the manipulator which appears when using the **Selection Mode**.

To Manually Duplicate a Polyline:

1. Select a drawn polyline using the **Selection Mode** . The **Change Move Mode** becomes enabled. The selected polyline color swaps from green to yellow and a manipulator (with two handles (Green and Red)) appears over it.
2. Select **Copy Selection and Create** from the pop-up menu. You cannot see the duplicated polyline because it is upon the original polyline.
3. If required, click on the **Change Move Mode** pull-down arrow.
4. Choose between **Pan** and **Rotate**.
5. Move the duplicated polyline in consequence. The duplicated polyline becomes selected (yellow) and the original polyline unselected (green).

Notes:

- You can also use the following short-cut key **Ctrl + D**.
- (*) If your polyline has been already created in the **RealWorks** database, first set it using the **Select Polyline**  command.

Creating Polylines

Once you are satisfied with the drawn polyline(s), you can create it (or them) in the database. The newly created object will be put in the current active folder under the **Models Tree**.

To Create a Polyline:

1. Click **Create**.
2. Click **Close Tool**.

Note: If you draw several unconnected polylines, they will be created into the same polyline.

Tips:


- You can also select **Create** from the pop-menu or press **Enter**.
- You can also select **Close Tool** from the pop-menu or press **Esc**.

Image Matching Tool

The **Image Matching Tool** allows you to match an imported image to a displayed 3D scene, or in other words, to find the camera position from which the image is shot. The basic principle behind this tool is to select at least four pairs of markers (points or segments or a combination thereof); each of them should be selected on the same physical objects. For example, you can select a point from the 3D scene, and another one in the 2D image, both correspond to the corner of a room. We call these two points a pair of markers. Then **RealWorks** will use these selected markers to calculate a best registration (or a best camera position) so that when you view from this camera position and along its axis, the image and the 3D scene will superposed. Once the image is matched to the 3D scene, you can use it to color the scanned points, to texture the meshed model, or just to enhance understanding of the scene.

Opening the Tool

To Open the Tool:

1. Select a point cloud (or mesh) from the **Models Tree**.
2. Select an image (or more) from the **Images Tree**.
3. From the **OfficeSurvey** menu, select **Image Matching Tool** . The **Image Matching** dialog opens as well as the **Picking Parameters** (in 3D constraint mode) toolbar.

This dialog opened as the third tab of the **WorkSpace** window is separated into four parts. The first part enables to select an image that will be used as reference for matching. The second part contains tools to select and edit markers. The third part enables to preview the matching, view stored and matched images and match under constraint. The last part enables to validate the matching and color points. The selected image is shown as a thumbnail in the left top corner of the **3D View**. If more than one image are selected, the first in the selection list is the one that is shown in the **3D View**.

Note: The **Image Matching** icon is not present in the **Tools** toolbar.

Selecting an Image

You should select an image from the **Select Image** combo box for matching. Only unmatched and already matched images are listed in that combo box.

To Select an Image:


1. Click on the **Select Image** pull-down arrow.
2. Select an image from the drop-down list.



Tip: Inside the **Image Matching Tool**, you can go back to the **Models Tree** tab to display or hide objects for marker selection purposes.

Hiding (or Showing)

To Hide (or Show) a Thumbnail:

1. Click on the  **Hide/Show Image** icon to hide the thumbnail.
2. And click again on the **Hide/Show Image** icon to display the thumbnail.

Tips:

- The **Hide/Show Image** icon can also be selected from the pop-up menu.
- You can also click anywhere in the **3D View** and use the short-cut key **V**.

Maximizing (or Minimizing)

To Maximize (or Minimize) a Thumbnail:

1. Click on the  **Maximize/Minimize Image** icon to make full the thumbnail.
2. And click again on the **Maximize/Minimize Image** icon to restore the thumbnail size.

Tips:

- The **Maximize/Minimize Image** icon can also be selected from the pop-up menu.
- You can also click anywhere in the **3D View** and use the short-cut key **M**.

Extending (or Stretching)

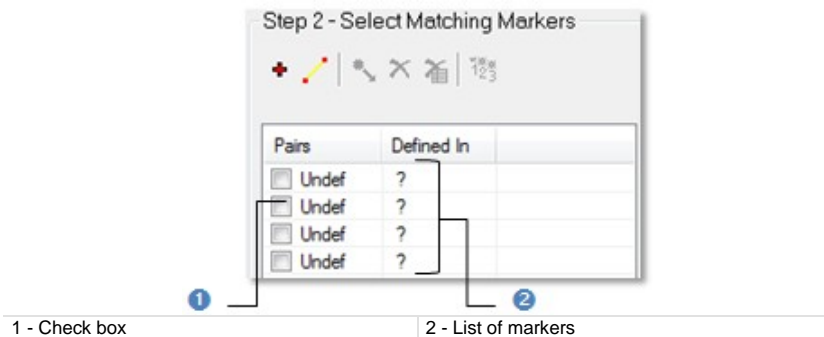
You can drag and drop a vertical (or horizontal) edge of the thumbnail image to extend it in width (or length).

Moving

Place your cursor over the symbol  and drag and drop the thumbnail image to a location within the **3D View**.

Selecting Markers

For a given image, if a matching has been already performed, you can view the stored matching. If any matching has been performed and when no information about the shooting position are available, you can start matching by selecting at least four pairs of markers. Before selecting, each pair is set as "Undef" in the **Pairs** column of the marker list and "?" in the **Defined In** column.



To Selecting Markers:

1. Pick a pair of makers.
2. Or Load a set of maker pairs.

Notes:

- Press twice the **Esc** button to close the **Image Matching Tool**.
- In the marker list table, each selected marker pair will be shown. You can see in the second column whether the pair is completely defined, or if it is just partially defined.

Picking Markers

You should pick at least four pairs of markers. Each pair may be either a 3D Point/2D Point pair or a 3D Line/2D Line pair. The order of these pairs has no influence on the final calculation. So you can pick them whenever you find a corresponding marker pair (no matter its type) in both **3D View** and **2D View** (for image). Movements are restricted in the **3D View** to **Pan**, **Zoom** and **Rotate** while picking markers.



To Pick Markers:


1. Pick a pair of points.
2. Or pick a pair of lines.
3. Or pick a combination of point pairs and line pairs.

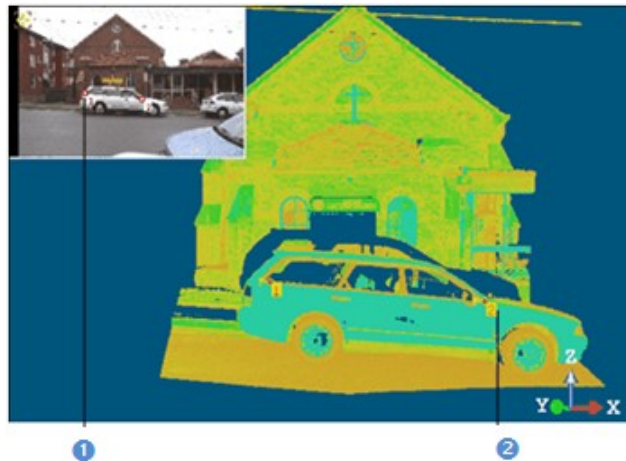
Notes:

- Pressing **Back** (or once on **Esc**) while the picking is in progress cancels the selected markers.
- A pair of points is always put at the first position in the marker list when you select a combination of points and lines.

Points

To Pick a Pair of Points:

1. Select an image that will be used as a reference image for matching.
2. Navigate in the 3D scene so that the displayed scene aligns approximately with the chosen image.
3. Click the **Add Point Marker**  icon. You are now in the picking mode.
4. Pick a point marker on the reference image.
 - This point is shown by a red square with One as order.
 - In the marker list, "Undef" becomes "Point#1" in the **Pairs** column and "?" swaps to "[2D]/" in the **Define In** column.
5. Pick a point marker on the 3D scene.
 - This point is shown by a yellow **P** with One as order.
 - In the marker list, "[2D]/" in the **Define In** column becomes "[2D]/[3D)".
 - A check mark beside this pair means that it is taken into account for the matching.
6. Repeat the steps from 4 to 5 for the three other pairs of point markers.



1 - 2D Point marker

2 - 3D Point marker


Notes:

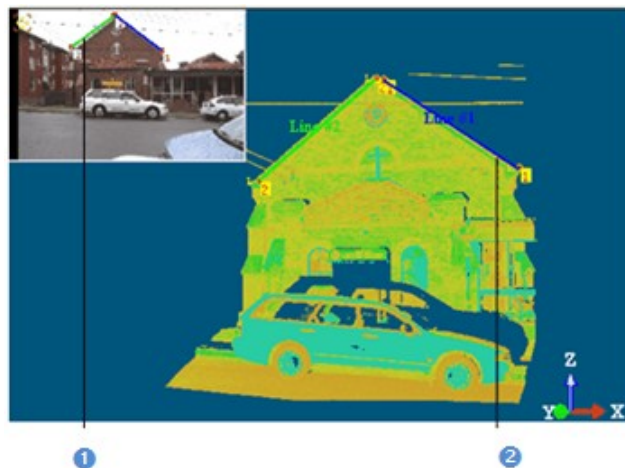
- Point markers can be selected by pair (one from a view and one from the other view) as described above or continuously (four from a view and four the other view).

- When you select a pair from the list, its related point marker **P** on the object becomes green (the number label remains in yellow) and the red square in the reference image becomes pink (the number remains unchanged).

Lines

To Pick a Pair of Lines:

1. Select an image from the selection list that will be used as a reference image for matching.
2. Click on the **Add Line Marker**  icon. You are now in the picking mode.
3. Pick in the 3D scene to set at least four line markers (if any point has been set).
4. Pick in the reference image to set at least four line markers (if any point has been set).



1 - 2D Line marker

2 - 3D Line marker

The mouse cursor shape changes. The arrow becomes a pointer with two numbers beside. The first number is a line order and the second to the picked point order. A point once picked on the selected object is shown by a yellow **L** with a number besides indicating a line marker order. Two points from the same rank once picked are linked by a color segment which is named **Line # X**. **X** is the line marker rank. A point once picked on the reference image is shown by a pink square marker with a number beside indicating a line marker order and a line is drawn through this point to the current mouse cursor position. Two points from the same rank once picked are linked by a color segment which named **Line # X**. **X** is the matching line rank. When a pair of line markers from the same rank (one in the reference image and one in the cloud) is defined, this pair is named **Line # X** and is put in a list. **X** is the pair rank. A check mark besides a pair of line markers means that this pair is taken into account for matching. Un-select a check mark if you don't want to take into account a pair. When you select a pair from the list, its related line markers are lengthened.

Loading Markers

You can import an image that has been already matched, load and use its parameters for matching new images. Usually, such parameters are stored in a TXT (or ASCII) format file.

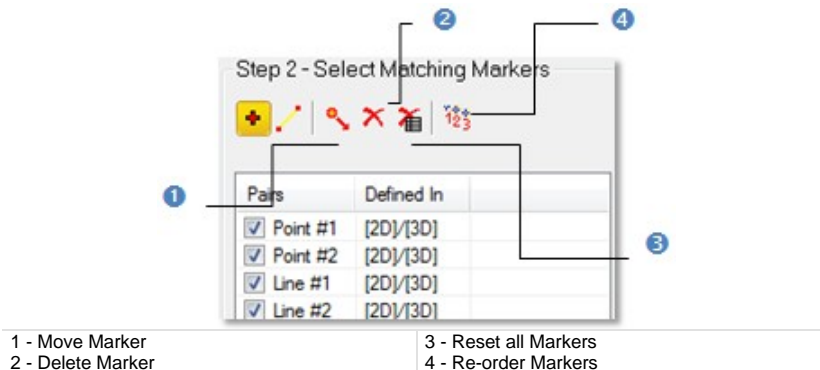
To Load Markers:

1. Right-click anywhere in the **3D View** to display the pop-up menu.
2. Select **Load Marker Pairs From File**. The **Open** dialog box opens.
3. Select a file type (*.txt or *.asc) from the drop-down list in the **File of Type** field.
4. Navigate to the drive/folder where the file to loaded is in the **Look In** field.
5. Select the file. Its name appears in the **File Name** field.
6. Click **Open**.

Tip: You can also use the short-cut key **L** instead of selecting **Load Marker Pairs From File** from the pop-up menu.


Modifying Markers

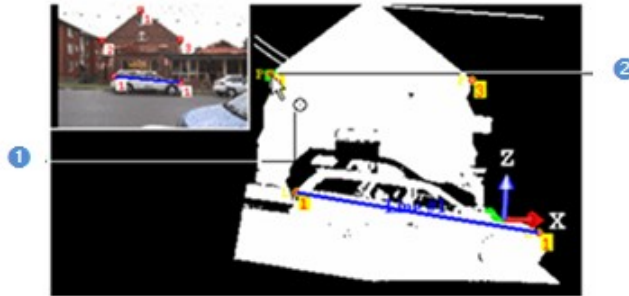
Once four pairs of markers (**Points** or **Lines**) are set, you can delete those that are not correctly set for matching, re-order a part (or the whole) of them, modify their location in each view or reset them all. Four tools, available for editing the marker features, are described hereafter.



Moving

To Move a Point (or Line):

1. Click on the **Move Markers**  icon. The cursor shape changes as shown in the illustration below.
2. Move the cursor over any marker in either the 2D image or the **3D View**. The marker will be highlighted in pink in the 2D image and in green in the **3D View**.
3. Drag and drop the marker to a new position. Its corresponding marker in the other view will not be moved too.




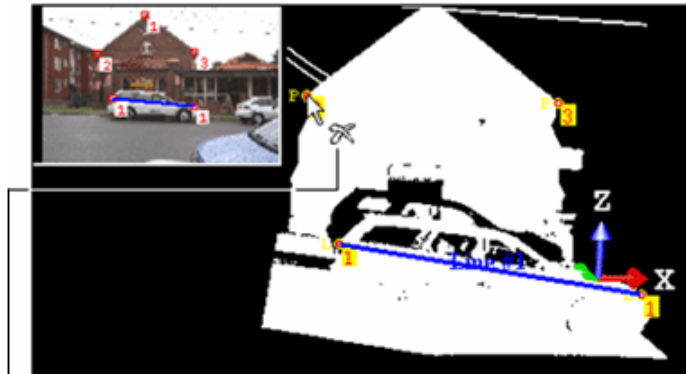
1 - The cursor shape in the displacement mode

2 - The marker is in green when highlighted in the 3D View

Deleting

To Delete a Point (or Line):

1. Click the **Delete Makers**  icon. The cursor shape changes as shown in the illustration below.
2. Move the cursor over any marker either in the image or in the 3D scene. The marker will be highlighted.
3. Click on the marker to delete it. Its corresponding marker in the other view will is also deleted.




The cursor shape in the deletion mode

When you place the mouse cursor upon a point (P), its color tilts from red to pink in the reference image and from yellow to green in the point cloud (or mesh). This means that the point (P) is selected and can be deleted. Deleting a point (P) will remove it from both the reference image and the point cloud (or mesh). A point (P) once deleted is replaced by one which is just after it and the pairs list is automatically updated. Once all markers deleted, you go back automatically to the selection mode. When you delete a line (L), you delete by the same way the line (L) in the other view.

Tip: You select a pair of markers from the marker list table and press the **Del** key to delete it.

Resetting


To Reset all Points (or Lines):

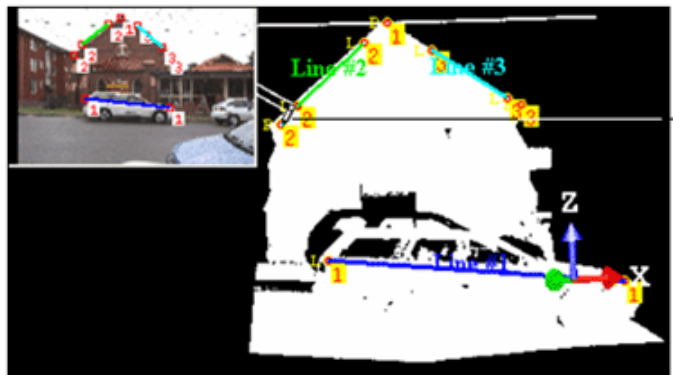
1. Click the **Reset All Markers**  icon. A question box opens.
2. Click **Yes** to delete the current set of point/line markers.
3. Click **No** to abort the operation.

Tip: The **Reset All Markers** icon can also be selected from the pop-up menu.

Reordering

To Reorder Points (or Lines):

1. Click the **Reorder Markers**  icon. The cursor shape changes as shown in the illustration below.
2. Move the cursor over any marker in either the **2D Image** or the **3D View**.
3. Pick the marker. Its order* is reversed with that of the first marker. The order of its corresponding marker in the other view does not change.



Cursor shape in the reordering mode

Note: (*) Only if it is not the first in the selection order.

Saving Markers

You can save the selected marker pairs into a text file and reload it later on to continue your image matching work.

To Save Markers:

1. Right-click anywhere in the **3D View** to display the pop-up menu.
2. Select the **Save Marker Pairs To File** command. The **Save As (or Open)** dialog box opens.
3. Select a file type (*.txt or *.asc) from the drop-down list in the **File of Type** field.
4. Navigate to the drive/folder where you want the file to save is in the **Look In** field.
5. Enter a file name in the **File Name** field. The selected extension will be added automatically.
6. Click **Save**.

Tip: You can also use the short-cut key **S** instead of selecting **Save Marker Pairs To File** from the pop-up menu.

Matching an Image

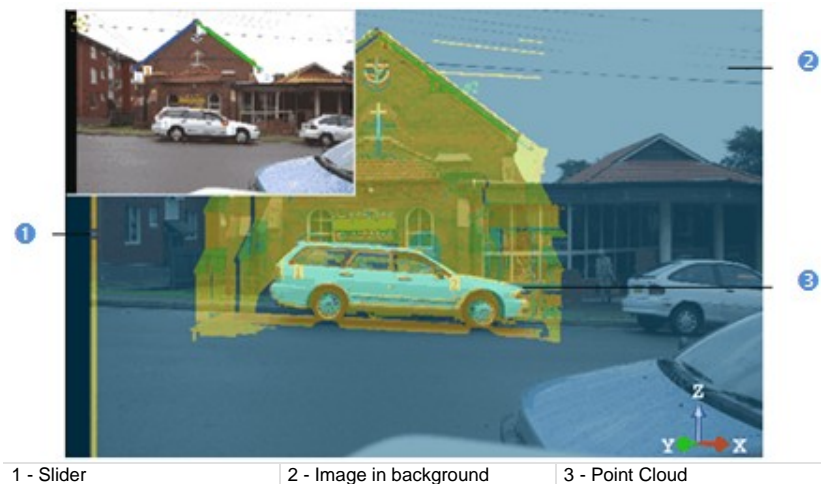
With an unmatched image, once four pairs of markers have been selected, the **Preview** button becomes enabled. You can proceed to a preview. With an already matched image, you can view the stored matching.

Previewing

You can now preview the matching. The image will be projected into the **3D View**. It is displayed in the background and blended with the 3D scene. You can use the slider to change the blending parameter in order to check the coincidence of the image with the 3D data. The markers set on the displayed scene appear on the reference image, this enables the user to visually compare them with those set on the reference image.

To Preview an Image Matching:

1. In **Step 3** of the **Image Matching** dialog, click **Preview**.



2. If required, extend the thumbnail image and compare the markers.



Gap between markers set on the reference image and on the displayed scene

3. If required, click **Hide**. The image in background will be removed from the **3D View**.

Tips:

- Instead of using the slider, you can click anywhere in the **3D View** and use the **Up** and **Down** keys of your keyboard.
- You can click anywhere in the **3D View** and use the short-cut key **A** to set the blending parameter to **0** (slider in **Low** position) or to **1** (slider in **Top** position).

Note: You can hide the image by right-clicking anywhere in the **3D View** and selecting **Projected Image**.

Improving

Where necessary, you can improve the matching by adding more markers, de-selecting some from the calculation (by using the check box in the marker list), or modifying the position of certain markers, and then re-perform the preview.

Adjusting

You can also manually adjust the 3D scene over the projected image by small movements. To do this, you can use either the free or the constrained movements. Free movements consist of using basic navigation tools (**Zoom**, **Pan** and **Rotate**) for moving the 3D scene while constrained movements require the use of constraint tools.

To Adjust an Image Matching:


1. Constrain to a pair of markers.
2. Or constrain to two pairs of markers.

Note: Because you need to adjust the 3D scene to cover the projected image by small movements in all directions, the **Head Always Up (Z axis)** option in **Preferences** if checked is then disabled.

Constraining to a Pair

You can select either a 3D Point/2D Point pair or a 3D Line/2D Line pair as constraint.

To Constrain to a Marker Pair:


1. Select a pair of markers from the marker list which you consider offer the best coincidence
2. The first **Constrained to Marker Pair**  icon becomes active. Click on it.
3. Go to the **3D View**, and move the scene. You can see that the movement is constrained in such a way that the selected marker pair will always be coincident.

Tip: To unset a pair of markers as constraint, click again on the **Constrained to Marker Pair** icon.

Constraining to Two Pairs

If the first constraint is a 3D Point/2D Point pair, you can add a new constraint which should be necessary a 3D Point/2D Point pair. You may see in the third column (called **Constraint**) of the marker list table if a pair is selected as constraint (or not). The first constrained pair is shown as follows ☒ Point #4 | [2D]/[3D] | [1]. The second constrained pair is shown as follows ☒ Point #3 | [2D]/[3D] | [2].

To Constrain to Two Marker Pairs:

1. Select another pair of markers from marker list. The second **Constrained to Marker Pair**  icon becomes enabled.
2. Click on the **Constrained to Marker Pair** icon.
3. Go to the **3D View**, and move the scene. You can see that the movement is constrained in such a way that both the selected pairs will always be coincident.

Tip: To unset a pair of markers as constraint, click again on the **Constrained to Marker Pair** icon.

Projecting

If the selected image is already matched, the **Project** button in **Step 3** of the **Image Matching** dialog becomes enabled. Clicking on it allows you to view the matching.

Coloring Points

You can now use the **Coloring** button to color the displayed cloud(s) with the currently matched image. It is important to note that this coloring operation has no **Undo**. After the coloring, the color attributes of the points can be changed again if you use this function with another image.



1 - Colored points

2 - Uncolored points remain in white

Note: The **Coloring** operation may take a long time for huge datasets.

Applying the Matching

If you are satisfied with the matching result, you can use the **Apply** button to save it to the database. It is important to note that this operation has no **Undo**. The selected image becomes matched.

To Apply the Matching:

1. Click **Apply**.
2. Click **Close**.


Volume Calculation Tool

The **Volume Calculation Tool** enables to compute the volume between a point cloud and a plane, between two point clouds, between a point cloud and a mesh or between two meshes. The volume computation is based on a grid method and the result is represented in the **3D View** by a graph of vertical color lines with scale. You can choose in the **Preferences** dialog the units to represent the computed volumes.

Opening the Tool

You need to select one or two surfaces from a project. You cannot exceed three.

To Open the Tool:

1. Select one or two surfaces from the **Project Tree**.
2. From the OfficeSurvey menu, select **Volume Calculation Tool** . The **Volume Calculation** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window and is composed of six parts. The first part contains two sub-tools (**Segmentation** and **Sampling**). The second part enables to define a plane. The third part lets you to set its resolution. The fourth part enables to preview the volume computation result, display (or hide) the selected surface(s), check the computed volume(s) to keep and edit a report. The fifth part is to edit the computed volume(s). The sixth part is to save the volume computation result, and close the tool.

If one surface has been selected, this surface will be automatically displayed in the **3D View** with its own color. If two surfaces have selected, the first (by selection order) will be set as **Reference Surface** and displayed in **Red** and the second as **Comparative surface** and in **Green**.

Notes:

- If the selected surface is a mesh, these two sub-tools (**Segmentation** and **Sampling**) are unavailable.
- If the input contains a point cloud and a mesh; the mesh will be by default **Reference Surface** and will not able to change to the **Comparison Surface**.

Defining a Plane

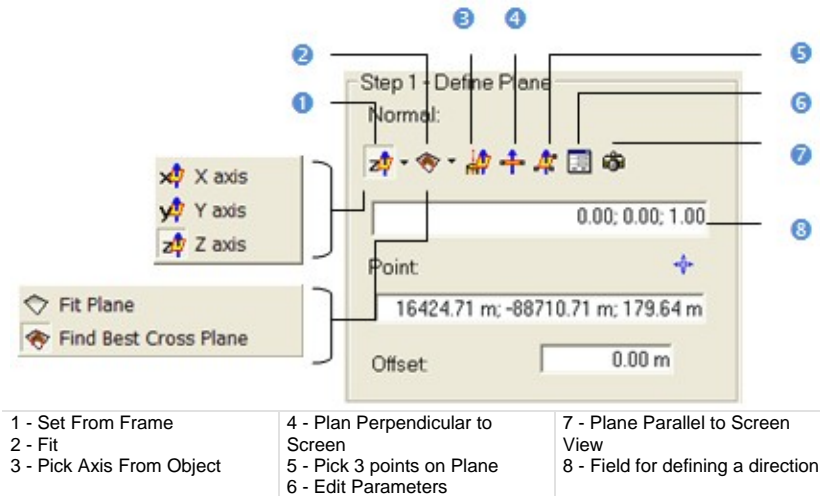
A plane perpendicular to the **Z**-axis of the active coordinate frame appears.

To Define a Plane:

1. Define a direction for the **Normal**.
2. Set a **Position** for the plane.
3. Define the **Offset**.

Normal

It is up to the user to orientate the initial plane. There are several tools available for this purpose.



To Define a Normal Direction:

Do one of the following:







- Select one of the frame's axes.
- Fit an extracted set of points with a plane.
- Find a perpendicular view plane from an extracted set of points.
- Pick one of the object's axes.
- Pick a plane perpendicular to the screen.
- Pick three points (1).
- Set the plane parallel to the screen view.

Note: After selecting (1), the **Picking Parameters** toolbar appears. You can pick free or constrained points.

Selecting a Frame Axis



This method consists of selecting an axis from the active frame as **Normal** direction. The initial plane will be moved so that its **Normal** will be parallel to the selected axis. Its position in the 3D scene will be kept and its **Offset** (altitude) will be reset.

To Select a Frame Axis:

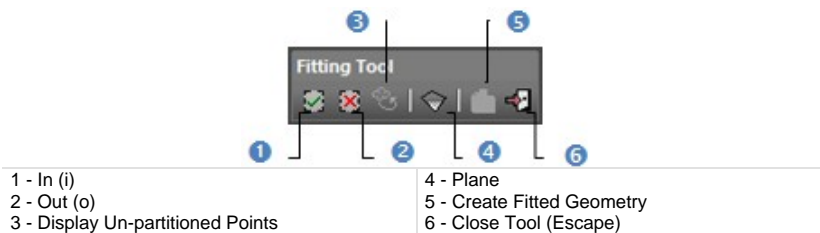
1. Click on the **Set from Frame** pull down arrow.
2. Choose among **X Axis** , **Y Axis**  and **Z Axis**  (in the **X, Y and Z Coordinate System**).
3. Or choose among **North Axis** , **East Axis**  and **Elevation Axis**  (in the **North, East and Elevation Coordinate System**).

Fitting With a Geometry

To Fit With a Geometry:

1. Click on the **Fit** pull-down arrow.
2. Choose one of the following:
 - Select **Fit Plane** . The **Fitting Tool** toolbar appears as well as an information window at the top right corner of the **3D View**.
 - Select **Find Best Cross Plane** . The **Fitting Tool** toolbar appears as well as an information window at the top right corner of the **3D View**.

Fitting a Plane

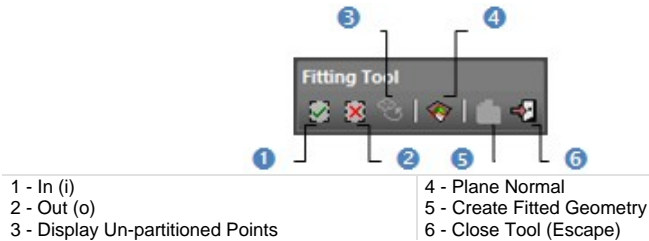


To Fit a Plane:

1. Fence a set of points for which you want to fit with a plane.
2. Click the **Plane** icon. Kept points are fitted with a plane.

Note: It is not necessary to fence a set of points; fitting a plane can be applied to the entire point cloud.

Finding the Best Cross Plane



To Find the Best Cross Plane:


1. Fence a set of points with which you want to fit a plane.
2. Click the **Plane Normal** icon. Kept points give the projection plane's normal direction.

Note: It is not necessary to fence a set of points; finding the best cross plane can be applied to the entire point cloud.

Picking an Object Local Frame

This method consists of picking an object's local frame. The initial plane will be moved so that its **Normal** will be parallel to the picked local frame. Its position in the 3D scene and its **Offset** (altitude) will be set by the picked point.

To Pick an Object Local Frame:


1. Click the **Pick Axis from Object**  icon. The initial cutting plane disappears from the **3D View**.
2. In the **WorkSpace** window, click on the **Models** tab.
3. Right-click on the selection to display the pop-up menu.
4. Select **Display Geometry**.
5. Click an object.

Note: Pressing **Esc** while the picking is in progress will cancel the selected point(s) and makes appeared the last defined plane.

Picking Two Points

This method consists of picking two points. The initial plane will be moved so that it will pass through the two picked points and perpendicular to the screen.

To Pick Two Points:


1. Click the **Plane Perpendicular to Screen**  icon.
2. Pick two points. No need to pick on the displayed object.

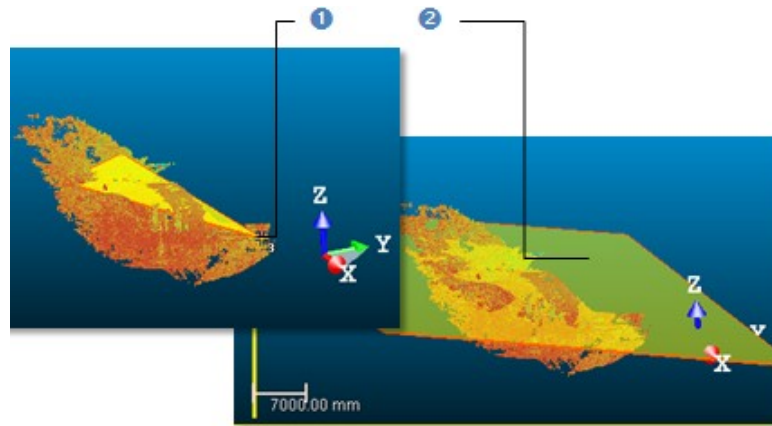
Note: Pressing **Esc** while the picking is in progress will cancel the selected point(s) and makes appeared the last defined plane.

Picking Three Points

This method consists of picking three points. The initial plane will be moved so that it will pass through the three picked points. Its position in the 3D scene and its **Offset** (altitude) will be the barycentre of the three picked points.

To Pick Three Points:

1. Click the **Pick 3 Points on Plane**  icon. The **Picking Parameters** toolbar appears.
2. Pick three points (free or constrained). Picking is always on the displayed object.




1 - Picked points

2 - The cutting plane passes through the picked points

Note: Pressing **Esc** while the picking is in progress will cancel the selected point(s) and makes appeared the last defined plane.

Editing Parameters

To Edit Parameters:

1. Click the **Edit Parameters**  icon. The **3D Plane Editing** dialog opens.
2. Click on the pull down arrow and do one of the following:
 - Chose **Normal + Point** to define a normal and a position.
 - a) Enter a direction in the **Normal** field.
 - b) Enter a point position in the **Point** field.
 - Chose **Point to Point** to define two points. **Points(To) - Point (From)** defines a normal and **Point (From)** gives a position.
 - a) Enter a point position in the **Point (To)** field.
 - b) Enter a point position in the **Point (From)** field.
3. Click **OK**. The **3D Plane Editing** dialog closes.

Plane Parallel to the Screen View


To Set a Plane Parallel to the Screen:

- Click **Plane Parallel to Screen View** .

Point

Once the initial plane is well oriented, you have to define its position in the 3D space.

To Define a Position:

1. Enter a 3D position in the **Point** field.
2. Or first click on the **Pick Point**  icon. The **Picking Parameters** toolbar appears in 3D constraint mode.
3. And then, pick a point in the 3D scene.
 - The initial plane passes through that point.
 - The **Offset** value is set to 0.00.



The initial plane in the **3D View** is hidden and the **Volume Calculation** dialog appears in grey. This means that the options and commands from this dialog are unavailable. The dialog comes back to its initial state when you have picked a point.

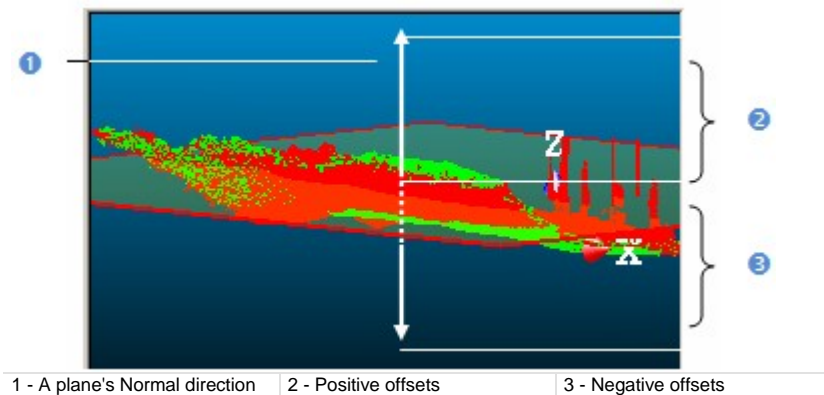
Note: The picking must be done on the selected surface (point cloud or mesh).

Offset

Once the initial plane's position has been defined, you have to set its position along its normal. By default, its current position corresponds to **Offset 0**. Setting a positive value will move the plane **Up** along its normal while a negative value will move it **Down**.

To Define an Offset Value:

1. Enter a value in the **Offset** field.
2. Or use the **Up**  and **Down**  buttons to select a value.



Defining a Grid Resolution

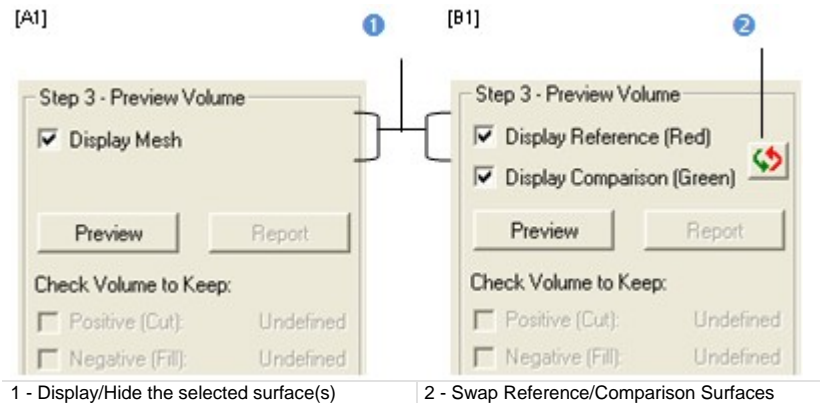
This step consists of defining a grid resolution which is square - the same in both of the defined plane directions (**Length** and **Width**).

To Define a Grid Resolution:

1. Enter a value in the **Resolution** field.
2. Or use the **Up**  and **Down**  buttons to select a value.

Previewing a Volume

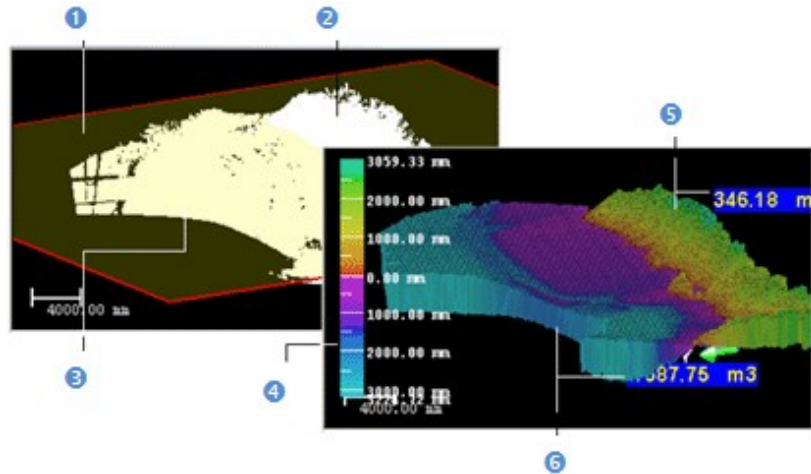
You can perform a preview to visualize the result before saving it in the database. You can change the parameters as many times as you please and perform a preview without leaving this tool. There are two display options (one per surface). If one surface has been selected as input, only one display option is available. See (A1). If two surfaces have been selected as input, the two display options are all available. You can reverse this comparison order by clicking on the **Swap Reference/Comparison Surfaces** icon. The **Reference Surface** becomes a surface to inspect and its color swaps to green. The **Comparison Surface** becomes a **Reference Surface** and its color turns to red. See (B1).



To Preview the Computed Volume(s):

1. Click **Swap Reference/Comparison Surfaces** if required.
2. Click on the **Preview** button.

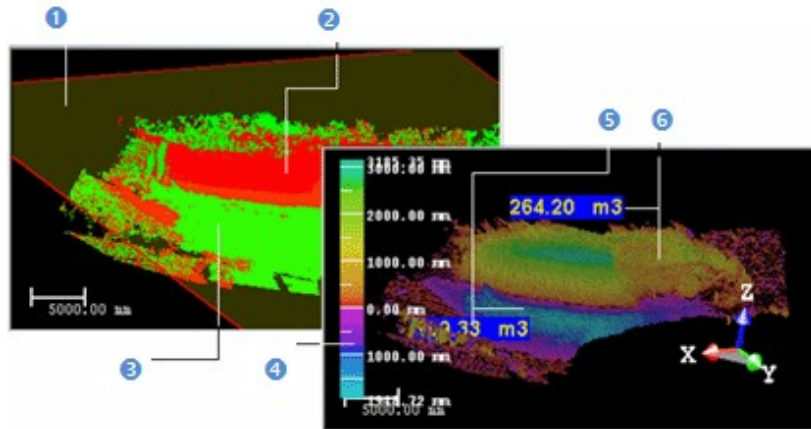
If one surface (point cloud or mesh) has been selected, the volume computation is done between this surface and the plane defined in **Step 1**. Two volumes are computed if the plane cuts the selected surface in two. The **Positive (Cut)** volume is the part (of the selected surface) above the plane while the **Negative (Fill)** volume is the part below. The sum of both is the volume computation result. A unique volume is computed if the plane does not cut the selected surface.



- 1 - The projection plane
 2 - Part of the surface above the plane
 3 - Part of the surface below the plane

- 4 - The Color scale
 5 - Positive part of the volume
 6 - Negative part of the volume

If two surfaces (point cloud(s) or mesh(es)) have been selected, the volume computation is done between both of them.



- 1 - The projection plane
 2 - The Reference Surface in red
 3 - The Comparison Surface in green

- 4 - The color scale
 5 - Negative part of the volume
 6 - Positive part of the volume

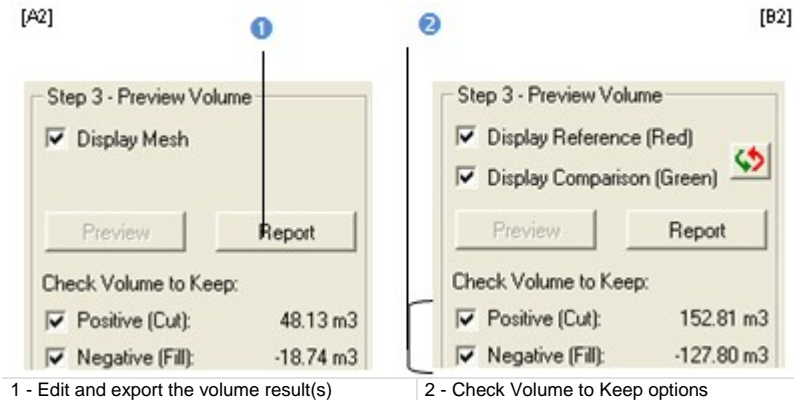
A volume is represented in the **3D View** by a graph of vertical color lines. You can estimate the height of each point (if a point cloud is selected) or of vertex (if a mesh is selected) compared to the defined plane using the graduated color scale on the left of the **3D View**.

Notes:

- The **Swap Reference/Comparison Surfaces** icon is not present when a single surface has been selected.
- After clicking **Preview**, if the projection resolution set in **Step 1** is too small; the following message **#Volume size is very high; Computation may take a long time. Do you want to continue?#** appears. Close the message and set a bigger projection resolution.

Checking a Volume to Keep

Once a preview has been performed, the **Preview** button becomes disabled and the **Check Volume to Keep** options become enabled. See (A2) and (B2).



To Check a Volume to Keep:

1. Clear the **Positive (Cut)** option. The positive representation of the volume is hidden.
2. Or clear the **Negative (Fill)** option. The negative representation of the volume is hidden.

Note: You cannot have the **Positive (Cut)** and **Negative (Fill)** options both unchecked. You need to have at least one checked.

Saving a Volume in a Report

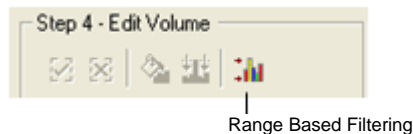
Once you are satisfied with the previewed volume(s), you can create a report and export to RTF format.

To Save a Volume in a Report:

1. Click **Report**. The **Volume Calculation Report** dialog opens.
2. Click **Export**. The **Export Volume Calculation Report** dialog opens.
3. Enter a name in the **File Name** field.
4. Specify a drive/folder where to store the file.
5. Click **Save**. The report is opened as an rtf file in the Microsoft Word application.
6. Print the report as required.

Editing a Volume

The volume previously computed may have irregularities like holes or peaks, you can then edit it by keeping (or removing) the part you want (or do not want), completing holes, smoothing or filtering according to two given elevations. The cursor is in the segmentation mode and only the **Range Based Filtering** feature is available. This means that it is up to the user to filter the computed volume or to fence an area for editing.

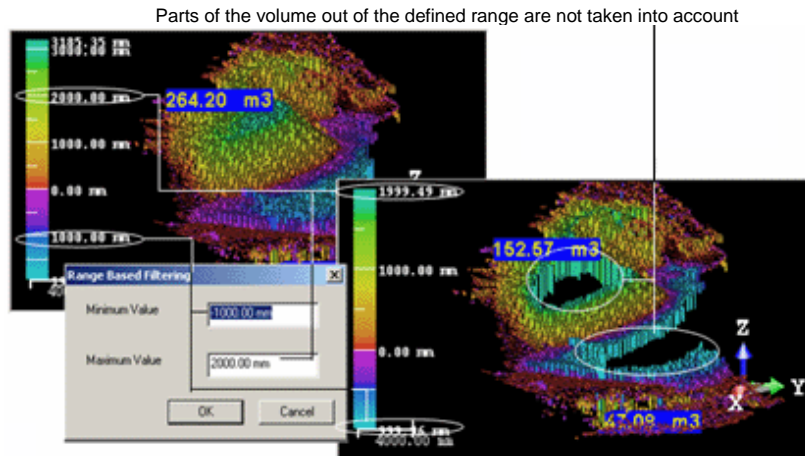


Filtering From an Elevation Range

You can filter the computed volume according a range of elevation values. Note that this filter cannot be applied to a part of the volume.

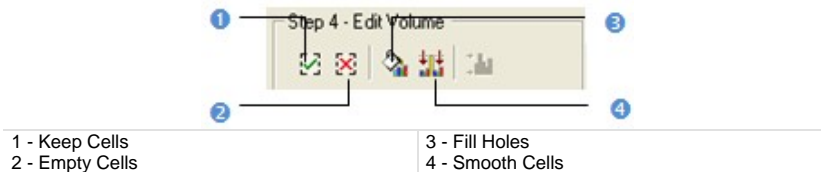
To Filter From an Elevation Range:

1. Click the **Range Based Filtering** icon. The **Range Based Filtering** dialog opens.
2. Enter a value in the **Minimum Value** field.
3. Enter a value in the **Maximum Value** field.
4. Click **OK**.



Fencing an Area

If you start by fencing an area on the volume, the **Fill Holes** and **Smooth Cells** icons become enabled and the **Range Based Filtering** icon swaps from enabled to disabled.



To Fence an Area:

1. Fence an area on the volume.
2. Right-click anywhere in the **3D View**.
3. Select **End Fence** from the pop-up menu.
4. Click the **Keep Cells** icon.
5. Or click the **Empty Cells** icon.

Tips:

- You can also right-click in the **3D View** to display the pop-up menu and select the command you want to use.
- Instead of selecting **Keep Cells** (or **Empty Cells**), you can also use the related short-cut key **I** (or **O**).
- Instead of selecting **End Fence** from the pop-up menu, you can also double-click.

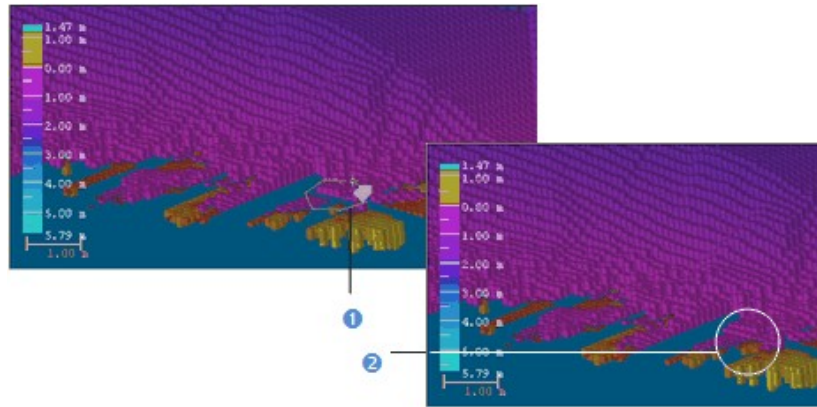
Notes:

- Once a fence has been drawn, the user can no longer manipulate the volume.
- To start a new fence, please cancel the current one by selecting **New Fence** from the pop-up menu or by pressing **Esc**.

Filling Holes

To Fill Holes:

1. Fence an area on the computed volume.
2. Click on the **Fill Holes** icon.



1 - A hole

2 - The hole is filled up

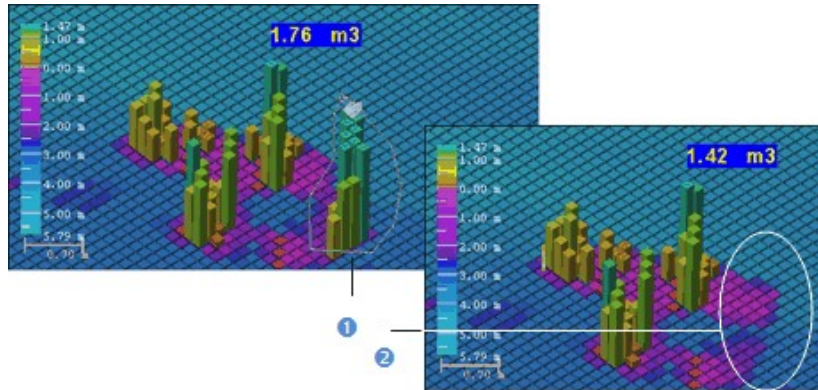
Tip: The **Fill Holes** icon can also be selected from the pop-up menu.

Note: The **Positive (Cut)** (or **Negative (Fill)**) value is then updated.

Smoothing Cells

To Smooth Cells:

1. Fence an area on the computed volume.
2. Click on the **Smooth Cells** icon.



1 - A set of peaks

2 - The set of peaks are smoothed down

Tip: The **Smooth Cells** icon can also be selected from the pop-up menu.

Note: The **Positive (Cut)** (or **Negative (Fill)**) value is then updated.

Saving a Volume in the Database

You can save the computed volume as a permanent object in the database, perform a screen printouts or export to the DXF file format. For each saved result, a volume object is created and is put under the **Active Group** in the **Models Tree**.

To Save a Volume in the Database:

1. Click **Create**.
2. Click **Close**.

Notes:

- **Close** can also be selected from the pop-up menu.
- Leaving the **Volume Calculation Tool** without saving the result will make appeared a warning message.


Profile/Cross-Section Tool

This tool is of particular use in civil engineering applications such as tunnel, bridge or road inspections. It is used for generating profiles and cross-sections from a point cloud (or from a set of point clouds) or from a mesh. A profile is a cut along a given polyline (also called **Path**). Cross sections are cuts performed perpendicularly to a given path.

Opening the Tool

If a point cloud has been selected, the step consists of working on it in order to delimit an area for the profile and cross section calculation, or to render the point cloud cleaner without parasite points or to simplify it. For these operations you can use two sub-tools: **Sampling** and **Segmentation**. Each of them, when used in such condition, prevents you from saving the result, the **Create** command is deactivated. If a mesh has been selected, the two upper sub-tools are unavailable.

To Open the Tool:

1. Select a point cloud (or mesh) from the **Project Tree**.
2. From the **OfficeSurvey** menu, select **Profile/Cross Section Tool** . The **Profile/Cross Section** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window and is composed of five parts. The first part contains two sub-tools: **Segmentation** and **Sampling**. The second part allows you to define the 2D cutting position. The third part is to set section parameters. The fourth part is assigned for calculating and displaying sections. The fifth and last part is to save the created sections in the database, close the tool and give access to the online help. The selected point cloud is displayed in white in the **3D View**, and the others are hidden.

Tip: You can also select a couple of point cloud (or mesh) and fitted polyline (or set of segments) as input of the **Profile/Cross Section Tool**.

Selecting a Method

There are two methods for computing a set of cross-sections: **From Segments** and **From Path**. If the loaded project does not contain any set of segments, the button **From Segments** stays dimmed. If there are some*, the button becomes enabled. The button **From Path** is always enabled even if there is no fitted polyline in the loaded project. Before selecting a method, the **No Path Selected** and **No Bounds** texts are displayed and the number of cross-sections is equal to 0.

To Select a Method:

- In the **Profile/Cross-Section** dialog, do one of the following:
 - Click the **From Segments** button. The **Cut Positions from Segments** dialog opens.
 - Click the **From Path** button. The **Cut Positions on Path** dialog opens.

Tips:

- If the input is a couple of point cloud (or mesh) and fitted polyline, the **No Bounds** text remains displayed. The fitted polyline name appears in the **Path** line. The number of cross-sections is calculated based on the default value of the **Interval** parameter.
- If the input is a couple of point cloud (or mesh) and set of segments, the **No Path Selected** and **No Bounds** texts remain displayed. The number of cross-sections is equal to the number of segments.

Note: (*) The user does not need to select them.

From Path

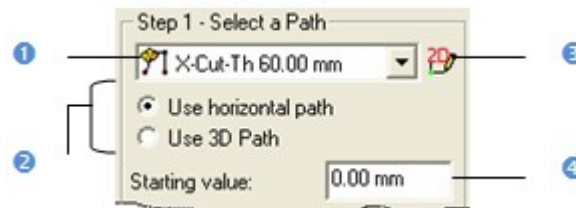
In this method, cross-sections are similar and regular along a given path. "Similar" means that all cross-sections are identical in terms of thickness and length. "Regular" means that all are equidistant i.e. the interval between two consecutive cross-sections is the same in all cases. A path can either be one that is in the project or one you draw. Paths are polylines.

Selecting

If there is at least one path (polyline) in the loaded project. You can select it for calculating the cross-sections. In that case, the selected point cloud (or mesh) and the current path (polyline) - the one listed in the selection box - with its projection (if existing) in the XY* plane are displayed in the **3D View**.

To Select a Path:

1. In the **Cut Positions on Path** dialog, click the pull down arrow.
2. Select a path (polyline) from the drop down list.



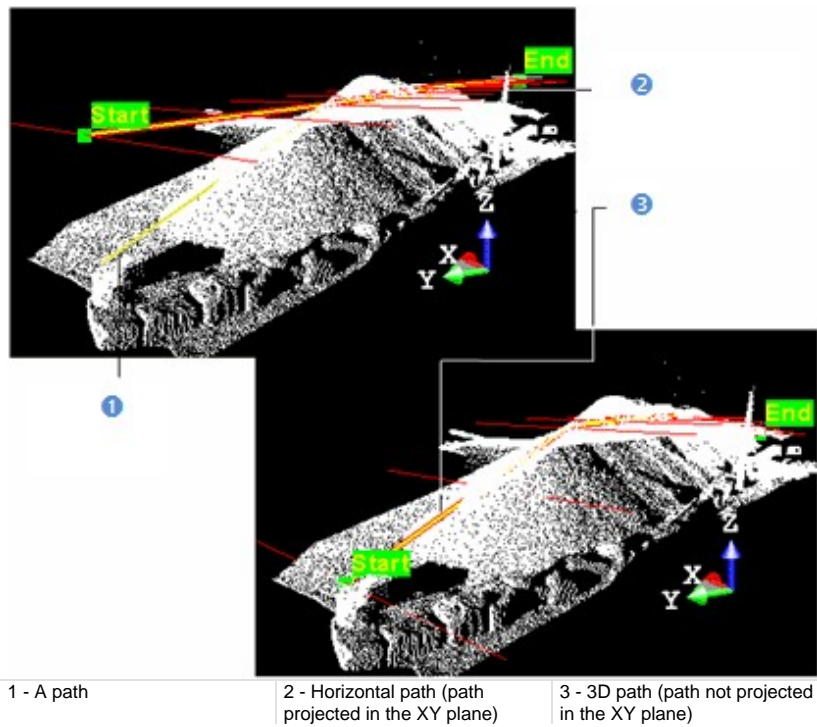
1 - Combo box for selecting an existing path

2 - Radio buttons for choosing between Horizontal Path or Use 3D Path

3 - Draw and Create Path in Database

4 - Field for setting the Starting value on the path

If the **Use Horizontal Path** option has been checked, cross-sections will be computed from the path projection in the XY plane. If the **Use 3D Path** option has been checked, cross-sections will be computed perpendicularly from the path in 3D (not projected in the XY plane). The **Starting** on a path is like its origin; its default-value is equal to zero but you can set it to a value that meets your need.



3. Enter a value in the **Starting** field.
4. Select a path.
5. Or draw a path.


Notes:

- The selected path (polyline) has to be regular (one chain with at least three points).
- (*) In the **X, Y, Z Coordinate System**.

Drawing

If any path (polyline) exists in your project, the combo box is grayed out. You have to create at least one in the database. In that case, only the selected scene (point cloud or mesh) is shown in the **3D View**. The scene is constrained in the XY^* plane of the active coordinate frame and movements while picking points are restricted to navigation movements. You can rotate the complete scene around the Z^* axis, zoom (in or out) along this same axis and pan in the XY^* plane.

To Draw a Path:

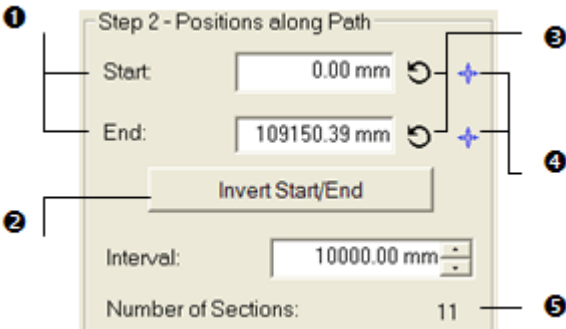
1. Click the **Draw and Create Path in Database**  icon. The **Drawing Tool** toolbar appears. The scene is locked in a 2D plane in the **Top** view with a 2D grid superimposed (if not hidden previously). The **Picking Parameters** toolbar appears in the 2D constraint mode. The mouse cursor changes to a pencil.
2. Pick at least two points (free or constrained).
3. Click **End Line**. The last picked point ends the line.
4. Or click **Close Line**. The start and end picked points are linked with a segment in order to form a closed line.
5. Click **Create**. The drawn line is saved and created in the database as a polyline.

Notes:

- If the 2D Grid had been hidden in a previous case, it will also be hidden when you activate the **Polyline Drawing Tool**.
- (*) In the **X, Y, Z Coordinate System**.

Start and End Positions

You need to define a portion from the selected or created path (polyline) - more exactly a portion from its projection on the XY* plane of the active coordinate frame (if **Use Horizontal Path** has been checked) or the path in 3D (if **Use 3D Path** has been checked) - for which you want to calculate cross sections. You need to define the **Start** and **End** points along the path (polyline). If the **Starting** value in **Step 1** is equal to zero; the **Start** and **End** points are set to the beginning and the end of the path (polyline). If this value is different from zero; the **Start** and **End** positions on the path are shifted of that value.



The screenshot shows the 'Step 2 - Positions along Path' dialog box. It contains the following fields and controls:

- Start:** A text input field with the value '0.00 mm'. A circular arrow icon is to its right.
- End:** A text input field with the value '109150.39 mm'. A circular arrow icon is to its right.
- Invert Start/End:** A button located below the End field.
- Interval:** A text input field with the value '10000.00 mm' and a spinner control to its right.
- Number of Sections:** A text input field with the value '11'.

Numbered callouts point to specific elements:

- 1** points to the 'Start' input field.
- 2** points to the 'Invert Start/End' button.
- 3** points to the circular arrow icon next to the 'Start' field.
- 4** points to the circular arrow icon next to the 'End' field.
- 5** points to the 'Number of Sections' input field.

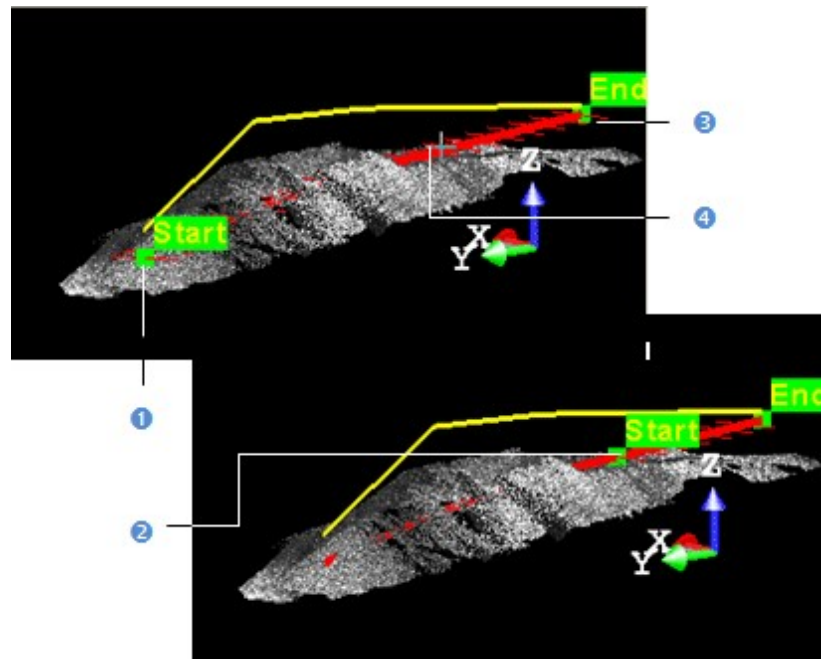
1 - Set the Start/End position by inputting a value	3 - Reload the initial Start/End position	4 - Pick the Start/End position
2 - Reverse the path direction		5 - An estimation of the number of sections

Defining

To Define the Start and End Positions:

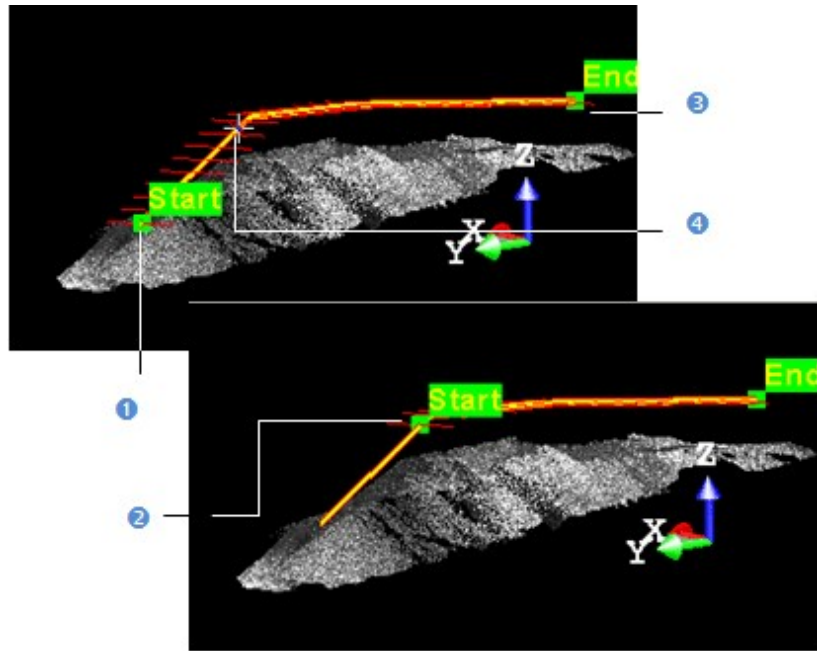
1. Enter a distance value in the **Start** field and press **Enter**.
2. Enter a distance value in the **End** field and press **Enter**.

Here below are two screen-captures showing the **Start** and **End** positions when the **Use Horizontal Path** option has been chosen.



- | | |
|--|--|
| 1 - The Start position on the horizontal (projected) path | 3 - The End position on the horizontal (projected) path |
| 2 - The Start position modified on the horizontal (projected) path | 4 - Pick to modify the Start position on the horizontal (projected) path |

Here below are two screen-captures showing the **Start** and **End** positions when the **Use 3D Path** option has been chosen.



1 - The Start position on the 3D path

2 - The Start position modified on the 3D path

3 - The End position on the 3D path

4 - Pick to modify the Start position on the 3D path

3. Click the **Reload Initial Start Position On Path** icon (if required).
4. Click the **Reload Initial End Position On Path** icon (if required).

Picking

To Pick the Start and End Positions:

1. Click the **Pick Start Position On Path** icon. The mouse cursor shape changes to a pointer.
2. Pick a point along the path (polyline). The picked point becomes the **Start** point.
3. Repeat the two above steps form the **End** position.
4. Click the **Reload Initial Start Position On Path** icon (if required).
5. Click the **Reload Initial End Position On Path** icon (if required).

Reversing

The **Start** and **End** positions gives a direction to the path. That's why the **Start** value must be positive and smaller than the **End** value. You can change the path's direction by reversing the **Start** and **End** positions.

Setting the Interval Parameter

Cross-sections will be calculated between the **Start** and **End** positions along the path (polyline). The distance between two consecutive cross-sections is defined in the **Interval** field. The estimated number of cross-sections is given in the dialog box. Each time you change the **Start** (or **End**) position or the **Interval** parameter, this number is updated.

To Set the Interval Parameter:

1. Enter a new value in the Interval field and press **Enter**.
2. Or use the **Up**  and **Down**  buttons to select a value.

Note: The given number of cross-sections is an estimation. If a cross-section contains no points, it will not be created in the database even if the **Create** command is selected.

Defining a Width

The **Interval** between two consecutive cross-sections is not enough to define them along the path (polyline). You also need to define their width by bounding them from each side. The left and right boundaries that delineate the width of each cross-section can be equal or different one from each other. The **Width** of a cross-section on a given position on the path (polyline) is the width of the point cloud.

To Define a Width:

1. Check the **Use Fixed Width** option. The **Left** and **Right** fields become editable.
2. Enter a new value in the **Left** field.
3. Enter a new value in the **Right** field.

Applying the Cut Positions

Once you have finished in defining the cut positions (path, positions on path, step length and cross section width) on the selected point cloud (or mesh), you can use the **Apply** button. Note that after leaving the **From Path** method, the name of the selected (or drawn) path (polyline) as well as the number of cross-sections are displayed in text in the **Profile/Cross-Section** dialog. If the **Use Fixed Width** option has been selected, the **With Bounds** text appears.

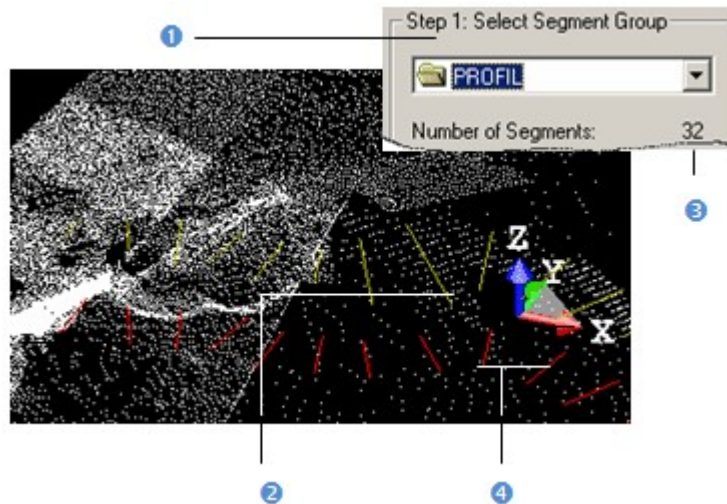
From Segments

Cross-sections that result from this method are all different and irregular along a given path. “Different” because they are not all identical in terms of thickness and length. “Irregular” because all are not equidistant. Such cross-sections can be obtained with a set of segments. Segments can come from a DXF (AutoCAD®) file that you import into your project. A path can either be one that is in the project or one you draw. Paths are polylines.

Selecting a Segment Group

To Select a Segment Group:

1. In the **Cut Positions from Segments** dialog, click on the pull-down button.
2. Select a group from the drop-down list.



1 - A combo box for selecting a group of segments

2 - The selected segments (in yellow)

3 - The number of segments in the selected group

4 - The selected segments projected in the XY plane (in red)

The number of segments in the selected group appears below the selection list. Each segment (in yellow) and its projection (in red) in the XY* plane of the active coordinate frame are shown in the **3D View**. Note that the segments in yellow are in 3D while those in red are in 2D.

Note: (*) In the X, Y, Z Coordinate System.

Defining a Path

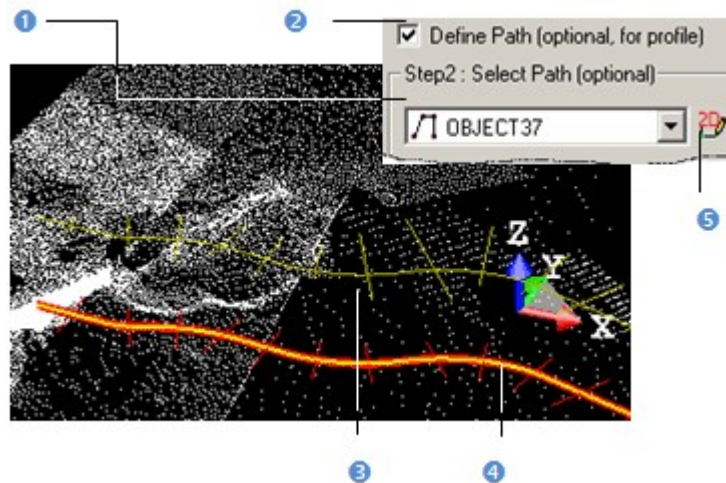
As in the **From Path** method, a path (if available in your project) allows you to generate a profile and to set the cross sectioning direction (**Start** and **End** positions). Note that a path is not necessary for generating cross-sections; that why this step (in the **From Segments** method) is optional. A path becomes necessary if you wish to order all cross-sections and to have the position of each of them along the profile (distance from the **Start** position to the current (active) cross-section). If no path exists in your project, you can use the **Draw and Create Path in Database** tool to draw one.

Tip: You can use a path that comes from the **From Path** method in the **From Segments** method, and conversely.

Selecting

To Select a Path:

1. Check the **Define Path** option. The **Select Path** field becomes active.
2. Click on the **Select Path** pull-down arrow.
3. Select a path (polyline). The path (polyline) representation appears in the **3D View**.



1 - A box for selecting an existing path
 2 - The Define Path option
 3 - A path


4 - The path projected in the XY plane
 5 - The Draw and Create Path in Database Tool

The selected path (in yellow) and its projection in the XY* plan (in red) are displayed in the **3D View** window.

Note: (*) In the X, Y, Z Coordinate System.

Drawing

To Define a Path:

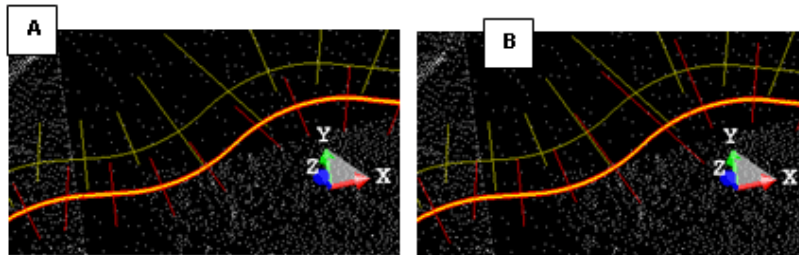
1. Click the **Draw and Create Path in Database**  icon. The **Drawing Tool** toolbar appears. The scene is locked in a 2D plane in the **Top** view with a 2D grid superimposed (if not hidden previously). The **Picking Parameters** toolbar appears in the 2D constraint mode.
2. Draw and create a path.

The drawn path (in yellow) and its projection in the XY* plan (in red) are displayed in the **3D View**.

Note: (*) In the X, Y, Z Coordinate System.

Using Segment Width

You can constrain the point cloud's sectioning with the segment's width by using the **Use Segment Width** option. In [A], this option is unused - each selected segment and its projection have differing widths. In [B], this option is used - each segment and its projection have identical widths.



Applying the Cut Positions

Once you have finished in defining the cut positions on the selected point cloud (or mesh), you can use the **Apply** button. After leaving the **From Segments** method, the number of cross-sections is displayed in text in the **Profile/Cross-Section** dialog box. If a path has been selected, its name is displayed. And if the **Use Segment's Width** option has been selected, **With Bounds** text appears.

Computing Cross-Sections

The **Compute Sections** panel remains dimmed if any selection of a set of segments (or of a fitted polyline) has been performed after

Setting a Thickness

The distance between two consecutive cross-sections defined in the previous step is not a sufficient parameter for computing the whole cross-sections. You must also define a value which will be used as a cutting thickness along the profile.

To Set a Thickness:

1. Enter a new value in the **Thickness** field.
2. Or use the **Up**  (or **Down** ) button to select a value.

Note: The **Thickness** field is enabled only if there is a fitted polyline (or set of segments) selected.

Setting a Tolerance

Tolerance is a parameter used for approximating the model (profile) to the reality (cloud). The smaller this parameter is, the closer the approximation will be. Tolerance zero means that the corresponding profile or cross-sections pass through all sectioned points.

To Set a Tolerance:

1. Enter a new value in the **Tolerance** field.
2. Or use the **Up**  (or **Down** ) button to select a value.

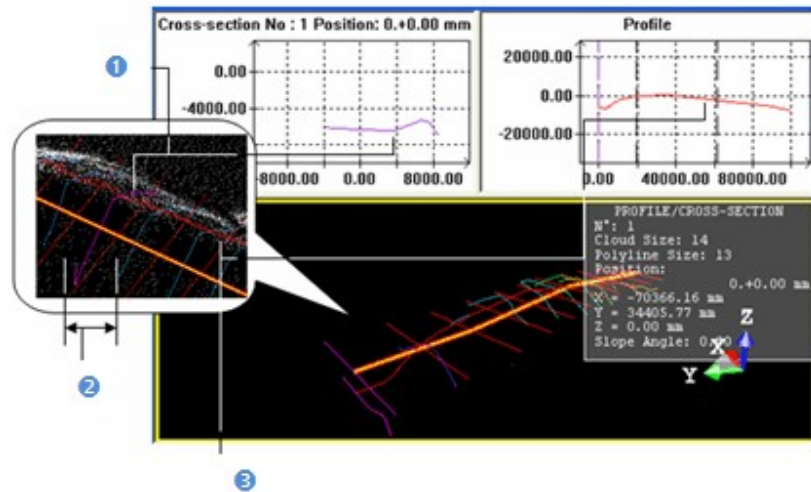
Note: The **Tolerance** field is enabled only if there is a fitted polyline (or set of segments) selected.

Previewing

You have to preview the results before saving in the **RealWorks** database or change the parameters and perform a new preview as many times as you please.

To Preview the Profile and Cross-Sections:

1. Click **Preview**. The **3D View** splits into three sub-windows.

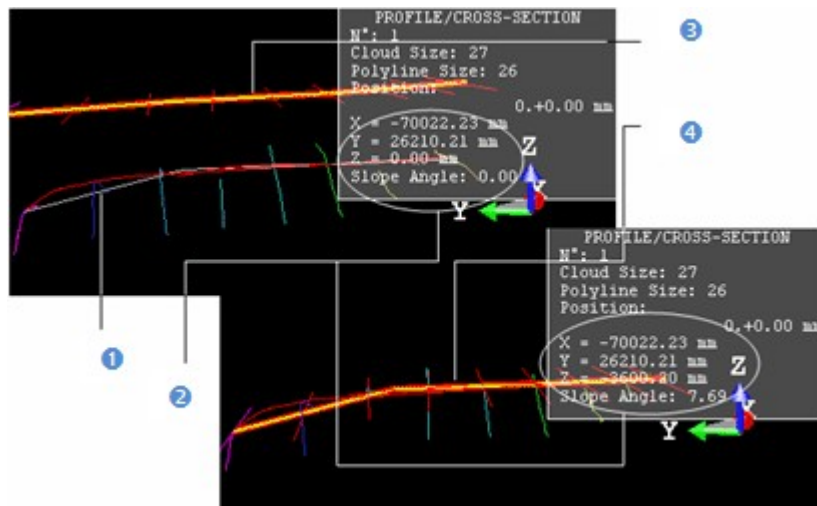


1 - The active cross-section (in fuchsia)
2 - The Step length

3 - The profile (in red)

The top left sub-window displays the active (or selected) cross-section as a linear graph. The sub-window in the top right corner displays the profile as a linear graph. The third sub-window displays all generated cross-sections with the path and the profile. The active cross-section is in fuchsia. You can organize these three sub-windows as you please using the commands of the toolbar at the bottom of the user interface.

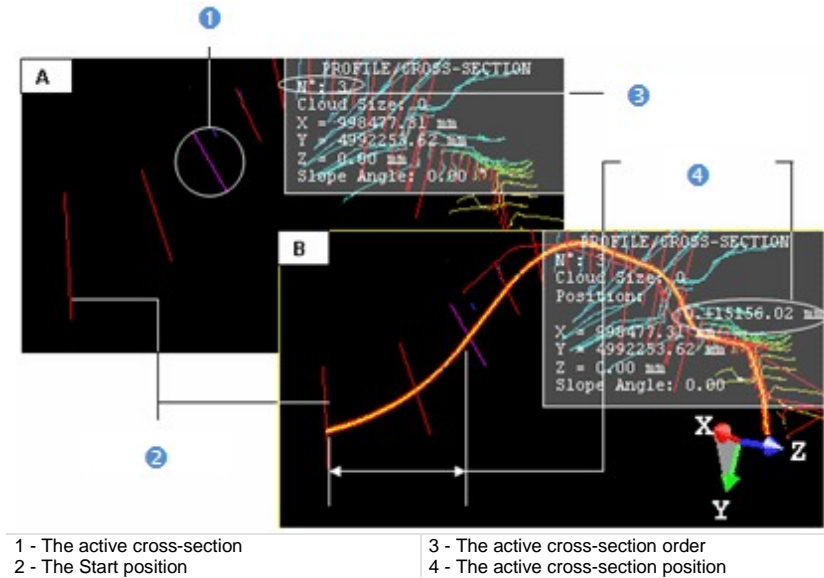
When selecting the **From Path** method, for a given cross-section, an information box located at the top right corner of the third sub-window lists information such as its order, the cloud size (number of points that it contains) if a point cloud has been selected, the polyline size, its distance from the **Start** position and its 3D position not on the path but on the path projected in the XY* plane if **Use Horizontal Path** has been checked and on the 3D path if **Use 3D Path** has been checked. The slope angle is obtained by intercepting the path (horizontal or 3D) with the horizontal plane. For this reason the Z coordinate and the **Slope** angle value are equal to 0 when computing cross-sections from a projected path.



1 - The selected path
2 - 3D coordinates of the current cross-section position on the horizontal (or 3D) path

3 - Horizontal path (path projected in the XY* plane)
4 - 3D path (path modeled from the path in selection)

When selecting the **From Segments** method - in (A) no path has been selected, the active (selected) cross-section's position is unknown while its order is known. In (B) a path has been selected, the same cross-section's position is in text in the information box and its order differs from the one in (A).



2. If a point cloud has been selected, un-check the **Display Cloud** option to remove the point cloud representation from the **3D View**, if required.
3. If a mesh has been selected, un-check the **Display Mesh** option to remove the mesh representation from the **3D View**, if required.

Note: (*) In the **X, Y, Z Coordinate System**.

Printing

To Print a Profile (or Cross-Sections):

1. Click inside a graph to select it.
2. From the **File** menu, select **Print** from the menu bar. The **Print Setup** dialog opens.
3. Set the print parameters (if required).
4. Click **OK**. The **Print Setup** dialog closes.

Tip: You can also right-click inside a graph for which you want to perform a print-out. The pop-up menu drops down. Then select **Print**.

Scaling

For a given linear graph, you can zoom it in/out, pan it or change its scale. Note that **Zooming In/Out** will change the linear graph scale. Changing a linear graph scale can be done by using the mouse wheel or by selecting the **Scales** command from the pop-up menu.

To Manipulate the Profile and Cross-Sections:

1. Click inside a graph to select it.
2. Do one of the following:
 - Drag and drop the graph (with the left button) to a new location to pan it.
 - Use the mouse scroll wheel to zoom in/out.
 - Or use the left and middle buttons.

Note: The **Zoom** is centered on the current mouse location.

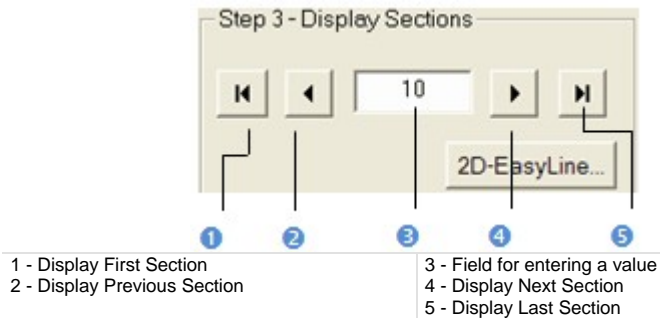
To Scale the Profile and Cross-Sections:

1. Click inside a graph to select it.
2. Right-click and select **Scales** from the pop-up menu. The **Plot Scale** dialog opens with the **Automatic Scaling** option checked by default.
3. Un-check the **Automatic Scaling** option to choose the manual scaling. Both the **Horizontal Scale** and the **Vertical Scale** fields become active.
4. Click on the **Horizontal Scale** (or **Vertical Scale**) pull-down arrow.
5. Choose a scale for the **Horizontal Scale** (or **Vertical Scale**) list.
6. Click **OK**. The **Plot Scale** dialog closes.

Displaying Cross-Sections

To Display Cross-Sections:

1. Use the buttons illustrated below to navigate through the cross-sections.
2. Or pick a cross-section in the **3D View**.
3. Or enter a cross-section order in the field and press **Enter**.



- If the selected cross-section is empty (see [A]), the **2D-EasyLine** button remains grayed out.
- If the selected cross-section contains points and fitted polyline (see [B]), the **2D-EasyLine** button becomes enabled.

[A]	[B]
<pre> PROFILE/CROSS-SECTION N°: 6 Cloud Size: 0 Position: 0.00+33.01 m X = 987.09 m Y = 4978.81 m Z = 0.00 m Slope Angle: 0.00 ° </pre>	<pre> PROFILE/CROSS-SECTION N°: 7 Cloud Size: 17 Polyline Size: 16 Position: 0.00+38.47 m X = 982.15 m Y = 4976.51 m Z = 0.00 m Slope Angle: 0.00 ° </pre>

Tip: Instead of clicking **Display Next Section** or (**Display Previous Section**), you can also use the **Up** (or **Down**) key on your keyboard.

Note: The **Display Sections** panel becomes enabled only if a preview of cross-sections has been performed.

Editing Cross-Sections

You can select and edit a cross-section using the **2D-EasyLine Tool**. The selected cross-section needs to have points inside and fit with a polyline.

Creating Profile and Cross-Sections

When saving the result in the database, a group named **Cross-Sec-Thick "Thickness value"-Tol "Tolerance value"** is created and rooted under the **Models Tree**. This group contains the calculated cross sections and the selected (or drawn) path.

These results are fitted polylines and can be exported via DXF/DGN formats to AutoCAD® and MicroStation®.

To Save the Profile and Cross-Sections:

1. Click **Create**.
2. Click **Close**.

Notes:

- **Close** can also be selected from the pop-up menu.
- Instead of selecting **Close**, click inside a sub-view and press **Esc**.
- Leaving the **Profile/Cross-Section Tool** without leaving the results will make appeared a warning message.


Feature Set Tool

This tool allows you to collect a set of surveying points/chains from a scanned point cloud in a way that simulates regular surveying methods. The use of this tool consists of two parts: one is for defining (or importing) the feature code; another is for collecting surveying points/chains and attaching them to the selected feature code.

Opening the Tool

If the loaded project contains no **Feature Code Library**, a default library named **LIB_1** will be created. You can rename or delete it.

To Open the Tool:

1. Select a project from the **Project Tree**.
2. From the **OfficeSurvey** menu, select **Feature Set Tool** . The **Feature Set Creation** dialog opens as well as the **Picking Parameters** toolbar.

The **Feature Set Creation** dialog opens as the third tab of the **WorkSpace** window and is composed of three parts. The first part is for collecting points. The second part is to define feature code. The third and last part allows you to save the results, close the tool and obtain access to the online help.


Feature Code Libraries

You can add a new **Feature Code Library** in your current project by creating one or by importing one that originates from surveying applications. If the new library is empty of feature codes, a warning message appears and it will be deleted. A new library is always named **LIB** plus **X** corresponding to its order of creation.

Adding

You can add a new **Feature Code Library** in your current project by creating one. If the new library is empty of feature codes, a warning message appears and it will be deleted. A new library is always named **FEATURE_CODE_LIBRARY** plus **X** corresponding to its order of creation.


To Add a Feature Code Library:

1. Click the **Edit Feature Code Library** . The **Feature Code Library Editing** dialog opens.
2. Click **New**. A new feature code library is added.
3. Click **OK**.

Importing


You can import a **Feature Code Library** that originates from surveying applications.

To Import a Feature Code Library:

1. Click the **Edit Feature Code Library**  icon. The **Feature Code Library Editing** dialog opens.
2. Click **Import**. The **Import Feature Code Library** dialog box opens.
3. Navigate through your hard disk to locate the library to be imported and select it.
4. Click **Open**.

Renaming

To Rename a Feature Code Library:

1. Click the **Edit Feature Code Library**  icon. The **Feature Code Library Editing** box opens.
2. Click **Rename**. The feature code library name becomes editable.
3. Enter a new name and press **Enter**.
4. Click **OK**.

Deleting

If the loaded project contains a **Feature Code Library** that you don't want anymore; you can delete it. If the **Feature Code Library** you delete is the last in your project, a warning message appears and a default **Feature Code Library** is created.

To Delete a Feature Code Library:

1. Click the **Edit Feature Code Library**  icon. The **Feature Code Library Editing** box opens.
2. Drop down the Library list and select a feature code library.
3. Click **Delete**. The selected feature code library is deleted.
4. Click **OK**.

Exporting

A library once filled with **Feature Codes** can be exported to a TXT format file so that it can be used for other **RealWorks** projects.

To Export a Feature Code Library:

1. Click the **Edit Feature Code Library**  icon. The **Feature Code Library Editing** box opens.
2. Click **Export**. The **Export Feature Code Library** opens.
3. Enter a new name in the **File Name** field or keep the default one.
4. Specify a location on your hard disk in which to store the library in the **Look In** field.
5. Click **Save**.

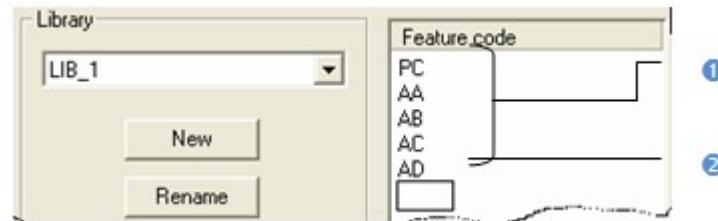
Feature Codes

A **Feature Code** is a system for codifying **Feature Points**. A **Feature Code** should reflect the **Feature Points** you intend to collect.

Defining

To Define a Feature Code:

1. Click **Edit Feature Code Library**. The **Feature Code Library Editing** dialog opens.



1 - Already defined feature codes

2 - Field for defining feature codes

2. Select a **Feature Code Library** from the list.
3. Or use the default one (no selection is required).
4. Click in the **Feature Code** panel below the PC **Feature Code**.
5. Define a new **Feature Code**.
6. Press **Enter** on your keyboard.
7. Click **Apply**.

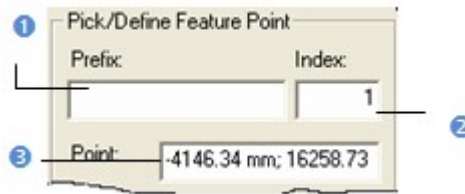
Tip: Instead of defining **Feature Codes**, you can import into your project a library which already contains a set of **Feature Codes**.

Deleting

Deleting a library will similarly delete the **Feature Codes** (if available) that are inside.

Feature Points

A **Feature Point** is composed of attributes like its name (**Prefix** + **Index**), its 3D coordinates, its **Feature Code** and optionally, a **Description**. The user can either pick or define a **Feature Point**.



1 - The Prefix of a Feature Point

2 - The coordinates of a Feature Point

3 - The order of a Feature Point

Picking

A **Feature Point** can be collected either by free (or constrained) picking a point. In that case, the mouse cursor's shape changes to a pencil.

To Pick a Feature Point:

1. Pick a **Feature Point** on the displayed scene in the **3D View**. Its 3D coordinates are displayed in the **Point** field.
2. If required, enter a name in the **Prefix** field.
3. Click on the **Feature Code Library** pull down arrow.
4. Select a library from the **Feature Code Library** list.
5. Click on the **Feature Code** pull down arrow.
6. Select a code from the **Feature Code** list.
7. Continue in collecting other **Feature Points**.

Note: An empty point will be not taken into account.

Editing

You can edit a **Feature Point** previously defined. Just pick on it in the **3D View**. Its features are displayed in the **Prefix**, **Index** and **Point** fields. You can then modify them as you please.

Modifying

You can delete the last collected **Feature Point** by undoing the operation or by removing it from your selection using the **Delete Point** command. You can insert or add a **Feature Point** to your collection once the selection is completed.

Deleting

To Delete a Feature Point:

1. Pick on an already picked point in the **3D View**.
2. Right-click in order to display the pop-up menu.
3. Select **Delete Point** from the pop-up menu.

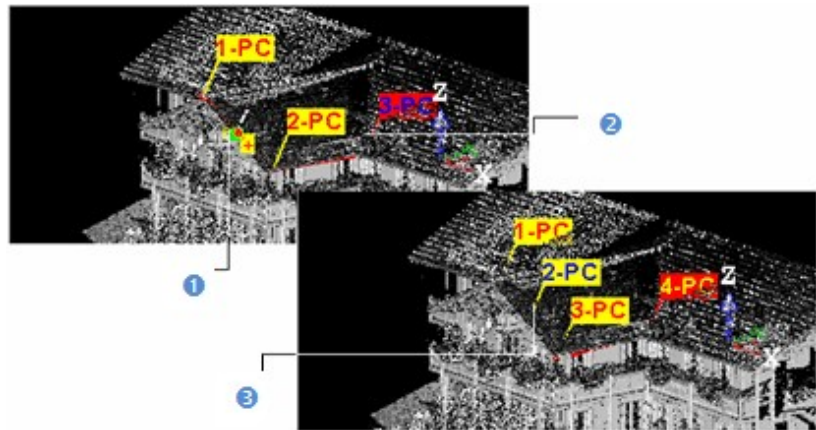
Note: Instead of selecting **Delete Point** from the pop-up menu, you can also use the **Del** key on your keyboard.

Inserting

You can insert a **Feature Point** between two **Feature Points** only if they are linked by a **Continuous** (or **Dash-Line**) **Segment**.

To Insert a Feature Point:

1. Place the cursor over a position between two **Feature Points**.
2. Click to insert a **Feature Point** at this position.



1 - The cursor's shape when placing it between two connected feature points

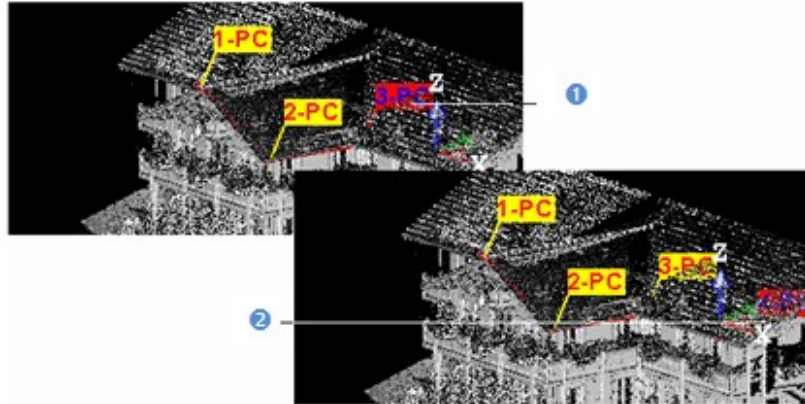
2 - The last collected feature point has a red label

3 - The inserted feature point

Adding

To Add a Feature Point:

1. Place the cursor over a point on the displayed object. An empty point is not taken into account.
2. Click to add a **Feature Point** at this position.



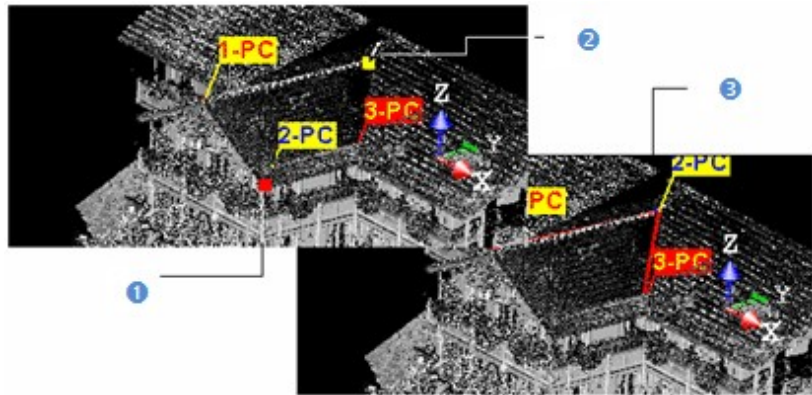
1 - The last collected Feature Point has a red label

2 - A Feature Point has been added at this position

Moving

To Move a Feature Point:

1. Place the cursor over a **Feature Point**. A green square appears.
2. Drag the **Feature Point** from its current position. The green square becomes red and remains over the selected **Feature Point**'s position.
3. And drop the **Feature Point** to a new position. A yellow square appears under the cursor while moving it.



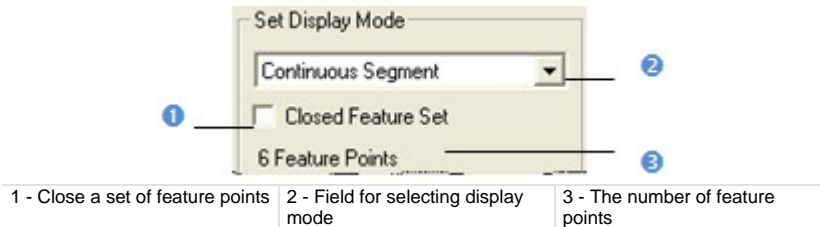
1 - The selected Feature Point current position

2 - The cursor' shape while dragging and dropping the selected Feature Point

3 - The selected Feature Point new position

Setting Display Mode

A **Feature Point** when collected may have two statuses: isolated points simply called points or connected points called chains. If points are connected, you can close the chain by checking the **Closed Feature Set** option.



To Set a Display Mode:

1. Click on the **Set Display Mode** pull down arrow.
2. Do one of the following:
 - Select **Points Only** to set collected points as isolated points.
 - Select **Continuous Segment** to connect collected points with plain line segments.
 - Select **Dash-Line Segment** to connect collected points with dotted line segments.

Creating Feature Sets

A **Feature Set** is created in the database for each collection of **Feature Points** saved in the database. The user can create as many **Feature Sets** as required without leaving the tool. A **Feature Set** has **OBJECTX** as name and is rooted in the **Models Tree**. You can export a **Feature Set** to an **ASCII** format file.

To Create a Feature Set:

1. Click **Create**.
2. Click **Close**.

Note: You can also right-click anywhere in the **3D View** to display the pop-up menu and select **Close**.


2D-Polyline Inspection Tool

This tool enables to compare a model to a point cloud. A model can come from an imported DXF (or DWG) file or results from cross-sectioning. An imported model must be already georeferenced with the point cloud. In both cases, a model is a 2D polyline which should be composed of at least three non-aligned points.

Opening the Tool

If your project you load has no 2D polyline, to activate the **2D-Polyline Inspection Tool** you need to import or create at least one. Otherwise, an error message will appear. If your project contains more than one 2D polyline, you can launch the tool and select one for inspection. A 2D polyline should be formed by at least three non-aligned points.

To Open the Tool:

1. Select one point cloud (or more) from the **Project Tree**.
2. From the **File** menu, select **Open**. The **Open** dialog opens.
3. Select **AutoCAD® files (*.dxf, *.dwg)** from the **File Of Type** field.
4. Select the **Add To Project** option (if needed).
5. In the **Look In** field, navigate to the drive/folder where the .dxf file is located.
6. Double-click on the file to select it. The **DXF File Import** dialog opens.
7. Select the DXF length unit and click **OK**.
8. From the **OfficeSurvey** menu, select **2D-Polyline Inspection Tool** . The **2D-Polyline Inspection** dialog opens.

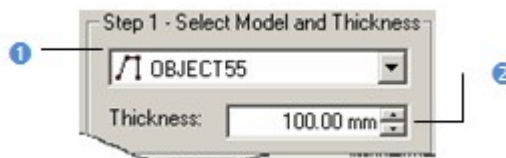
This dialog opens as the third tab of the **Workspace** window and is composed of four parts: The first part contains two sub-tools: **Segmentation** and **Sampling**. The second part allows you to select a model and to set a thickness. The third part is to perform the inspection. The fourth and last part is to save the inspection result in the **RealWorks** database, close the tool and obtain access to the online help. The selected cloud is displayed in white in the **3D View**.

Selecting a Model for Inspection

Inspecting in 2D means by plane. A virtual plane drawn by the non-aligned points of the selected model cuts the selected point cloud. The thickness you set corresponds to the thickness of this plane and it should be higher than 0. The 2D inspection is done by comparing the selected model to the points that are in this plane. All available models in your project are stored in the selection list in the **2D-Polyline Inspection** dialog, except those with less than three points and you can select one of them by dropping down this list. If you select a set of models before opening the **2D-Polyline Inspection Tool**, these models will be unselected except one which is the first in the selection list.

To Select a Model for Inspection:

1. Click on the **Select Model and Thickness** pull down arrow.





1 - Field for selecting a model

2 - Field for setting a thickness

2. Select a model from the drop down list.

Setting the Thickness Parameter

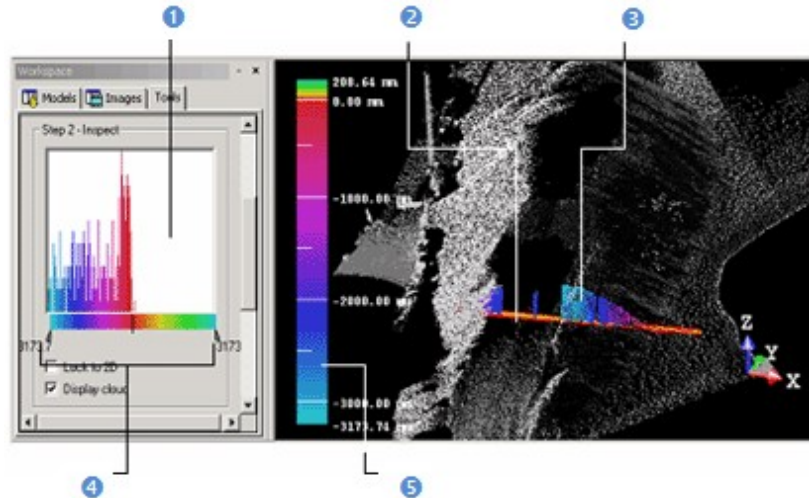
To Set a Thickness:

1. Enter a value in the **Thickness** field.
2. Press **Enter** on your keyboard.
3. Or use the **Up**  and **Down**  buttons to select a value.

Viewing the 2D Inspection Results

The 2D inspection is automatically performed once a model is selected, even if you have not set a thickness. **RealWorks** will set a thickness by default. The inspection result can be viewed in the **3D View** window and in a graph at the bottom of the **2D-Polyline Inspection** dialog (see A). The selected (active) model and points resulting from the inspection are displayed in the **3D View**. Each distance, which separates a point from the model, is shown with a color (see B). Two options are available: you can lock the result to 2D and display or hide the selected point cloud. A color scale at the left side of the **3D View** gives an idea of distance for a given color.

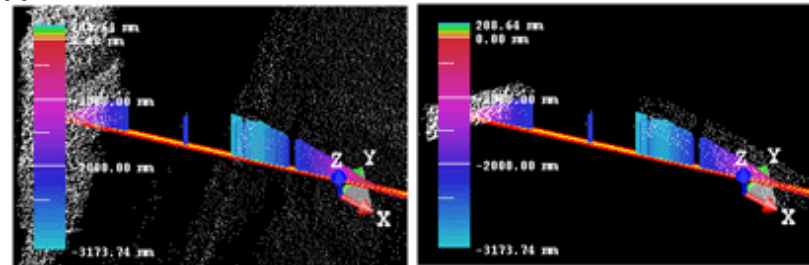
[A]



- 1 - The inspection graph
- 2 - The selected model
- 3 - The inspection between the model and the cloud at this position

- 4 - Sliders for filtering points based on a given distance
- 5 - The color scale giving the distance information

[B]



In this picture, the selected point cloud, the selected model and the inspection result are displayed in the 3D View window.

In this picture, only points resulting from the inspection with the selected model are displayed

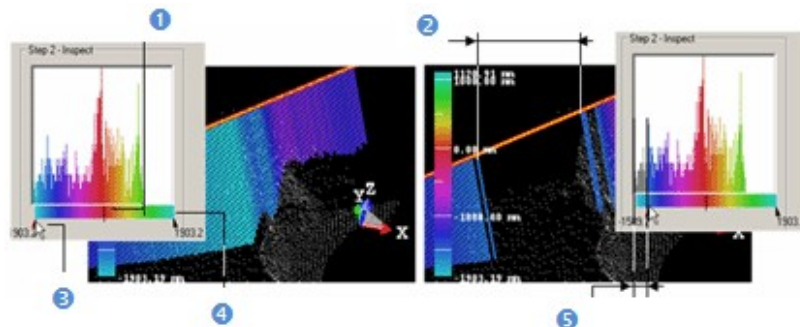
Note: In the 2D locked position, you can display the 2D grid. To do this, you can either right-click anywhere in the **3D View** to display the pop-up menu and select **Show 2D Grid** or select the **2D Grid/Show 2D Grid** command from the **3D View** menu bar.

Filtering the 2D Inspection Results

Two sliders (one at each end of the inspection graph) allow you to filter points according to a given distance. The **Right** end corresponds to the maximum distance from the model while the **Left** end corresponds to the minimum distance from the model. Filtering is done by moving one of the two sliders.

To Filter the 2D Inspection Results:

1. Place your cursor over a slider. It becomes red.
2. Drag the cursor from its current position (maximum or minimum distance).
3. Drop the cursor until it comes closed to the distance you attempt to reach. The filtering will be done between this range of distances.



- 1 - The model's position
- 2 - The filtering result
- 3 - The minimum distance from the model

- 4 - The maximum distance from the model
- 5 - The filtering range

Note: In the 2D locked position or not, you can print the inspection result. To do this, you can either right-click anywhere in the **3D View** to display the pop-up menu and select **Print** or select the **Print** command from the **File** menu bar.

Saving the 2D Inspection Results

The inspection results are summarized in the **Step 3** of the **2D-Polyline Inspection** dialog. You may find four major information: the maximum distance from the model, the minimum distance from the model, the positive area and the negative area. If more than one 2D polyline are available in your project, you can select a new one and set a new thickness; the inspection is automatically updated. Once the inspection is done, you can the result in **RealWorks** database.

To Save the 2D Inspection Results:

1. Click **Save**.
2. Click **Close**.

Note: Instead of clicking **Close** in the dialog, you can also right-click anywhere in the **3D View** to display the pop-up menu and then select **Close** or press twice **Esc**.


Twin Surface Inspection Tool

The **Twin Surface Inspection Tool** enables to compare two surfaces together. These surfaces should be similar as much as possible or not so different one from the other. You can compare together two point clouds, a point cloud and a mesh or two meshes. The surface inspection is based on a grid method and the result is an **Inspection Map**. You can choose in the **Preferences** dialog the units required to represent this map.

Opening the Tool

You should select two surfaces from a project in order to be able to activate the **Twin Surface Inspection Tool**. The first selected surface will be a reference surface (called **Reference**) and the second selected surface will be a surface to inspect (called **Comparison**).

To Open the Tool:

1. Select two surfaces from the **Project Tree**.
2. In the **OfficeSurvey** menu, select **Twin Surface Inspection Tool** . The **Twin Surface Inspection** dialog opens

This dialog opens as the third tab of the **WorkSpace** window and is composed of six parts. The first part contains two sub-tools: **Segmentation** and **Sampling**. If point clouds have been selected, you can use these two sub-tools to delineate an area for the inspection, to render them cleaner (i.e. reduce parasite points) or to simplify them. The second part allows you to define a projection surface. The third part allows you to set the projection surface's resolution. The fourth part allows you to preview the inspection result, display (or hide) the selected point cloud(s)/mesh(es). The fifth part is to edit the inspection result. The sixth part is to save the inspection result, close the tool and give access to the online help.

Before opening the **Twin Surface Inspection Tool**, each selected surface is shown with its own color in the **3D View**. After you open the tool, the reference surface (the first selected) will appear in red and the second surface (the second selected) in green. If the selection contains a point cloud and a mesh as surfaces; the mesh will be by default a reference surface (in red) and will not be able to be changed to a comparison surface (in green). The information box (at the top right corner of the **3D View**) lists the name of the reference surface and of the comparison surface*.

Tip: You can also click **Twin Surface Inspection Tool** in the **Tools** toolbar.

Note: If the **Project Cloud** has been selected as input, it is by default the **Reference Surface**. Its name does not appear in the information box.

Defining a Projection

A projection surface can be a 3D Plane, a 3D Cylinder or a 3D Tunnel.

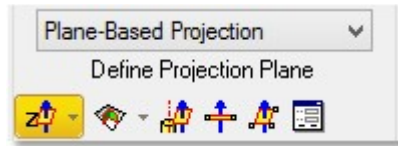
To Define a Projection Surface:

1. In the **Twin Surface Inspection** dialog, click the pull-down arrow.
2. Choose among **Plane-Based Projection**, **Cylinder-Based Projection** (on page 679) and **Tunnel-Based Projection**.

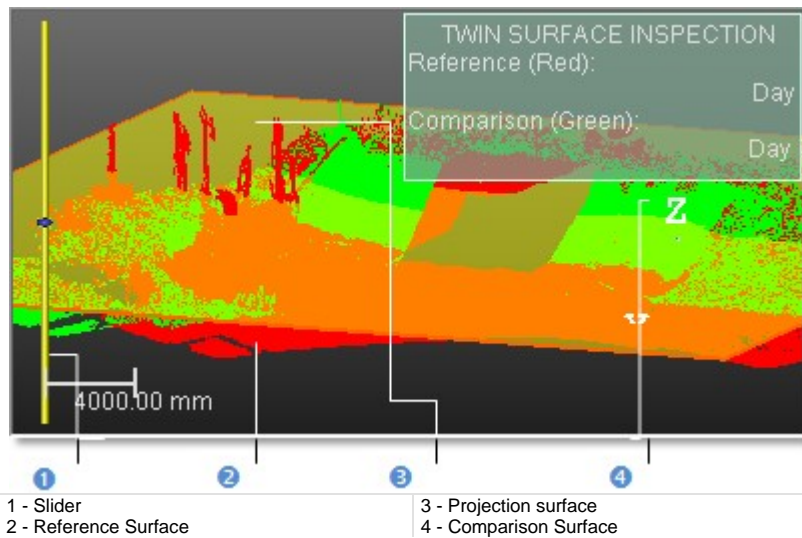
Note: The projection type which comes first is the one you have selected during the last use of that tool.

Plane-Based Projection

If the two selected surfaces are of plane shape, choose **Plane-Based Projection**. Step1 of the **Twin Surface Inspection** dialog becomes as shown below.



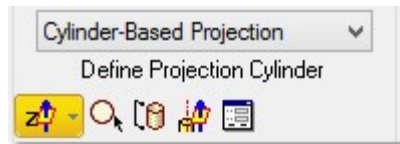
A projection plane with a normal direction parallel to the Z axis* of the active coordinate frame appears in the **3D View**. A slider at the left side of the **3D View** allows you to move the projection plane from the top to the bottom and vice versa. The way of defining a 3D plane is the same as in the **Cutting Plane Tool**.



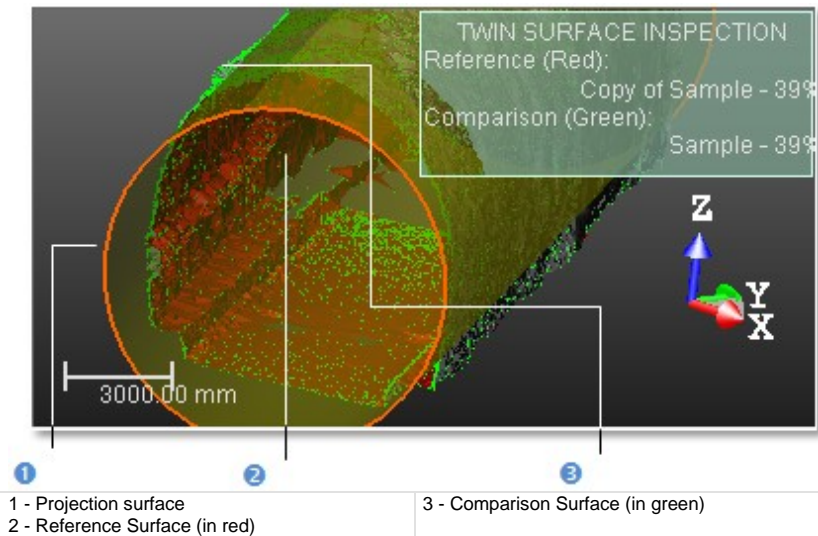
Note: (*) In the X, Y and Z Coordinate System.

Cylinder-Based Projection

If the two selected surfaces are of cylinder shape, choose **Cylinder-Based Projection**. **Step1** of the **Twin Surface Inspection** dialog becomes as shown below.









A projection cylinder with an axis parallel to the Z axis* of the active coordinate frame appears in the **3D View**. You can change the projection cylinder axis direction according to the two other axes (X* and Y*). If the selection (only point cloud) that you performed in **Step 1** contains an entity, you can pick on it so that its axis becomes the new axis of the projection cylinder. You can also pick points on your selection (point cloud or mesh) to define a projection cylinder or edit one manually. In all cases, the bounding box that highlights the selection (point cloud or mesh) delineates the height of the projection cylinder. You cannot exceed it.



Note: (*) In the X, Y and Z Coordinate System.


Setting From Frame

To Define a Projection Cylinder by Selecting a Frame Axis:

1. Click on the **Set from Frame** pull down arrow.
2. Choose among **X Axis** , **Y Axis**  and **Z Axis**  (in the **X, Y and Z Coordinate System**),
3. Or choose among **North Axis** , **East Axis**  and **Elevation Axis**  (in the **North, East and Elevation Coordinate System**).

Drawing Circle


To Pick Two Points to Define a Projection Cylinder:

1. Click **Draw Circle** . The **Picking Parameters** toolbar appears in 3D constraint mode below the **3D View** window.
2. Pick one free (or constrained) point. After picking this point, the **Picking Parameters** toolbar switches to the **Cartesian System** constraint.
3. It's up to you to use this constraint or to pick a free point.

Note: Picking can be done anywhere on the selection (point cloud or mesh) or not. These two points determine the projection cylinder's diameter and its direction is perpendicular to the screen view.

Drawing Cylinder


To pick three points to define a projection cylinder:

1. Click **Draw Cylinder** . The **Picking Parameters** toolbar appears in the 3D constrain mode.
2. Pick three points (free or constrained).

Note: Picking should be on the selection (point cloud or mesh) for the two first points and anywhere for the third point (on selection or not). The first and second picked points give the projection cylinder's direction and the second and third picked points determine its diameter.


Picking Axis from Object

To Pick an Axis from an Object:

1. Click the **Pick Axis from Object**  icon. The initial projection disappears from the **3D View**.
2. Click on the **Models** tab.
3. Right-click on the selection to display the pop-up menu.
4. Select **Display Geometry**.
5. Click one point.

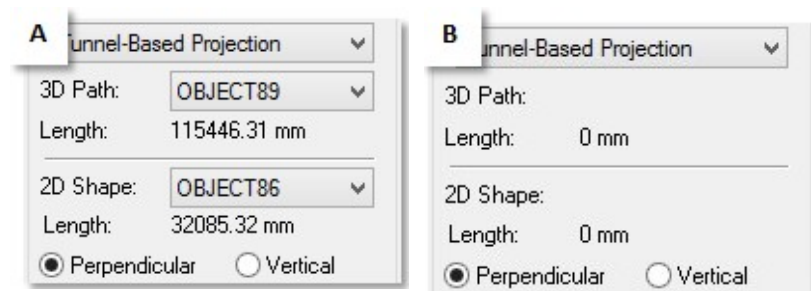
Editing Parameters

To Edit Manually a Cylinder:

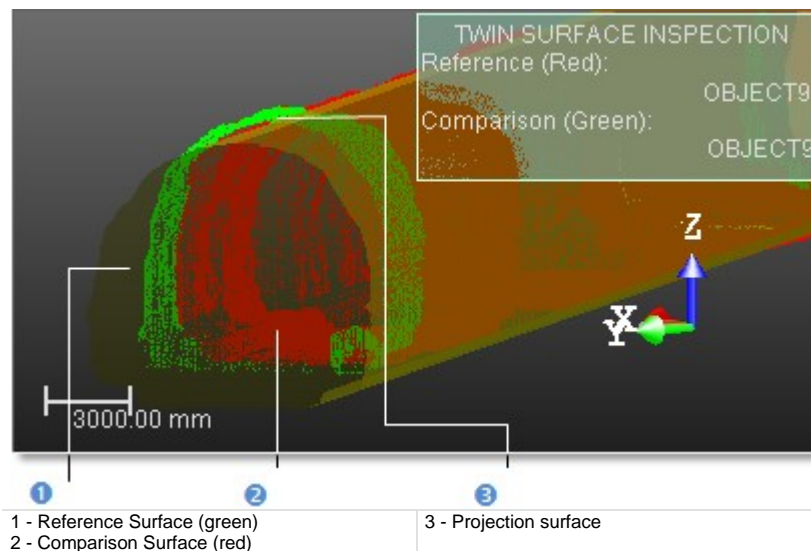
1. Click **Edit Parameters** . The **Cylinder Editing** dialog opens.
2. Click the pull down arrow and do one of the following:
 - Define a projection cylinder with two points and a radius.
 - a) Select **2 Points + Radius**.
 - b) Enter a point's position in the **Point1** field.
 - c) Enter another point's position in the **Point2** field.
 - d) Enter a value in the **Radius** fields.
 - Define a projection cylinder with one point, on direction and a radius.
 - a) Select **Point + Direction + Radius**.
 - b) Enter a point position the **Point** field.
 - c) Define a direction in the **Direction** field.
 - d) Enter a value in **Radius** field.
3. Click **OK**. The **Cylinder Editing** dialog closes.

Tunnel-Based Projection

If the two selected surfaces are of tunnel shape, choose **Tunnel-Based Projection**. If your project contains a **3D Path** and a **2D Shape** (respectively a 3D polyline and a 2D polyline), the name of each with the length information is displayed. If there is more than one path or shape, you can drop-down the selection list for each and select another **3D Path** or **2D Shape**. See [A]. If your project contains no **3D Path** and no **2D Shape**; the length of both the **3D Path** and the **2D Shape** is equal to 0. See [B].



A projection tunnel of the 3D polyline's length and of the 2D polyline's shape appears with the reference and comparison surfaces respectively in red and green.



You can inspect the **Comparison Surface** (in green) with the **Reference Surface** (in red) along or perpendicular to the 3D Path by checking **Perpendicular** or along the 3D path and in the Z axis* direction (of the active coordinate) by checking **Vertical**.

Note: (*) In the **X, Y and Z Coordinate System**.

Determining a Resolution

The same resolution will apply to both directions and will differ according to the projection surface you have defined in **Step 2**. If the projection surface is a 3D plane, the resolution directions will be the width and length directions of that plane. If the projection surface is a 3D cylinder, the resolution directions will be the axis direction of this cylinder and the direction of its circumference when you unfold it. If the projection surface is a 3D tunnel, the resolution directions will be the **3D Path** and **2D Shape** directions.

In Plane/Cylinder-Based Projection

To Determine a Resolution in Plane/Cylinder-Based Projection:

1. Enter a value in the **Resolution** field.
2. Or use the **Up**  and **Down**  buttons to select a value.

In Tunnel-Based Projection


To Determine a Resolution in Tunnel-Based Projection:

1. Enter a value in the **Along 3D Path** field.
2. Enter a value in the **Along 2D Shape** field.
3. Or use the **Up**  and **Down**  buttons to select a value.

Previewing the Inspections

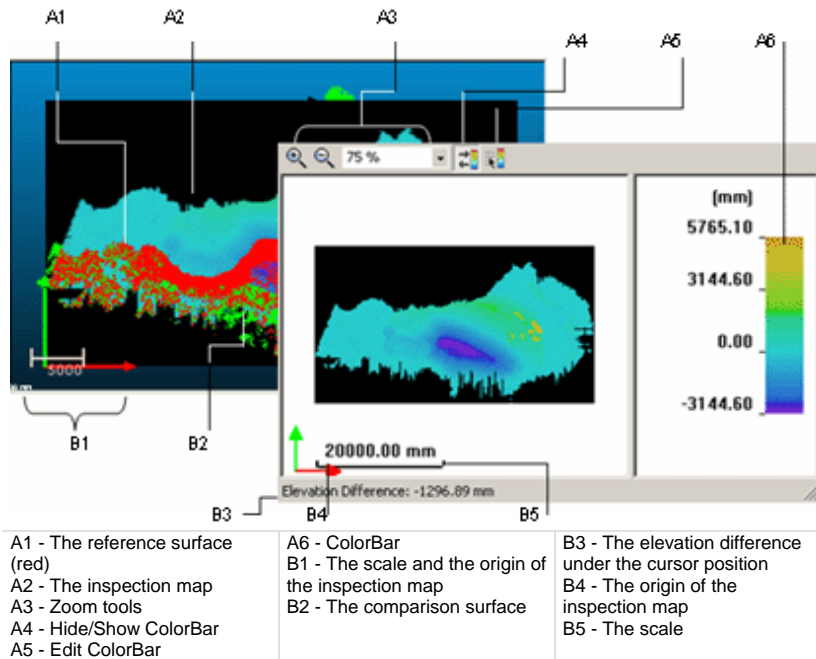
Once you have finished defining the projection surface and setting its resolution, you can preview the result before saving it in the database. You can change the parameters (projection surface and resolution) as many times as you please and perform a preview without leaving this tool. By default, an object selected for activating the tool is automatically displayed in the **3D View**. To hide it, you have to un-check the display option. And to display it again, you should re-check the option. There are two display options (one for each selected object).

To Preview the Inspection:

1. Un-check the **Display Reference (Red)** option, if needed.
2. Un-check the **Display Comparative (Green)** option, if needed.
3. Click the **Preview**.
 - If the two surfaces are of the same type (**Point Cloud** (or **Mesh**) for both), the **Swap Surfaces**  icon is available. Click the **Swap Surfaces** icon. The **Reference Surface** becomes a surface to inspect (**Comparison**) and the **Comparison Surface** swaps for **Reference**.
 - If the two surfaces are from different type (**Point cloud** for one and **Mesh** for the other), the **Swap Surfaces** icon is then unavailable
4. Click again the **Preview**.

Inspection Maps

The inspection result is a map shown in a specific window (called **Map Preview**) which opens beside the **3D View**. This map is a 2D image inside which each pixel is colored according to the difference (expressed in terms of elevation) between the two surfaces. A **ColorBar** located at the right side of the inspection map is a scale of elevation values and each color corresponds to a range of elevation values.



In the **Map Preview** window, you can zoom the inspection map **In** or **Out**. You can do this in three ways. The first one is to magnify (or reduce) an area of the inspection map using **Image Zoom In** and **Image Zoom Out** . The second way is to magnify (or reduce) the inspection map using the mouse wheel (if present). The last way is to select a rate from the drop-down list. If the inspection map is larger than the **Map Preview** window can show, you can pan it on left-click in four directions: **Up**, **Down**, **Right** and **Left**. In the **3D View**, the inspection map is shown in superposition with the two selected surfaces and the projection surface. A frame (red and green) corresponding to the inspection map's origin appears in both the **3D View** and the **Map Preview** window.

Printing

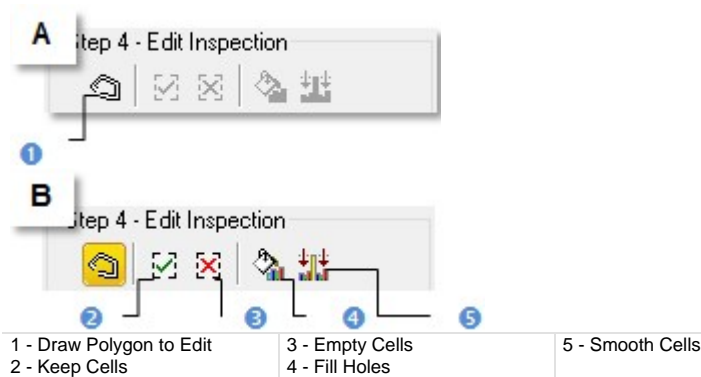
To print an inspection map, click inside the **Map Preview** window and do one of the following:

To Print an Inspection Map:

1. Click **Print** in the **Main** toolbar,
2. Or select **Print** from the **File** menu
3. Or select **Print** from the pop-up menu.

Editing

The inspection map computed previously may have irregularities like holes or spades, you can then edit it in order to keep or to remove the part you want, to complete the holes or to smooth the map. In edition mode, the mouse cursor will change its shape and picking is always inside the inspection map. Before you draw a polygonal fence, only the **Draw Polygon To Edit** tool can be used (see [A]). After you draw and validate a polygonal fence, the other tools become active (see [B]).



Fencing an Area

To Fence an Area:

1. Click the **Draw Polygon to Edit**.
2. Fence an area on the inspection map.
3. Right-click in the **3D View** window.
4. Select **End Fence** from the pop-up menu.

Notes:

- To cancel the current polygonal fence, you can press **Esc** or select **New Fence** from the pop-up menu.
- To leave the **Draw Polygon to Edit** tool, click again the **Draw Polygon to Edit**, select **Close Polygon Tool** from the pop-up menu or press **Esc**.

Tips:

- Instead of selecting **End Fence** from the pop-up menu, you can either double-click or press on the **Space Bar**.
- You can also select **Keep Cells** (or **Empty Cells**) from the pop-up menu or use their related short-cut key **I** (or **O**).

Filtering an Area

To Filter an Area:

1. Click the **Fill Holes**, if the fenced area contains holes.
2. Or click **Smooth Cells**, if the fenced area contains spikes for example.

Notes:

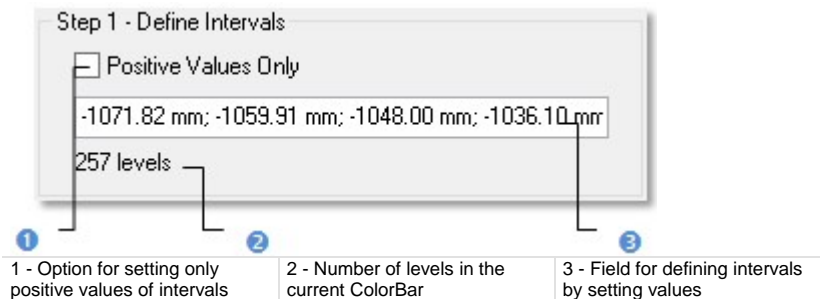
- You can also select **Fill Holes** (or **Smooth Cells**) from the pop-up menu or use their related short-cut key **F** (or **S**).
- An inspection map, once created, becomes un-editable.

Filtering the Inspections

Do one of the following if you wish to filter the inspection map between a set of intervals: edit the current **ColorBar** and change its interval values to those you need to use for filtering; or create a new **ColorBar** and set the interval values required for filtering. The elevation differences which are out of this range of intervals will disappear from the inspection map.

To Filter the Inspection Results:

1. Click **Edit ColorBar**. The **ColorBar** dialog opens.
2. Click **Edit**. The **ColorBar Editing** dialog opens.



3. Click in the interval value field and key in new values. The number of levels is displayed below.
4. Select the **Positive Values** option (if required).
5. Click in an interval line. A pull down arrow appears.
6. Click on the pull down arrow. A color palette appears.
7. Choose an existing color or click **Other** to define yours.
8. Click **Save**.

Checking the Inspections

The map gives you a global vision of the inspection between both selected surfaces. You can move your cursor over a point on the inspection map in order to have the difference of elevations between these two surfaces at this point. The difference of elevations appears in text beneath the **Map Inspection** window.

Saving the Inspections

In this last step, you can save the inspection result as a permanent object in the database, perform a screen printout or go further in analyzing the result thanks to the **Inspection Map Analyzer Tool**. For each inspection map you save, a geometric object is created and is put under the active group in the **Models Tree**. The **ColorBar(s)** that is (or are) related to it will be saved too. As any geometric object in **RealWorks**, you can display its representation in the **3D View** by double-clicking it in the **Project Tree** or by selecting the **Open Inspection Map** command from the **Display** menu. A **ColorBar** has no representation in the **Project Tree**.

Notes:

- To leave the **Twin Surface Inspection Tool**, you can press **Esc** or select **Close** from the pop-up menu.
- Leaving the **Twin Surface Inspection Tool** without saving the inspection result will open an information box which prompts you to confirm, undo or cancel the operation you attempt to execute.


Surface to Model Inspection Tool

This tool allows you to compare a surface with a model of plane, cylinder or tunnel shape. The surface and the model should be fairly similar in the sense that the tool is designed to be used in "before and after" situations. You can select a point cloud or a mesh for comparison. The surface inspection is based on a grid method and the result is an **Inspection Map**. You can choose in the **Preferences** dialog box the units required to represent this map.

Opening the Tool

You need to select a surface (point cloud or mesh) and a model from a project in order to be able to open the **Surface to Model Inspection Tool**. A model (when of plane and cylinder shape) can now be created using e.g. the **Geometry Creator Tool** in the **Modeling** processing mode. A model of tunnel shape is computed from a **2D Shape** and a **3D Path** within the **Surface to Model Inspection Tool**; both are polylines respectively in 2D and 3D. These polylines can be created using e.g. the **Polyline Drawing Tool**. You can also import a model either from 3Dipsos (a Trimble software application) or by loading a DXF/DWG format file. The selected model will be a reference surface and the point cloud (or mesh) will be a surface to inspect.

To Open the Tool:

1. Select one of these two sets (surface and model of plane/cylinder shape or surface and 2D polyline and 3D polyline) from the **Project Tree**.
2. From the **OfficeSurvey** menu, select **Surface to Model Inspection Tool** . The **Surface to Model Inspection** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window and is composed of five parts. The first part contains two sub-tools: **Segmentation** and **Sampling**. The second part allows you to set a projection surface (3D plane, 3D cylinder or 3D tunnel) resolution. The third part allows previewing the inspection result, display (or hide) the selected point cloud/mesh or model. The fourth part is to edit or to analyze the inspection result. The fifth part is to save the inspection result, close the tool and give access to the online help.

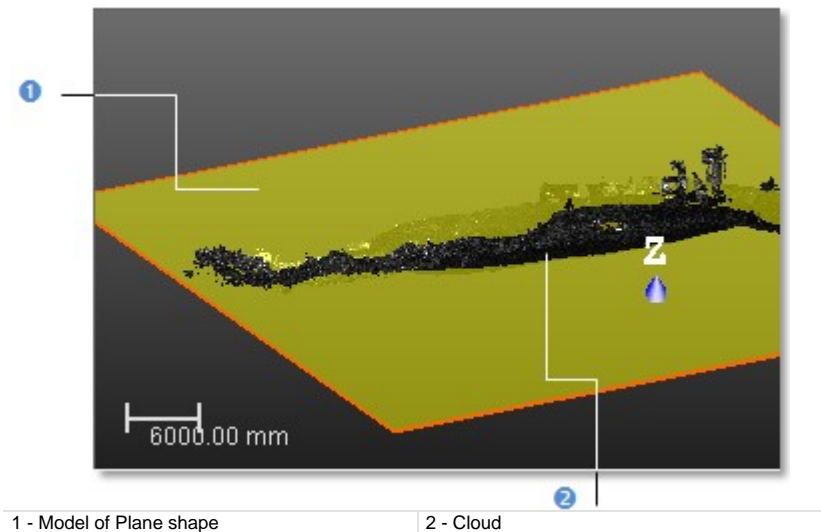
Tip: You can also click **Surface to Model Inspection Tool** in the **Tools** toolbar.

Defining a Projection

A projection model will be automatically set after opening the **Surface to Model Inspection Tool**. The surface to inspect (point cloud or mesh) remains with its own color and the reference surface (model) is shown in yellow.

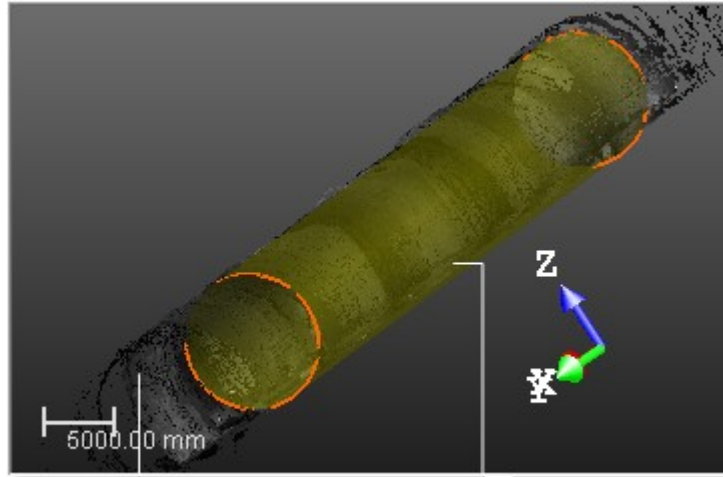
Plane-Based Projection

If a Model of plane shape has been selected, the projection is based on a 3D plane of the same direction (**Normal**) and whose dimensions (**Width** and **Length**) are delineated by the highlighting box.



Cylinder-Based Projection

If a model of cylinder shape has been selected, the projection is based on a 3D cylinder of the same size (**Diameter** and **Length**).

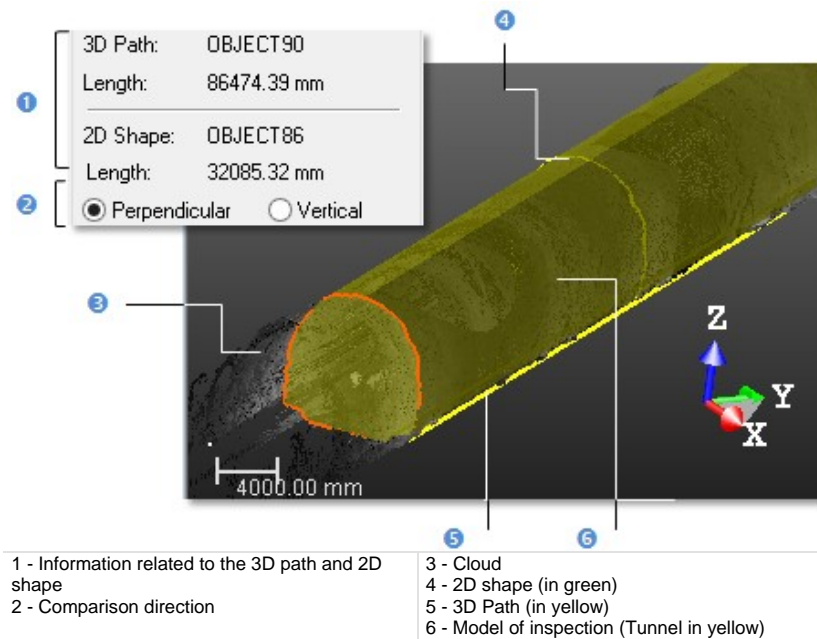


1 - Point cloud

2 - Model of Cylinder shape

Tunnel-Based Projection

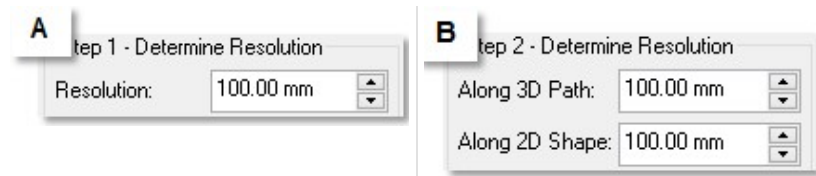
If a **2D Shape** and **3D Path** (respectively a 2D polyline and a 3D polyline) have been selected, the projection is based on a **3D Tunnel** of the 3D polyline's length and of the 2D polyline's shape. The **2D Shape** and **3D Path** name with the length information are displayed in **Step 1**. You can inspect the **Comparison Surface** (point cloud or mesh) with the **Reference Surface** (model) along or perpendicular to the **3D Path** by checking **Perpendicular** or along the **3D path** and in the Z axis direction (of the active coordinate) by checking **Vertical**.



Note: The step **Define Projection** is only available when selecting a 2D shape and a 3D path.

Determining a Resolution

A same resolution will be applied to both directions of the projection surface except when a **2D Shape** and a **3D Path** have been selected - see [A] and [B]. These directions will differ according to the projection model set in **Step 1**. If the projection model is a **3D Cylinder**, the projection directions are equal to the axis direction of the **3D Cylinder** and to the direction along the **3D Cylinder's** circumference. If the projection model is a **3D Plane**, the projection directions correspond to the width and length directions of the **3D Plane**. If the projection surface is a **3D Tunnel**, the resolution directions will be the **3D Path** and **2D Shape** directions.



In Plane/Cylinder-Based Projection

To Determine a Resolution in the Plane/Cylinder-Based Projection:

1. Enter a value in the **Resolution** field.
2. Or use the **Up**  and **Down**  buttons to select a value.

In Tunnel-Based Projection

To Determine a Resolution in the Tunnel-Based Projection:

1. Enter a value in the **Along 3D Path** field.
2. Enter a value in the **Along 2D Shape** field.
3. Or use the **Up**  and **Down**  buttons to select a value.

Previewing the Inspections

Once you have finished defining the projection surface resolution, you can perform a preview to visualize the result before saving it in the **RealWorks** database. You can change the parameter as many times as you please and perform a preview without leaving this tool. By default, an object selected for activating the tool is automatically displayed in the **3D View**. To hide it, you have to un-check the display option. And to display it again, you should re-check the option. There are two display options (one for each object selected). If the selected object is a point cloud, the display option will take the name of **Display Cloud**. If the selected object is a mesh, the display option will take the name of **Display Mesh**. The second display option's name is always **Display Model**.



To Preview the Inspection:

1. Un-check the **Display Cloud** (or **Display Mesh**) option, if needed.
2. Un-check the **Display Model** option, if needed.
3. Click **Preview**.

Note: You do not necessary need to validate the value you key in the **Resolution** field. Clicking the **Preview** button will validate the value by its own.

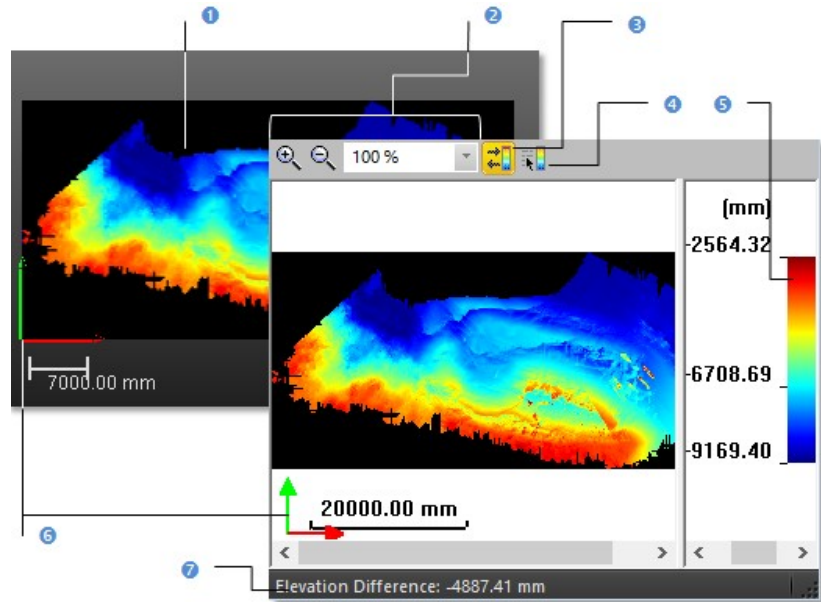
Inspection Maps

The inspection is done by comparing the **Model** with the selected **Surface**. The inspection result is a map shown in a specific window (called **Map Preview**) which opens above the **3D View**. This map is a 2D image inside which pixels are colored according to the difference (expressed in terms of elevation) between the **Surface** and the **Model**. A **ColorBar** located at the right side of the inspection map is a scale of elevation values and each color corresponds to an elevation value.

In the **Map Preview**, you can zoom the inspection map **In** (or **Out**) in three ways. The first one is to magnify (or reduce) an area of the inspection map by using **Image Zoom In**  and **Image Zoom Out** . The second way is to magnify (or reduce) the inspection map using the mouse wheel (if present). The last way is to select a rate from the drop-down list. If the inspection map is larger than the **Map Preview** window can display, you can pan it on left-click in four directions: **Up**, **Down**, **Right** and **Left**. You can also hide/show the current **ColorBar** or edit a new one.

In the **3D View**, the inspection map is shown in superposition with the two selected items (**Surface** and **Model**). A frame (of red and green colors) corresponding to the origin of the inspection map (also called **Orientation**) appears in both the **3D View** and the **Map Preview**.

Plane Shape



1 - Inspection map

2 - Zoom features

3 - Hide/Show ColorBar

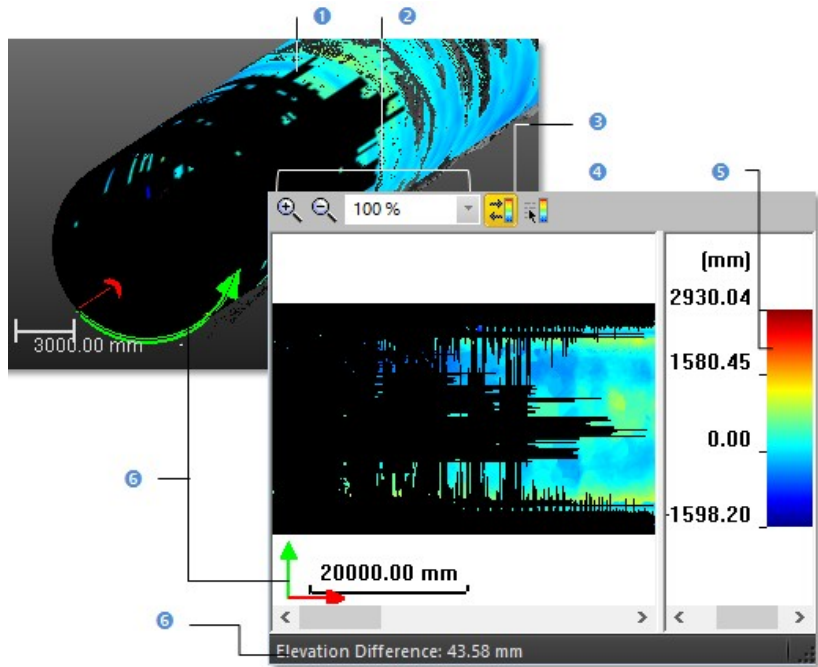
4 - Edit ColorBar

5 - ColorBar

6 - Origin of the inspection map

7 - Elevation difference at the cursor position

Cylinder Shape



1 - Inspection map

2 - Zoom features

3 - Hide/Show ColorBar

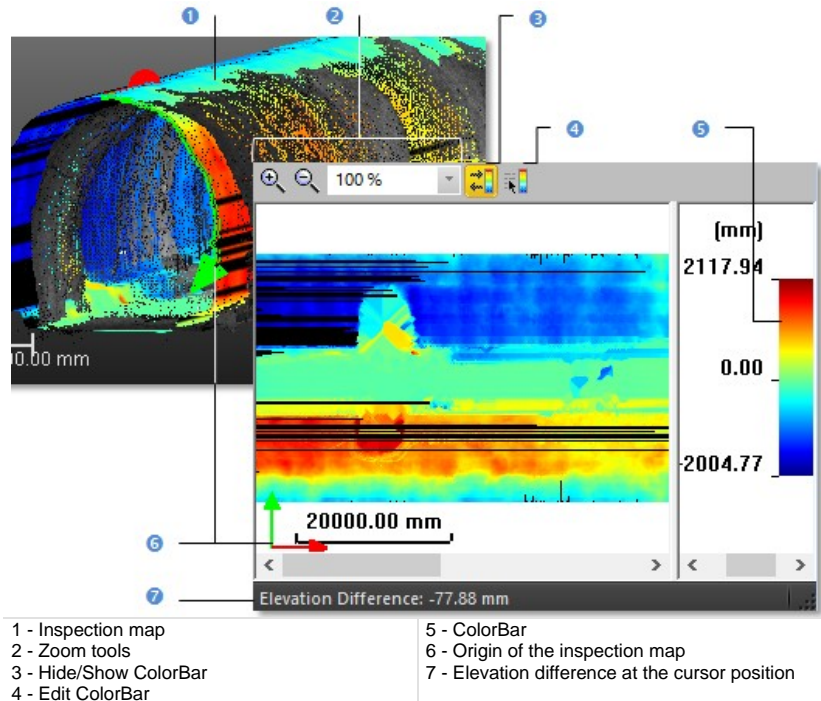
4 - Edit ColorBar

5 - ColorBar

6 - Origin of the inspection map

7 - Elevation difference at the cursor position

Tunnel Shape



Printing

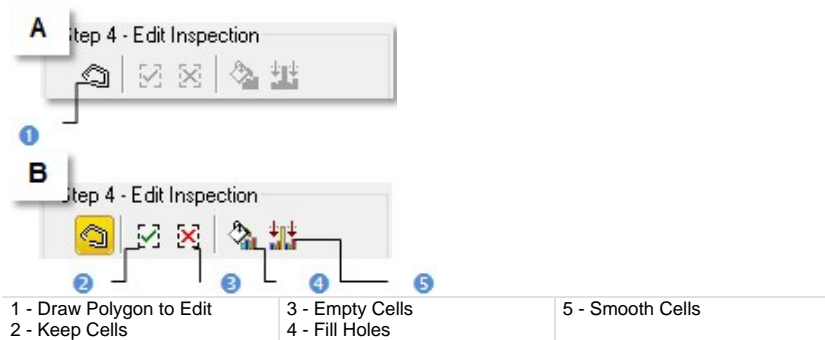
To print an inspection map, click inside the **Map Preview** window and do one of the following:

To Print an Inspection Map:

1. Click **Print** in the **Main** toolbar,
2. Or select **Print** from the **File** menu
3. Or select **Print** from the pop-up menu.

Editing

The inspection map computed previously may have irregularities like holes or spades, you can then edit it in order to keep or to remove the part you want, to complete the holes or to smooth the map. In edition mode, the mouse cursor will change its shape and picking is always inside the inspection map. Before you draw a polygonal fence, only the **Draw Polygon To Edit** tool can be used (see [A]). After you draw and validate a polygonal fence, the other tools become active (see [B]).



Fencing an Area

To Fence an Area:

1. Click the **Draw Polygon to Edit**.
2. Fence an area on the inspection map.
3. Right-click in the **3D View** window.
4. Select **End Fence** from the pop-up menu.

Notes:

- To cancel the current polygonal fence, you can press **Esc** or select **New Fence** from the pop-up menu.
- To leave the **Draw Polygon to Edit** tool, click again the **Draw Polygon to Edit**, select **Close Polygon Tool** from the pop-up menu or press **Esc**.

Tips:

- Instead of selecting **End Fence** from the pop-up menu, you can either double-click or press on the **Space Bar**.
- You can also select **Keep Cells** (or **Empty Cells**) from the pop-up menu or use their related short-cut key **I** (or **O**).

Filtering an Area

To Filter an Area:

1. Click the **Fill Holes**, if the fenced area contains holes.
2. Or click **Smooth Cells**, if the fenced area contains spikes for example.

Notes:

- You can also select **Fill Holes** (or **Smooth Cells**) from the pop-up menu or use their related short-cut key **F** (or **S**).
- An inspection map, once created, becomes un-editable.

Checking the Inspections

The map gives you a global vision of the inspection between the selected surface and the model. You can move your cursor over a point on the inspection map in order to have the difference of elevations between both the surface and the model at this point. The difference of elevations appears in text beneath the **Map Inspection**.

Saving the Inspections

In this last step, you can save the inspection result as a permanent object in the database, perform a screen printout or go further in analyzing the result thanks to the **Inspection Map Analyzer Tool**. For each inspection map you save, a geometric object is created and is put under the active group in the **Models Tree**. The **ColorBar(s)** that is (or are) related to it will be saved too. As any geometric object in **RealWorks**, you can display its representation in the **3D View** by double-clicking it in the **Project Tree** or by selecting the **Open Inspection Map** command from the **Display** menu. A **ColorBar** has no representation in the **Project Tree**.

Notes:

- To leave the **Surface to Model Inspection Tool**, you can press **Esc** or select **Close** from the pop-up menu.
- Leaving the **Surface to Model Tool** without saving the inspection result will open an information box which prompts you to confirm, undo or cancel the operation you attempt to execute.


Inspection Map Analyzer Tool

This tool allows you to extract five categories of information (**Points & Polylines**, **Sections & Shifts**, **Volumes & Surfaces**, **Iso-curves** and **Colored Meshes**) from an inspection map and each category corresponds to a sub-tool.

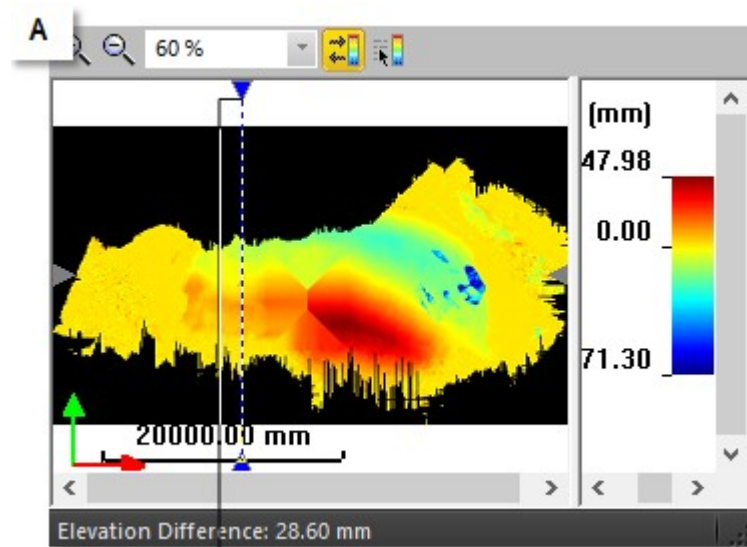
Opening the Tool

You can open the **Inspection Map Analyzer Tool** either from the main menu after selecting an existing inspection map from the **Project Tree** or inside the **Twin Surface Inspection** (or **Surface to Model Inspection**) **Tool** after inspecting.

To Open the Tool:

1. Select an inspection map from the **Project Tree**.
2. In the **OfficeSurvey** menu, select **Inspection Map Analyzer Tool** . The **Inspection Map Analyzer** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window. Depending on the sub-tool you select, the **Inspection Map Analyzer** dialog changes appearance. The selected inspection map is displayed in the **Inspection Map Analyzer Viewer [A]** with two sliders (horizontal and vertical).



Vertical slider (in this example)

The **Section Viewer** [B] shows the extraction result(s) in 3D and optionally the inspection map if the **Display 3D Map** option is checked. The **Lock in 2D** option (when not dimmed and if checked) locks the extractions result(s) in 2D (constrained in the XZ* plane of the active frame) with a **2D Grid** in superposition. You can hide and display again the **2D Grid**. When the **Lock in 2D** option is checked, you can only pan the results(s) in the YZ* plane, zoom (in or out) or rotate around the X* axis. Un-checking **Lock in 2D** will hide the **2D Grid** and will free the result(s) from the 3D locked constraint.



The way the two viewers (**Inspection Map Analyzer** and **Section**) will be represented depends on the sub-tool you select. You can rearrange the viewers as you please using the **View Manager** tools

Notes:

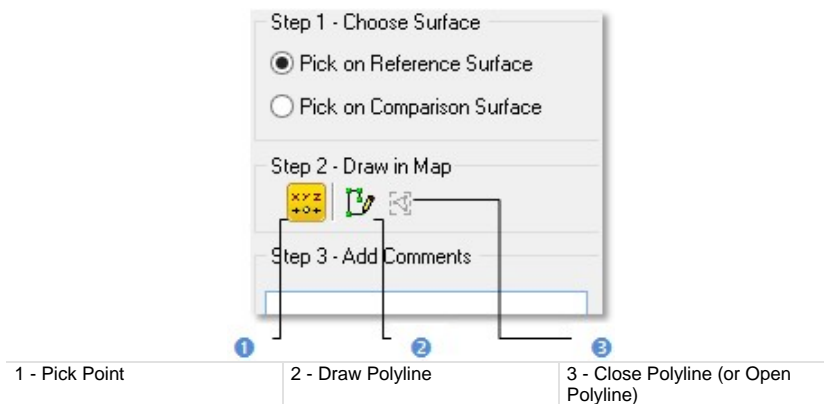
- Pressing **Esc** will close the **Inspection Map Analyzer Tool**.
- Clicking **Close** in the **Inspection Map Analyzer** dialog (or selecting **Close** from the pop-up menu in the viewers) will not close the current sub-tool but will close the main tool.
- (*) In the **X, Y, Z Coordinate System**.

Selecting Points & Polylines

Because the metric information is still stored in an inspection map, you can get the 3D position of a point as in the **Feature Set Tool** (surveying point) or as in the **Measurement Tool** (measured point) or simply define an area of interest by drawing a polyline.

To Select Points & Polylines:

1. In the **Inspection Map Analyzer** dialog, click on the pull down arrow.
2. Select **Points & Polylines**. The **Inspection Map Analyzer** dialog appears as shown.



The **Inspection Map Analyzer** and **Section Viewers** are both opened. In the **Section Viewer**, the inspection map is by default hidden and the navigation is locked in 2D. In the **Inspection Map Analyzer Viewer**, the horizontal and vertical sliders are shown and only one is active.


3. If required, check the **Display 3D Map** option to view the inspection map in the **Section Viewer**.
4. If required, uncheck the **Lock in 2D** option to free the navigation from 2D lock in the **Section Viewer**.
5. In **Step 1**, choose between **Pick on Reference Surface** and **Pick on Comparative Surface**.

Tip: You can use the **Up/Down** (or **Right/Left**) arrows of your keyboard to vertically (or horizontally) move the slider with constant step. This step corresponds to one pixel on the inspection map (not one pixel on the screen). To do this, you should first click inside the **Inspection Map Analyzer Viewer** to select it.

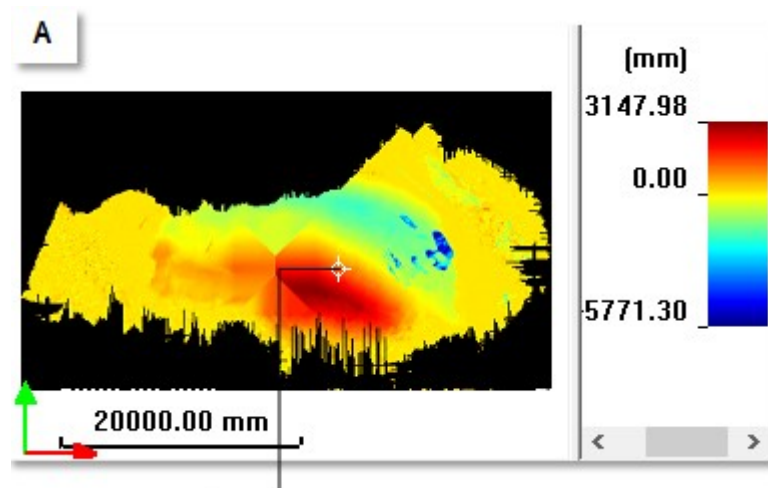
Picking Points

Picking should always be done inside the **Inspection Map Analyzer Viewer** and anywhere over the color area of the inspection map (except on black area).

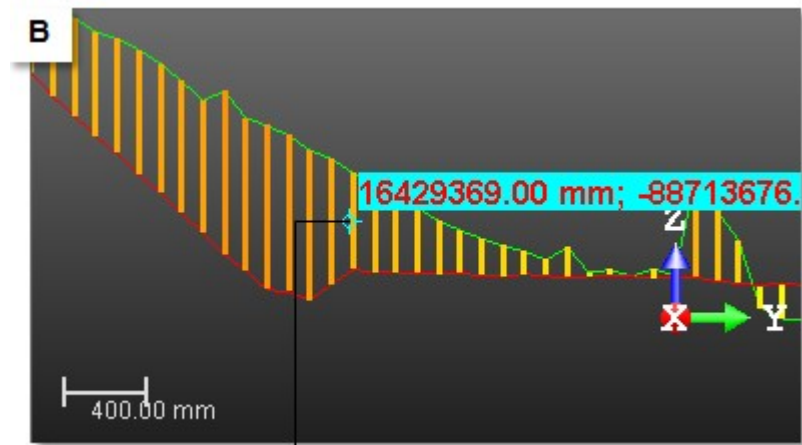
To Pick a Point:

1. Pan (or zoom) the inspection map in or out (if required).
2. Click the **Pick Point**  icon. The two sliders disappear from the **Inspection Map Analyzer Viewer**.
3. Pick a point on the inspection map in the **Inspection Map Analyzer Viewer**.

The picked point is shown in the **Inspection Map Analyzer Viewer** [A] and in the **Section Viewer** [B]. Its 3D position (XYZ coordinates) is displayed in text in the **Section** viewer. Starting a new picking will cancel the current one.



Picked point in the Inspection Map Analyzer Viewer



Picked point in the Section Viewer


Tips:

- You can also select **Quit Point Creation Mode** from the pop-up menu or press on the **Esc** key.
- You can remove the picked point labels from the **3D View** by first selecting **Rendering**, then **Display 3D Labels** from the **3D View** menu.

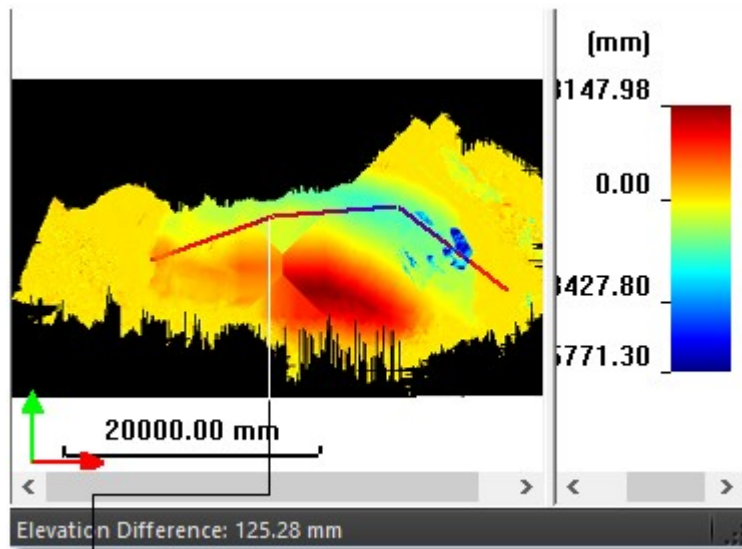
Drawing Polylines

Picking should be done on the colored areas of the inspection map.

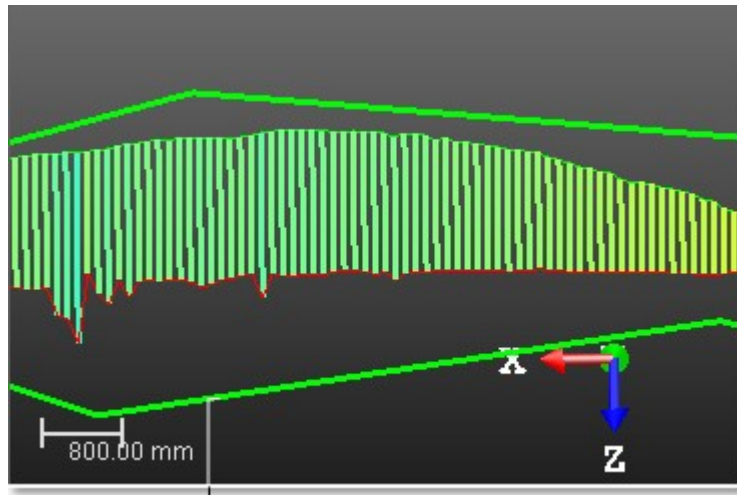
To Draw a Polyline:

1. Click the **Draw Polyline**  icon. The two sliders disappear from the **Inspection Map Analyzer Viewer** and the **Close Polyline** icon becomes active*.
2. Pick a series of points on the inspection map in the **Inspection Map Analyzer Viewer**.
3. Right-click and select **Close Polyline** from the pop-up menu. The start point is connected to the end point. **Close Polyline** becomes **Open Polyline**.
4. Right-click again and select **Open Polyline** from the pop-up menu. The start point is disconnected to the end point. **Open Polyline** becomes **Close Polyline**.
5. Double-click to end the polyline.

The drawn polyline appears in both of the two viewers (**Inspection Map Analyzer** and **Section**).



Drawn polyline in the Inspection Map Analyzer Viewer



Drawn polyline in the Section Viewer

6. If required, right-click in the **Inspection Map Analyzer viewer** and select **New Polyline** from the pop-up menu. The drawn polyline is cancelled. You can then start a new polyline.

Notes:

- You can neither draw a circle arc nor a series of discontinuous polylines.
- (*) The **Close Polyline** icon may define two creation modes. Not-clicked-on **Close Polyline** icon sets the open polyline creation mode. Clicked-on **Close Polyline** icon sets the close polyline creation mode.
- You cannot combine the **Pick Point** and **Draw Polyline** features together.

Tips:

- You can also select **Draw Polyline**, **Close Polyline**, **Open Polyline** and **End Polyline** from the pop-up menu.
- Pressing once **Esc** will cancel the current polyline. Pressing twice **Esc** will cancel the current polyline and leave the polyline tool.

Saving the Results

You can create the extracted result(s) in the database and start a new point picking (or polyline drawing) without leaving the tool. An object of **3D Point** (or **Polyline**) type is created and rooted in the **Models Tree**. Before saving a point (or polyline), add comments to it. After saving a point (or polyline), you can edit its related comments. To do this, display the created object properties and go to the **Label** line for edition.

To Save the Results:

1. For each picked point (or drawn polyline), you can add comments.
2. Click **Create** to save the result.

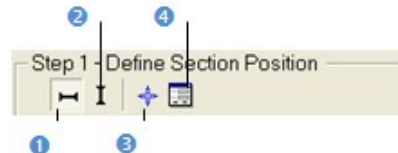
Note: Be sure to first save the picked point (or drawn polyline) before leaving the **Point Creation** (or **Draw Polyline**) **Tool**. Otherwise, it will be cancelled.

Selecting Sections & Shifts

Because the metric information is still stored in the inspection map, you can extract profiles and cross-sections from it as in the [Profile/Cross-Section Tool](#) or inspect along a given path as in the [2D-Polyline Inspection Tool](#). Profile/cross-section extraction or inspection is based on the use of sliders in the [Inspection Map Analyzer](#) viewer. In [Section & Shifts](#), the [Inspection Map Analyzer](#) and [Section](#) viewers are opened. The [Section](#) viewer is split in two sub-viewers*. The inspection map is by default hidden and the navigation is locked in 2D. In the [Inspection Map Analyzer](#) viewer, the horizontal and vertical sliders are shown and only one is active a time.

To Select Sections & Shifts:

1. In the [Inspection Map Analyzer](#) dialog, click on the pull down arrow.
2. Select [Sections & Shifts](#). The [Sections & Shifts](#) dialog appears.



1 - Set Horizontal Slider
2 - Set Vertical Slider



3 - Set Slider Position
4 - Set Slider Point Position

Note: (*) The left sub-viewer displays a section, a couple of sections, a set of section couples, a shift or a set of shifts. This depends on the option(s) ([Comparison Section](#), [Reference Section](#), [1D Inspection](#) or [Multi-Slices](#)) checked in [Step 2](#) and [Step 3](#). The right sub-viewer displays the difference plot between a couple of sections.


Defining a Slider Direction

To extract sections and shifts from an inspection map, you need to define a direction and a position over this map. The direction is given by a slider direction and the position by its position. You have at all two sliders (**Vertical** and **Horizontal**) in the **Inspection Map Analyzer** viewer and only one can be activated a time. Note that an active slider is the one in blue.

To Define a Slider Direction:

- Click **Set Horizontal Slider**  to define the horizontal direction.
- Or click **Set Vertical Slider**  to define the vertical direction
- Or position the cursor over a slider end and click on it.

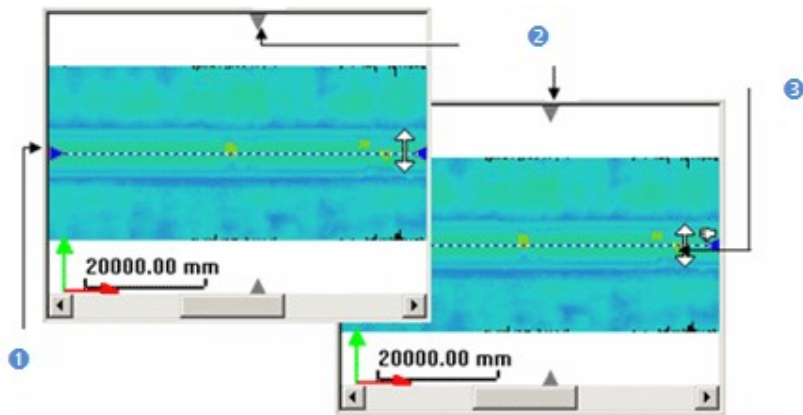
Notes:

- The **Set Horizontal Slider**  is grayed out when the input (the inspection map required to open the tool) is a **Tunnel**.
- You can move the two sliders in the four directions (**Right**, **Left**, **Up** and **Down**) using the arrow keys of your keyboard. In that case, the displacement is done with a constant interval. The interval value is the one you set in the **Step** field (see **Step 3** in the **Sections & Shifts** dialog).

Defining a Slider Position

To Drag and Drop a Slider Position:

1. Choose a slider's direction.
2. If **Horizontal Slider** has been selected, drag and drop it from **Up** to **Down** and reversely until it meets the position required. Section will be done along this horizontal direction and at this position.
3. If **Vertical Slider** has been selected, drag and drop it from **Right** to **Left** and reversely until it meets the position required. Section will be done along this vertical direction and at this position.



1 - The horizontal slider is the active slider

2 - The vertical slider

3 - The slider displacement directions


Note: The **Set Horizontal Slider** is grayed out when the input (the inspection map required to open the tool) is a **Tunnel**.

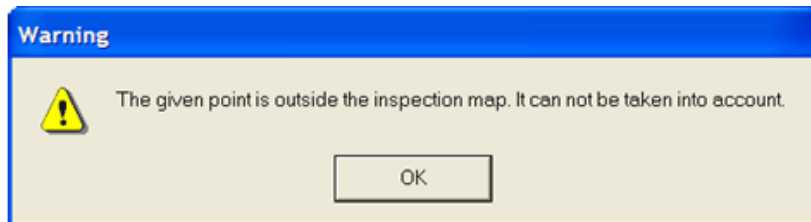
To Pick a Slider Point Position:

1. Choose a slider's direction.
2. Click **Set Slider Position**. The cursor changes its default shape to a pointer and the two sliders disappear from the **Inspection Map Analyzer** viewer.
3. Pick a point on the inspection map in the **Inspection Map Analyzer** viewer.
 - If the inspection map is a **Plane** (or a **Cylinder**); the active slider moves to the picked position appositively to the defined direction.
 - If the inspection map is a **Tunnel**; the **Vertical Slider** moves horizontally to the picked position.

Tip: You can also right-click anywhere in the **Inspection Map Analyzer** viewer to display the pop-up menu and select **Set Slider Position**.

To Define a Slider Point Position:

1. Click **Set Slider Point Position** . The **Slider Position Definition** dialog opens.
2. Do one of the following:
 - If the inspection map is a **Plane** (or a **Cylinder**); choose between **Horizontal Slider** and **Vertical Slider**.
 - If the inspection map is a **Tunnel**; only **Vertical Slider** can be chosen.
3. Enter a point 3D position in the **Point** field.
 - If the 3D position is out of the inspection map. A warning message appears in an independent dialog as shown below and in red in the **Slider Position Definition** dialog.
 - Clicking **OK** will close the warning message but do not hide it in the **Slider Position Definition** dialog.



4. Click **OK**. The **Slider Position Definition** dialog closes.
 - If **Horizontal Slider** has been chosen, the horizontal slider in the **Inspection Map Analyzer** viewer becomes active and jumps vertically to the defined position.
 - If **Vertical Slider** has been chosen, the vertical slider in the **Inspection Map Analyzer** viewer becomes active and jumps horizontally to the defined position.

To Double-Click a Slider Point Position:

1. Choose a slider's direction.
2. Position your cursor over a position on the inspection map in the **Inspection Map Analyzer** viewer.
3. Double-click.
 - If the inspection map is a **Plane** (or a **Cylinder**); the active slider moves to the picked position appositively to the defined direction.

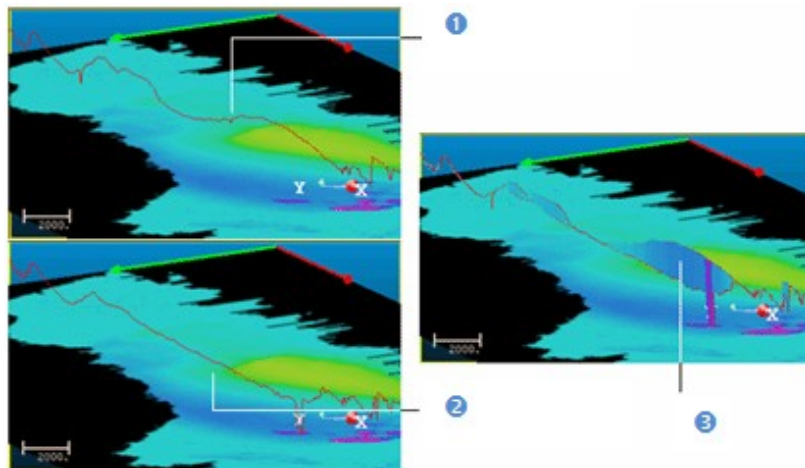
- If the inspection map is a **Tunnel**; the **Vertical Slider** moves horizontally to the picked position.

Extracting a Couple of Section and Shift

You can now select from which surface (**Reference** or **Comparison**) will be extracted the section, inspect the difference between both of them (**1D Inspection**) or combine all of them.

To Extract a Couple of Section and Shift:

- Clear **Reference Section**. The section (shown in red) done from the direction and the position defined previously on the **Reference Surface** is hidden.
- Clear **Comparison Section**. The section (shown in green) done from the direction and the position defined previously on the **Comparison Surface** is hidden.
- Clear **1D Inspection**. The shift done from to the direction and the position defined previously on the inspection map is hidden.



1 - Section along the first surface
2 - Section along the second surface

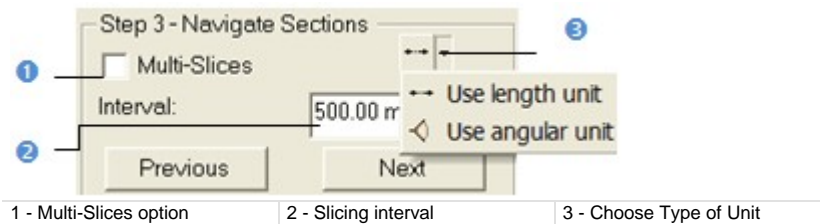
3 - 1D inspection between these two surfaces

Extracting a Set of Sections and Shifts

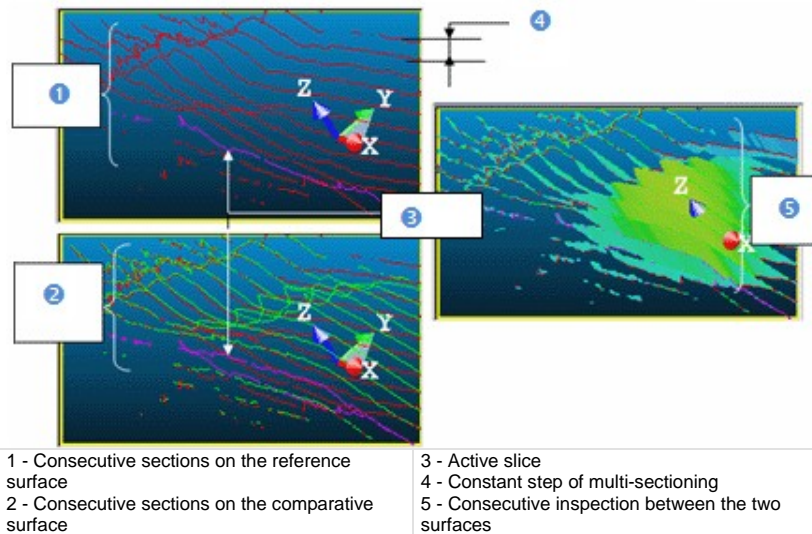
You can section the inspection map with constant interval along to the direction and position defined previously. The section results over the reference surface are always in red while the ones that result from the comparative surface sectioning are in green. Only one couple of sections (**Reference** and **Comparison**) can be active a time, it's the one in fuchsia.

To Extract a Set of Sections and Shifts:

1. In **Step 3**, check the **Multi-Slices** option.



2. Click on the **Choose Type of Unit** pull-down arrow*.
3. Do one of the following:
 - Select **Use Length Unit**. The unit of measurement in the **Interval** field will change according the unit of measurement set in **Preferences / Units / Unit System / Length**.
 - Select **Use Angular Unit**. The unit of measurement in the **Interval** field will change according the unit of measurement set in **Preferences / Units / Unit System / Angle**.
4. Enter a value in the **Interval** field.
5. Click (or) to set a value.
6. Click **Next** (or **Previous**).



Clicking **Next** or **Previous** moves the active slider forward/backward or up/down of one **Interval** step in the **Inspection Map Analyzer** viewer and will set the next or previous section/shift as active in the **Section** viewer.

Notes:

- Use the **Up/Down** or **Left/Right Arrows** of your keyboard to view the next (or previous) section (or inspection).
- (*) The **Choose Type of Unit** icon is only available when selecting an inspection map of cylinder shape and if the horizontal slider has been set.

Saving the Extracted Data

Saving a section result will create and root a polyline object type under the current project in the **Models Tree**. If the result is an inspection, a 1D inspection object type will then be created. If the multi-slices option is used, a folder of polyline objects (or 1D inspection objects) will be created.

To Save the Extracted Data:

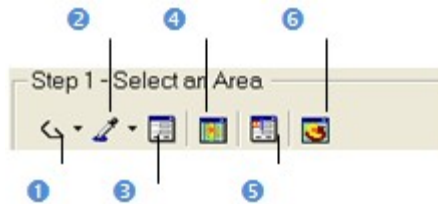
- Click **Create** (or **Create All**).
- Click **Close**.

Selecting Volumes & Surfaces

Because the metric information is still stored in the inspection map, you can extract the volume and surface information (respectively in cubic meters and in square meters) from it as in the **Volume Calculation Tool**. You can do this by fencing a specific area or a series of areas. You can also filter according to a color defined by picking, from a range of colors by specifying values or using surface (or altitude) values. In selection, the mouse pointer changes depending on the mode you are using. In **Volumes & Surfaces**, only the **Inspection Map Analyzer** viewer* is opened and the horizontal and vertical sliders are hidden**.

To Select Volumes & Surfaces:

1. In the **Inspection Map Analyzer** dialog, click on the pull down arrow.
2. Select **Volumes & Surfaces**. The **Volumes & Surfaces** dialog appears.



- | | |
|---------------------------|--------------------------------------|
| 1 - Polygonal Selection | 4 - Select By Using Surface Values |
| 2 - Color Range Selection | 5 - Altitude Filter |
| 3 - Define Color Range | 6 - Select Whole Inspection Map Area |

3. Select the whole map by clicking **Select Whole Inspection Map Area**.
4. Or fence an area,
5. Or pick a color,
6. Or define a color range,
7. Or use surface values,
8. Or filter altitudes.


Notes:

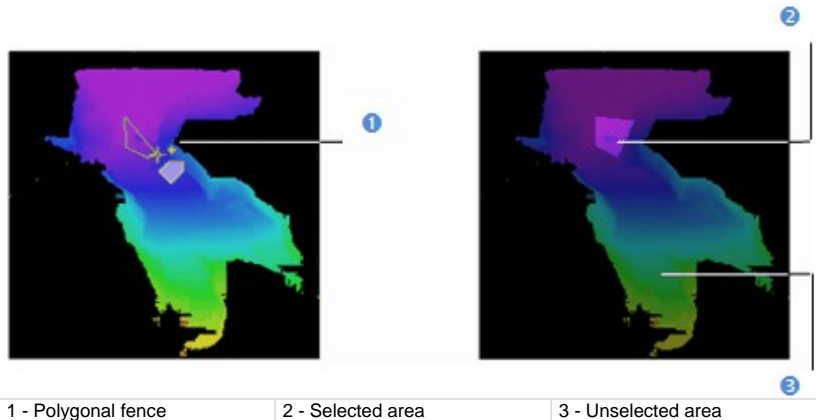
- You can right-click anywhere in the **Inspection Map Analyzer** viewer to display the pop-up menu and select the command you wish to use.
- * Click **Restore Down** in the **Inspection Map Analyzer** viewer to swap to the **Inspection Map Analyzer** and **Section** viewers' display.
- ** To make one of the two sliders appeared, select **Display Slider** from the pop-up menu.
- To be able to view the 2D lock (or the inspection map) when checking the **Lock 2D** (or **Display 3D Map**) option in the dialog, please first restore the default layout (**Inspection** and **Section** viewers).

Fencing an Area


Fencing an area is always done by picking anywhere (colored and black areas) on the inspection map. If no area has been fenced, the whole inspection map will be selected by default.

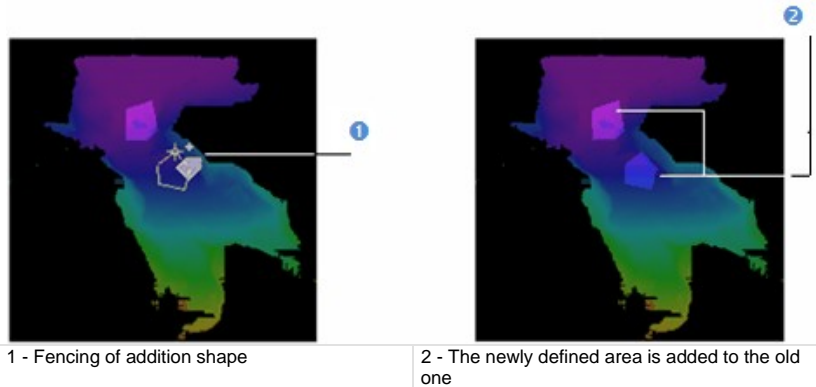
To Fence an Area:

1. Click **Polygonal Selection** .
2. Pan or zoom the inspection map in or out (if required).
3. Draw a polygonal fence by picking and double-click to end.



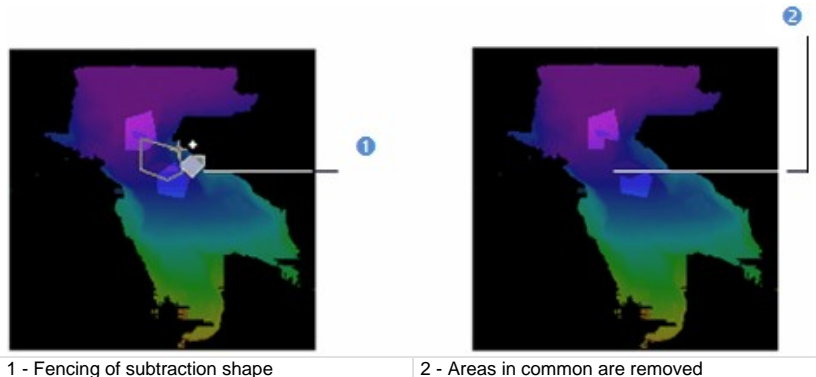
Tip: Pressing **Esc** or selecting **New Fence** or **Close Polygon Tool** from the pop-up menu will undo the polygonal fence in progress.

4. Click on the pull down arrow of the **Polygonal Selection** list.
5. Select **Add by Polygonal Selection** .
6. Fence another area. This newly fenced area is added to the previous one.



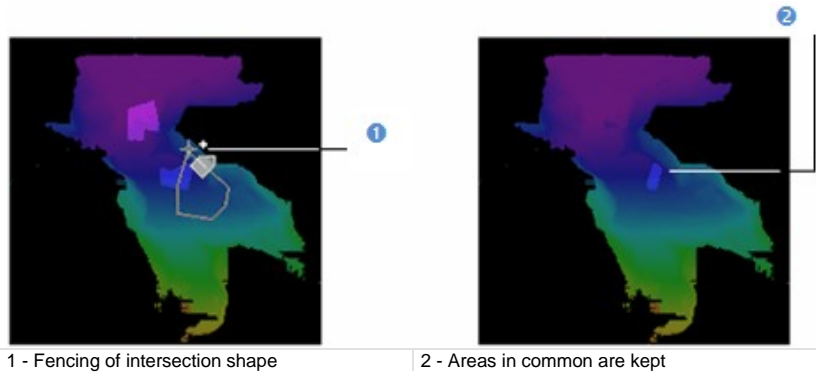
Tip: Instead of selecting **Add By Polygonal Selection**, you can also use its related shortcut key **Ctrl**. You should do this before ending the polygon.

7. Click on the pull down arrow of the **Polygonal Selection** list.
8. Select **Subtract by Polygonal Selection**.
9. Draw another polygonal fence. Areas in common are subtracted.



Tip: Instead of selecting **Subtract By Polygonal Selection**, you can also use its related shortcut key **Shift**. You should do this before ending the polygon.

10. Click on the pull down arrow of the **Polygonal Selection** list.
11. Select **Intersect with Polygonal Selection**.
12. Draw another polygonal fence. Areas in common are kept.




Tip: Instead of selecting **Intersect With Polygonal Selection**, you can also use its related shortcut key **Shift + Ctrl**.

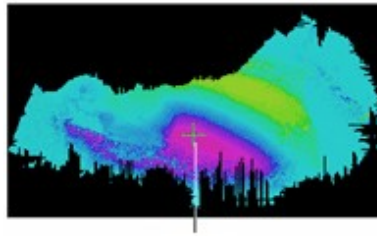
Note: If there is no common area, all fenced areas will be cancelled.

Tip: To leave the polygon selection mode, you can select **Close Polygon Tool** from the pop-up menu, click on any icon in the **Volumes & Sections** dialog.

Picking a Color

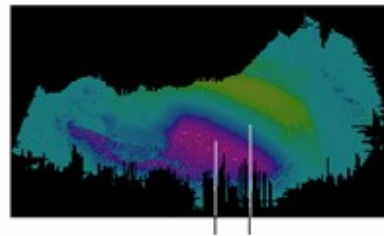
To Pick a Color:

1. Picking should always be done inside the **Inspection Map Analyzer** viewer and anywhere over the colored areas of the map.
2. Pan or zoom the inspection map in or out (if required).
3. Click **Select Color Range** .
4. Pick a pixel on the inspection map. The map is filtered according to the picked point color; the areas of this map sharing the same color than the picked point are kept.



1


1 - Cursor shape when selecting Select Color Range

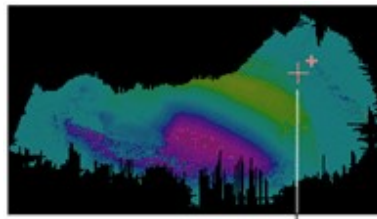


2

3

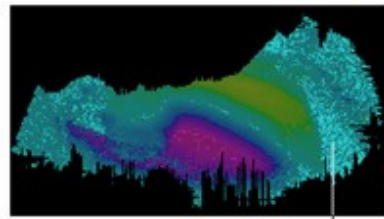
2 - Filtering color
3 - Unfiltered colors are darken

5. Click on the pull down arrow of the **Color Range Selection** list.
6. Click **Add Color Range Selection** .
7. Pick another pixel on the inspection map. The map is filtered according to the newly picked color.



1

1 - Cursor shape when selecting Add Color Range Selection




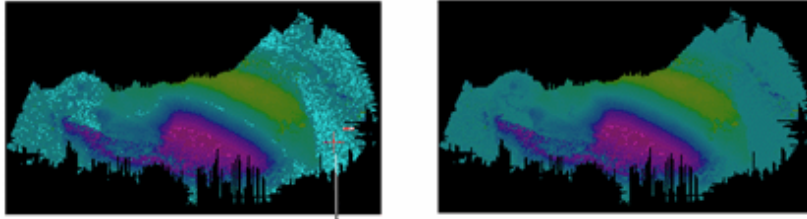
2

2 - New filtering color

Tip: Instead of selecting **Add Color Range Selection**, you can also use its related shortcut key **Ctrl**. You should do this before picking another pixel.


8. Click on the pull down arrow of the **Color Range Selection** list.

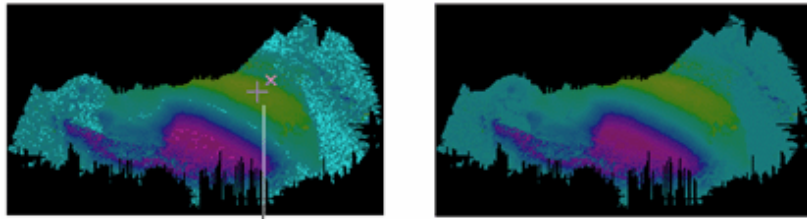
9. Select **Subtract Color Range Selection** .
10. Pick another pixel on the inspection map. The areas of the map having the same color than the picked point are subtracted from the selection.



Cursor shape when selecting Subtract Color Range Selection

Tip: Instead of selecting **Subtract Color Range Selection**, you can also use its related shortcut key **Shift**. You should do this before picking another pixel.

11. Click on the pull down arrow of the **Color Range Selection** list.
12. Select **Intersect Color Range Selection** .
13. Pick another pixel on the inspection map. The areas of the map having the color in common are kept. Those not having the same color are unkept.



Cursor shape when selecting Color Range Selection

Tip: Instead of selecting **Intersect Color Range Selection**, you can also use its related shortcut key **Shift + Ctrl**.

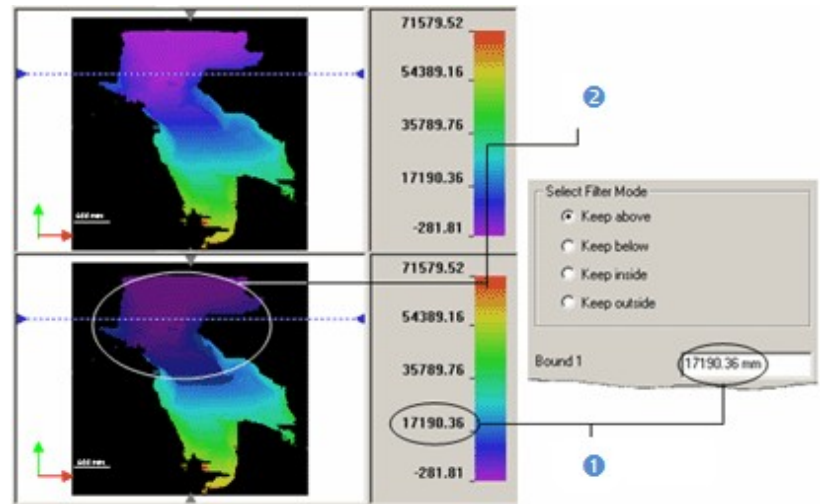
Note: If there is no pixel area, all picked pixels will be cancelled.

Tip: To leave the selection by color range mode, you can select **Quit Selection** by **Color Range** from the pop-up menu, click on any icon in the **Volumes & Sections** dialog.

Defining a Color Range

To Define a Color Range:

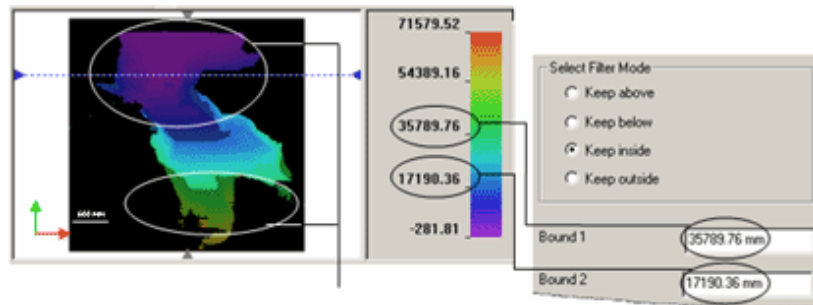
1. Pan or zoom the inspection map in or out (if required).
2. Click **Define Color Range**. The **Color Range Definition** dialog opens.
3. Do one of the following:
 - Filter according to one bound.
 - a) Choose between **Keep Above** and **Keep Below**.
 - b) Enter a value in the **Bound 1** field.



1 - Bound value

2 - The elevation differences above or below the bound value are kept and the others are darkened

- Filter between two bounds.
 - a) Select **Keep Inside** or **Keep Outside**.
 - b) Enter a value in the **Bound 1** and **Bound 2** fields.



The elevation difference inside or outside the bound values are kept and the others are darkened

4. Click **OK**.

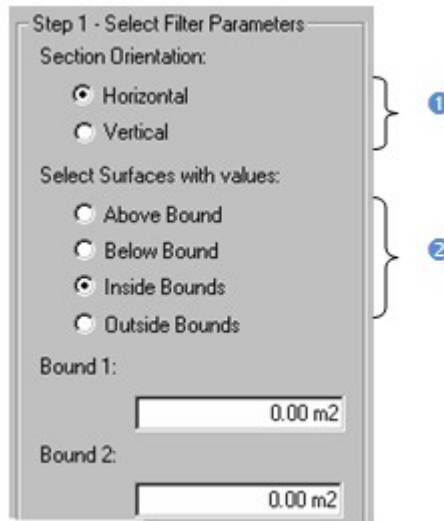
Using Surface Values

The shift between two surfaces on an inspection map along a given direction and at a defined position can be viewed in the **Section** viewer when selecting **1D Inspection** in **Sections & Shifts**. This shift is a surface of its own. That's why; the purpose of this new feature in **RealWorks** is to allow the user to filter an inspection map according to a surface value or between two values.

To Use the Surface Values:

1. Click **Select by Using Surface Values**. The **By Surface Selection** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window in place of the **Inspection Map Analyzer** dialog. In **Select by Using Surface Values**, only the **Inspection Map Analyzer** viewer is opened and the horizontal and vertical sliders appear again.

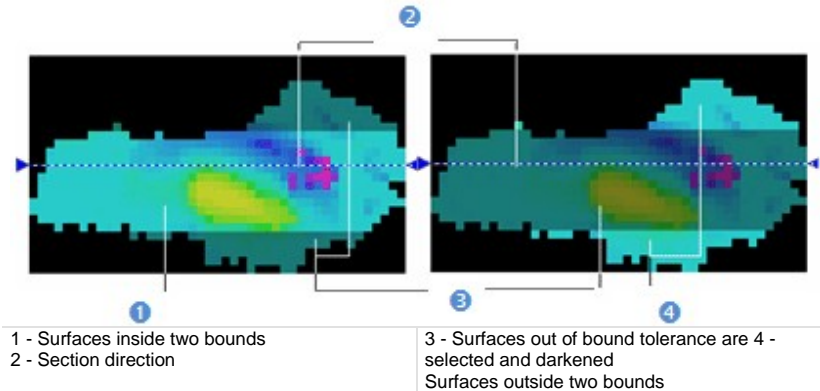


1 - Allows setting a section direction

2 - Allows filtering above/below a value or inside/outside two values

2. Do one of the following:
 - Check the **Horizontal** option. The horizontal slider becomes active.
 - Check the **Vertical** option. The vertical slider becomes active.
3. Do any of the following:

- Select surfaces with values above a bounded value.
 - a) Select the **Above Bound** option.
 - b) Enter a value in the **Bound 1** field.
 - Select surfaces with values below a bounded value.
 - a) Select the **Below Bound** option.
 - b) Enter a value in the **Bound 1** field.
 - Select surfaces with values between two bounded values.
 - a) Select the **Inside Bounds** option.
 - b) Enter a value in the **Bound 1** field.
 - c) Enter a value in the **Bound 2** field.
 - Select surfaces with values outside two bounded values.
 - a) Select the **Outside Bounds** option.
 - b) Enter a value in the **Bound 1** field.
 - c) Enter a value in the **Bound 2** field.
4. Click **Preview**. Shifts between surfaces are extracted.



5. Check the results in **Step 2**.
6. Click **Apply**.

Tip: You can also choose **Select by Using Surface Values** from the pop-up menu.

Filtering Altitudes

The new **RealWorks** feature allows you to filter an inspection map based on a given altitude or between two defined altitudes. The inspection map needs to be of **Cylinder** shape.

To Filter an Altitude:

1. Click **Altitude Filter**. The **Altitude Filtering** dialog opens.
2. Do one of the following:
 - Filter according to one bound.
 - a) Choose between **Keep Above** and **Keep Below**.
 - b) Enter a value in the **Bound 1** field.
 - Filter between two bounds.
 - a) Select **Keep Inside** or **Keep Outside**.
 - b) Enter a value in the **Bound 1** and **Bound 2** fields.
3. Click **OK**. The **Altitude Filtering** dialog closes.

Reporting the Volume and Surface Information

The extraction results are listed in text in the **Volumes & Surfaces** dialog. You can save these results in a report file in rft format. Note that you cannot save the results as permanent objects in the **RealWorks** database as in the **Volume Calculation**, **Twin Surface Inspection** and **Surface to Model Inspection** tools.

To Report the Volume and Surface Information:

1. Click **Report**. The **Volume Calculation Report** dialog opens.
2. Click **Export**. Another **Volume Calculation Report** dialog opens.
3. Enter a name for the report file in the **File Name** field.
4. Find a location where you want the report file to be stored.
5. Click **Save**.

Iso-Curves

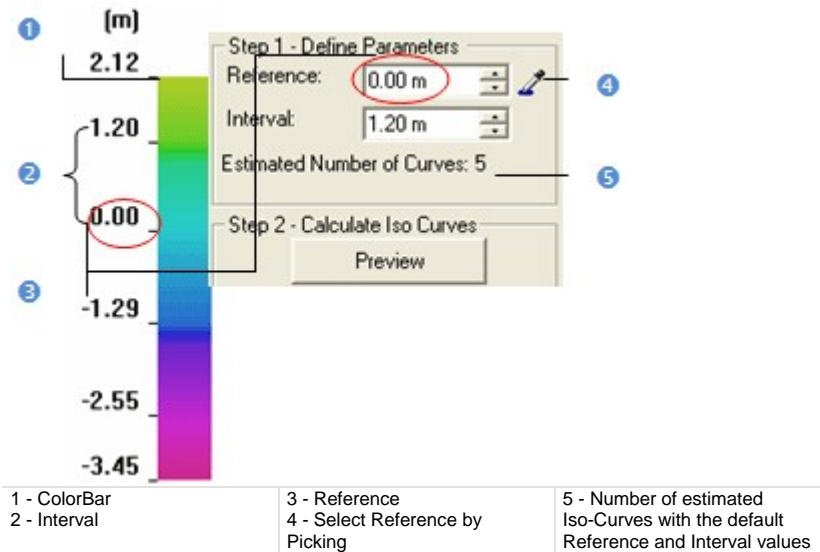
You can extract and create **Iso-Curves** (of polyline type) from an inspection map. **Iso-Curves** are computed from a position (called **Reference**) with a certain spacing (called **Interval**).

Extracting Iso-Curves

In **Iso-Curves**, the **Inspection Map Analyzer** and **Section** viewers are opened. The horizontal and vertical sliders are hidden in the **Inspection Map Analyzer** viewer and the **Section** viewer is empty of contents*.

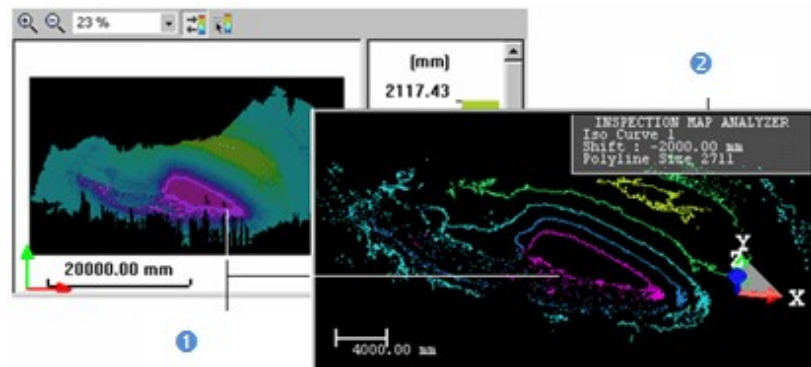
To Extract Iso-Curves:

1. In the **Inspection Map Analyzer** dialog, click on the pull down arrow.
2. Select **Iso-Curves**. The **Iso-Curves** dialog appears. The inspection map is by default not displayed in the **Section** viewer.
3. If required, check the **Display 3D Map** option to view the inspection map.



4. To define the **Reference**, do one of the following:
 - Enter a value in the **Reference** field.
 - Click (or) to set a value.
 - Click **Select Reference by Picking** and pick one point on the inspection map.
5. To define the **Interval**, do one of the following:
 - Enter a value in the **Interval** field.
 - Click (or) to set a value.
6. Click **Preview**.

The first computed **Iso-Curve** will be the **Reference Iso-Curve**; this means that it will have height the value set in the **Reference** field. The other **Iso-Curves** will then be computed from the **Reference Iso-Curve** by propagation in two directions (up and down) with the defined Interval. The active **Iso-Curve** (not necessarily the **Reference Iso-Curve** but the first in the computation's order) is of fuchsia color in the **Section** viewer and is colored according to its value in the **ColorBar** in the **Inspection Map Analyzer** viewer. For a given active **Iso-Curve**, the information window at the top right corner of the **Section** viewer shows in text its order, shift and size (of the polyline).



1 - Active iso-curve

2 - Information window

Notes:

- Computing a huge quantity of **Iso-Curves** may take a very long time. In that case, an information dialog appears and prompts you to continue or abort the operation.
- * To make one of the two sliders appeared, select **Display Slider** from the pop-up menu.
- To be able to view the iso-curves in the **Section** viewer, hide the inspection map in the 3D View by un-checking the **Display 3D Map** option.

Browsing Iso-Curves

The active **Iso-Curve** is the first in the computation's order. It is of fuchsia color in the **Section** viewer and is colored according to its value in the **ColorBar** in the **Inspection Map Analyzer** viewer.

To Browse Iso-Curves:

1. Click **Display Next Iso-Curve** ► (or **Display Previous Iso-Curve** ◀) to set the next (or previous) iso-curve as active.
2. Or click **Display First Iso-Curve** ►◀ (or **Display Last Iso-Curve** ◀◀) to set the first (or last) iso-curve as active.
3. Or key in an iso-curve's number and press **Enter**.
4. Or pick an iso-curve in the **Inspection Map Analyzer** viewer.

Notes:

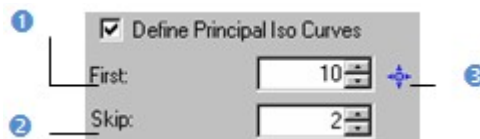
- You can set several iso-curves as actives. To do this, multi-select the ones you need in the **Section** viewer by combining the use of **Ctrl** with clicking.
- You can use the **Up** (or **Down**) **Arrow** key instead of **Display Next Iso-Curve** (or **Display Previous Iso-Curve**).

Defining Principal Iso-Curves

After previewing the result, you can choose and assign some of the **Iso-Curves** as principals. The remained **Iso-Curves** are then considered as intermediate **Iso-Curves**. Each principal **Iso-Curve** has a label which contains its height information in text.

To Define Principal Iso-Curves:






1. Check the **Define Principal Iso-Curves** option.

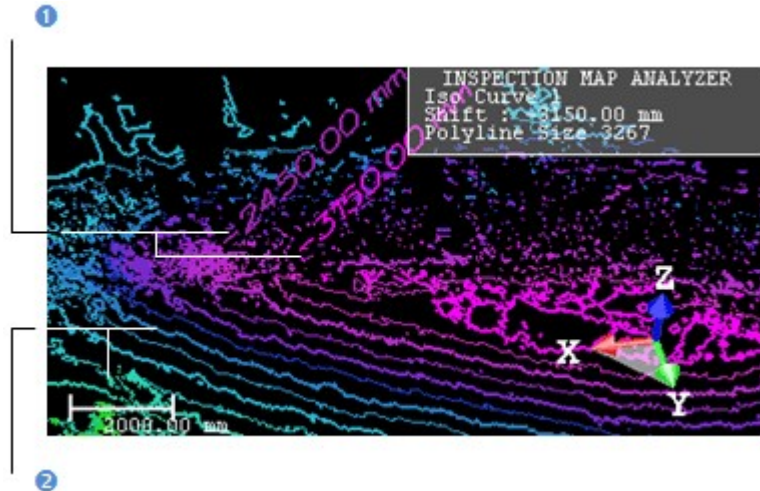


1 - Defines the first principal Iso-Curve by entering a number

2 - Defines the number of Iso-Curves to skip

3 - Select First Principal Iso-Curve by Picking

2. To define the **First** value, do one of the following:
 - Enter a number in the **First** field.
 - Or click  (or ) to set a number.
 - Or click  and pick an **Iso-Curve** in the **Section** viewer.
3. To define the **Skip** value, do one of the following:
 - Enter a number in the **Skip** field.
 - Or click  (or ) to set a number.



1 - Principal Iso-Curves

2 - Intermediate Iso-Curves

Creating Iso-Curves

Iso-Curves are created in the **RealWorks** database as 3D polylines in a set (named **Cross-MapData-IsoCurves XX** where **XX** is the **Interval** value) under the current (active) project. You can export them to the AutoCAD application.

To Create Iso-Curves:

1. Click **Create All**.
2. Click **Close**.

Notes:

- **Close** can also be selected from the pop-up menu.
- Leaving the **Iso-Curves** sub-tool without saving the result will make appeared a warning message.

Colored Meshes

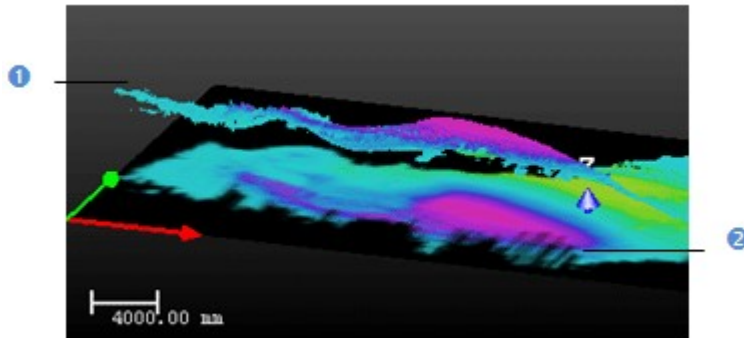
You can extract and create a **Colored Mesh** not based on the point clouds, meshes or model (prerequisites for doing the inspection) but directly from a surface (**Reference** or **Comparison**) of an inspection map. A **Colored Mesh** is built by using both the grid information and the color information found in the inspection map. The grid information is used for computing vertices while the color information is required for coloring.

Extracting Colored Meshes

In **Colored Mesh**, the **Inspection Map Analyzer** and **Section** viewers are opened and the horizontal and vertical sliders are hidden*.

To Extract Colored Meshes:

1. In the **Inspection Map Analyzer** dialog, click on the pull down arrow.
2. Select **Colored Mesh**. The **Colored Mesh** dialog appears. The inspection map is by default not displayed in the **Section** viewer.
3. If required, check the **Display 3D Map** option to view the inspection map.
4. Do one of the following:
 - Check the **Reference Surface** option.
 - Check the **Comparison Surface** option.
5. Click **Preview**. A colored mesh is then extracted.



1 - Colored mesh

2 - Inspection map

Notes:

- After clicking **Preview**, the number of vertex and the number of triangles related to the computed mesh are shown in the **Colored Mesh** dialog.
- * To make one of the two sliders appeared, select **Display Slider** from the pop-up menu.

Creating Colored Meshes

To Create Colored Meshes:

1. Click **Create**.
2. Click **Close**.

Notes:

- **Close** can also be selected from the pop-up menu.
- Leaving the **Colored Mesh** sub-tool without saving the result will make appeared a warning message.

Printing Inspection Maps

To Print an Inspection Map:

1. Right-click inside the **Inspection Map Analyzer** view.
2. Select **Print** from the pop-up menu. The **Print Setup** dialog opens.
3. Choose a printer from the **Printer** panel.
4. Choose a size and a source from the **Paper** panel.
5. Define an orientation for the paper by checking either **Portrait** or **Landscape**.
6. Fill in comments in the **Legend** panel.
7. Choose between **Auto Scale** and **Fixed Scale**.
8. If **Fixed Scale** has been chosen, click on the pull-down arrow and select a scale from the list.
9. Click **Preview**. A preview of the inspection map appears.
10. In the preview mode, do one of the following:
 - Click **Print** to print the inspection map.
 - Click **Zoom In** to magnify the inspection map.
 - Click **Zoom Out** to reduce the inspection map (after zooming in).
 - Click **Next Page** (if there is more than one page).
 - Click **Prev Page** (if the current page is other than the first page).
 - Click **Two Pages** (if there is more than one page).
 - Press **Esc** to leave the preview mode.
11. Click **OK**. The **Print Setup** dialog closes.

Tips:

- You can also click **Print** in the **Main** toolbar or select **Print** from the **File** menu.
- You can also select a printer from the network (or set the current printer's properties) by clicking on the **Network** (or **Properties**) button in the **Print Setup** dialog.


Profile Matcher Tool

The idea behind this tool is to allow the user to match or position a profile (2D curve, cross-section, polyline, etc.) at a specific point and in a given direction in a 3D scene. We mean by “matching a profile” not just to move it from its current position to a new one in a 3D scene but also to create this profile in the **RealWorks** database. This tool is useful when you import a profile from a CAD application, and you wish to position it within a 3D scene in **RealWorks**, or when you use the **EasyProfile™ Tool** or when you wish to duplicate a profile's pattern in different locations in a 3D scene.

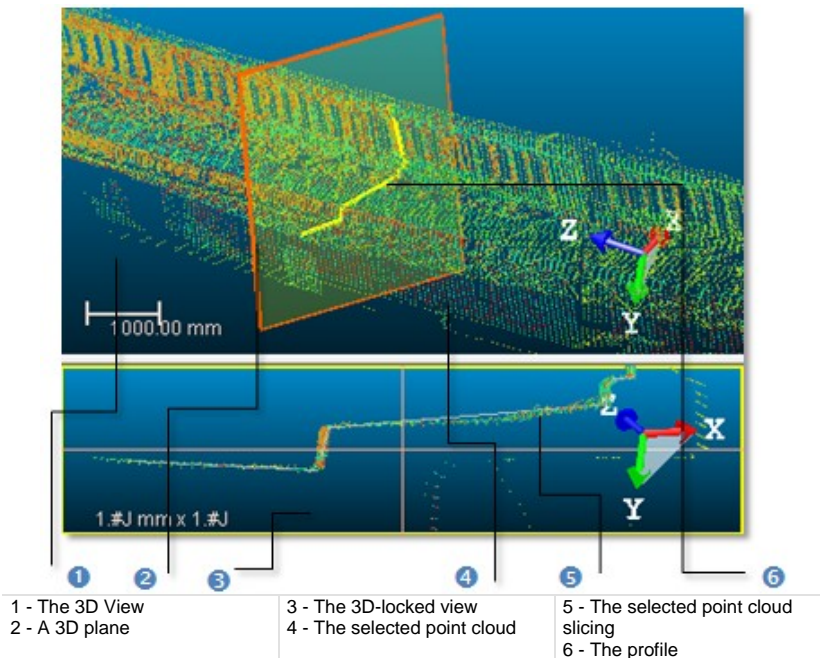
Opening the Tool

You need to have at least one profile and a point cloud or mesh selected in your current project to be able to activate the tool. You can activate it from the **EasyProfile™ Tool** or by selecting its related command from the menu bar.

To Open the Tool:

1. Select both a point cloud/mesh and a profile from the **Project Tree**.
2. In the **OfficeSurvey** menu, select **Profile Matcher Tool** . The **Profile Matcher** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window and is composed of four parts. Each part corresponds to one step in the matching process. The **3D View** splits into two horizontal viewers. The top viewer (a 3D viewer) displays in 3D the selected point cloud or mesh and the selected profile as well as a 3D-plane. The selected profile is in the 3D-plane. This means that both have the same position and orientation in the 3D scene. The lower viewer (a 3D-locked viewer) displays in 2D (locked in the XY* plane of the active coordinate frame) the selected profile and a set of points with a 2D-grid in superimposition. This set results from slicing the selected point cloud.

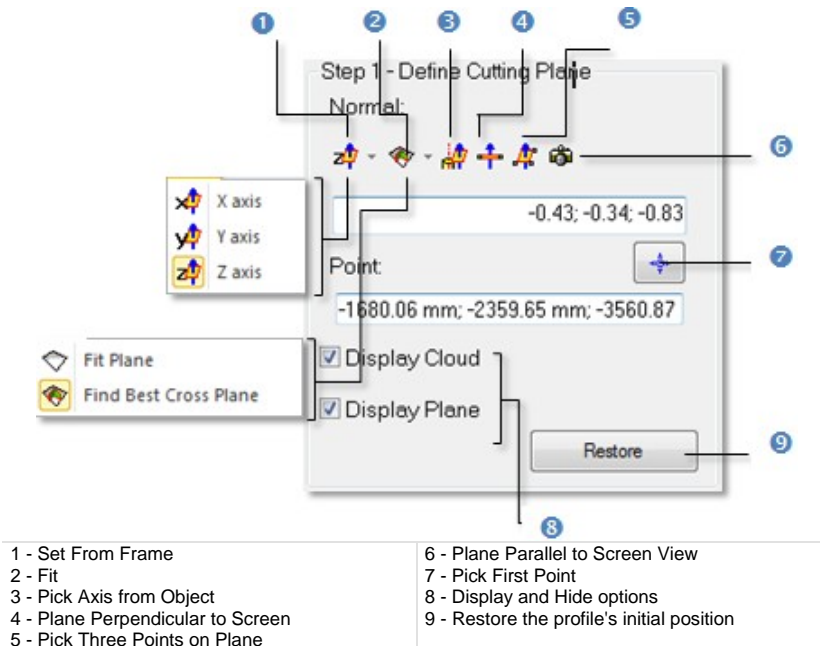


Notes:

- The **Picking Parameters** toolbar opens in the distance constrain mode below the **3D View**.
- (*) In the **X, Y, Z Coordinate System**.

Defining a Cutting Plane

In this step, we are going to define and use a 3D-plane to change the selected profile's position and orientation. The method for defining a 3D-plane (mostly the same as in the **Cutting Plane Tool**) is based on picking which can be free or constrained.



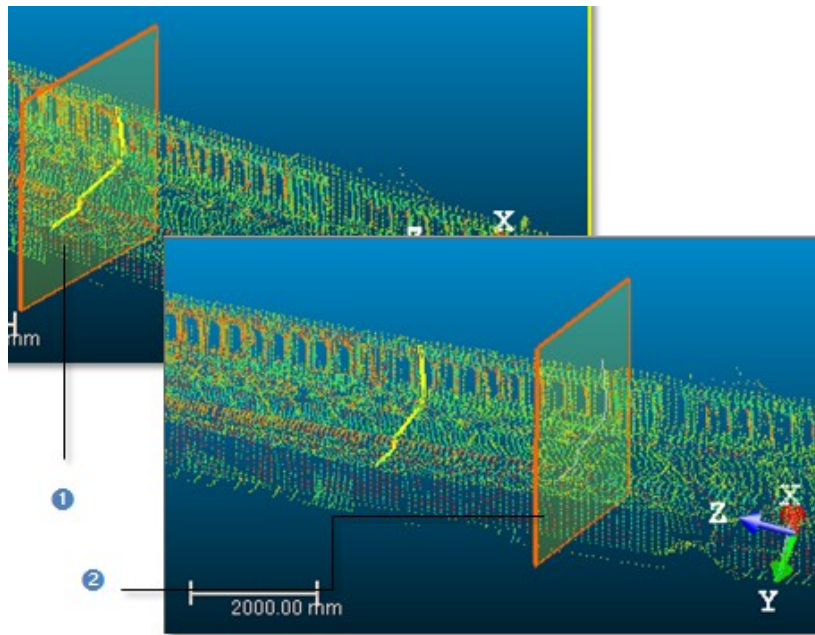
To Define a Cutting Plane:

1. Do one of the following:

- Select a frame's axis (1).
- Fit an extracted set of points with a plane (1).
- Find a perpendicular view plane from an extracted set of points (1).
- Pick an object's axis (1).
- Pick a plane perpendicular to the screen (1).
- Pick three points (1) (2).
- Set the plane parallel to the screen view.
- Define a position and set a direction:
 - a) Enter a direction in the **Normal** field.
 - b) Or click **Pick Point**. The **Picking Parameters** toolbar appears.

- c) Pick on point (free or constrained) in the **3D View**.
 - d) Enter a position in the **Point** field.
2. If required and if a point cloud has been selected, un-check **Display Cloud**.
 3. If required and if a mesh has been selected, un-check **Display Mesh**.
 4. If required, un-check **Display Plane**.
 5. If required, click **Restore**.

The **Restore** button remains unavailable as long as any 3D-plane has been validated. After validating a 3D-plane, a profile of the same shape as the selected one (the one required to activate the tool) is created. This profile is in the validated 3D-plane. Any transformation has been applied to the selected profile; it remains unchanged in position and direction. Clicking **Restore** will undo the new 3D-plane as well as the new profile.

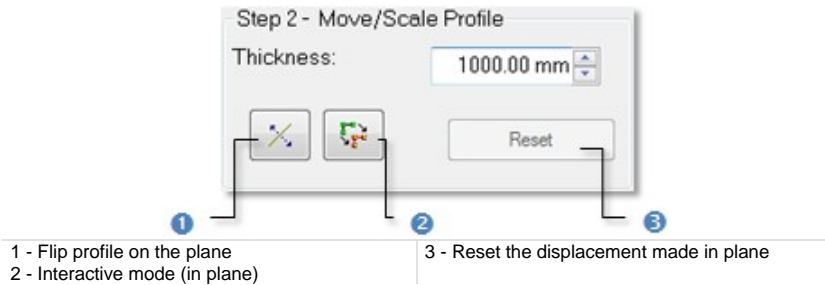


1 - The initial 3D plane with the selected profile 2 - The new 3D plane with new profile inside

Note: For more information related to (1), see **Step 2** of the **Cutting Plane Tool**. For (2), see the **Picking Parameters**.

Defining a Profile

You can arrange the newly defined profile in the 3D-plane by rotating, panning, scaling up and down or reversing it. The **Thickness** field is not unavailable (grayed out) if a mesh has been selected in **Step 1**. The **Thickness** value (cannot be equal to zero) is used for slicing the selected point cloud from each side of the 3D plane.




Setting a Thickness

To Set a Thickness:

1. Enter a value in the **Thickness** field.
2. Or use the **Up** and **Down** buttons to select a value.

Moving the Profile

To Move the Profile:

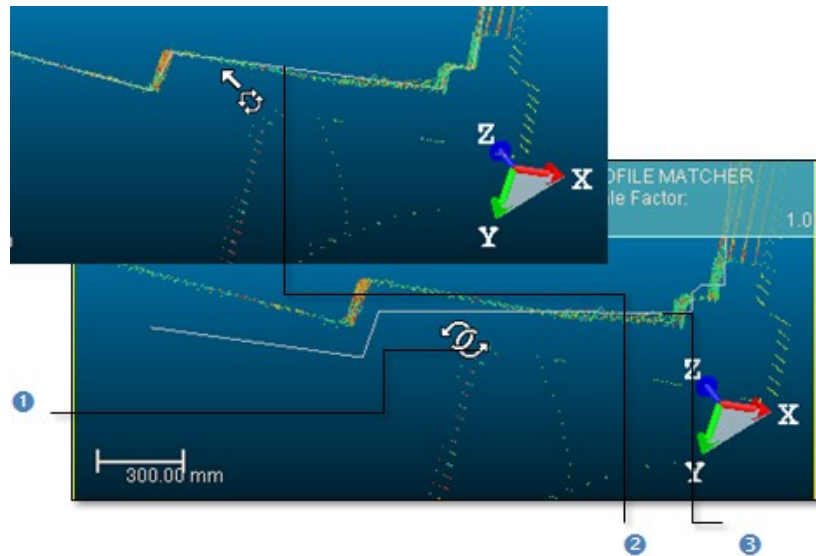
1. Click **Interactive Mode (in Plane)** . The **Profile Matcher** information window appears at the top right corner of the 3D-locked viewer. This window displays the profile's current scale factor.
2. Do one of the following:
 - Rotate the profile in the 3D plane.
 - Pan the profile in the 3D plane.
 - Scale the profile in the XY plane.
 - Scale the profile up and down using the mouse wheel.
3. If required, click **Reset**.

Tip: You can also right-click in the 3D-locked viewer and select **Start Interactive Mode** or **Quit Interactive Mode** from the pop-up menu.

Rotating the Profile

To Rotate the Profile:

1. In the 3D-locked viewer, hold the left button pressed.
2. Rotate the profile in the 3D plane.



1 - The mouse shape in Rotate mode

2 - Before rotating the profile

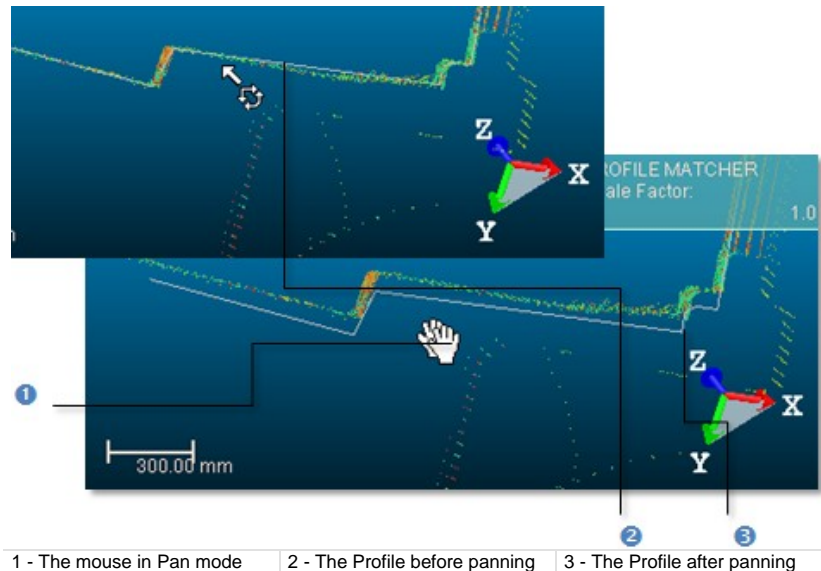
3 - After rotating the profile

Tip: You can also click in the 3D-locked viewer and use the **m** or **M** key to activate or deactivate the **Interactive Mode**.

Panning the Profile

To Pan the Profile:

1. In the 3D-locked viewer, hold the middle button pressed.
2. Move the mouse in any direction to pan the profile.

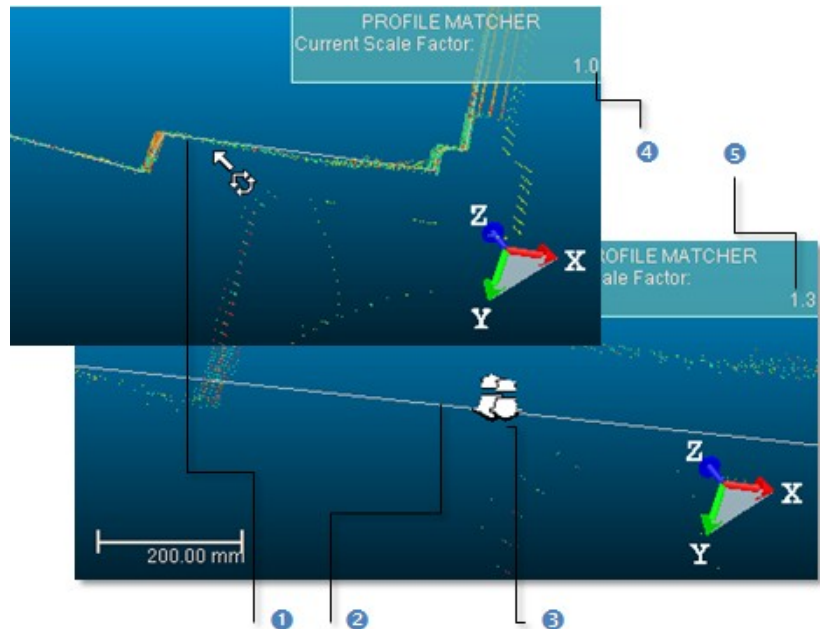


Tip: You can also click in the 3D-locked viewer and use the <Arrow> keys to pan the profile **Up**, **Down**, **Right** and **Left**.

Scaling the Profile

To Scale the Profile:

1. In the 3D-locked viewer, hold the left and middle buttons pressed.
2. Move the mouse forward to scale down the profile.
3. Move the mouse backward to scale up the profile.



- 1 - The profile before scaling down
2 - The profile after scaling down
3 - The mouse shape in Scale mode

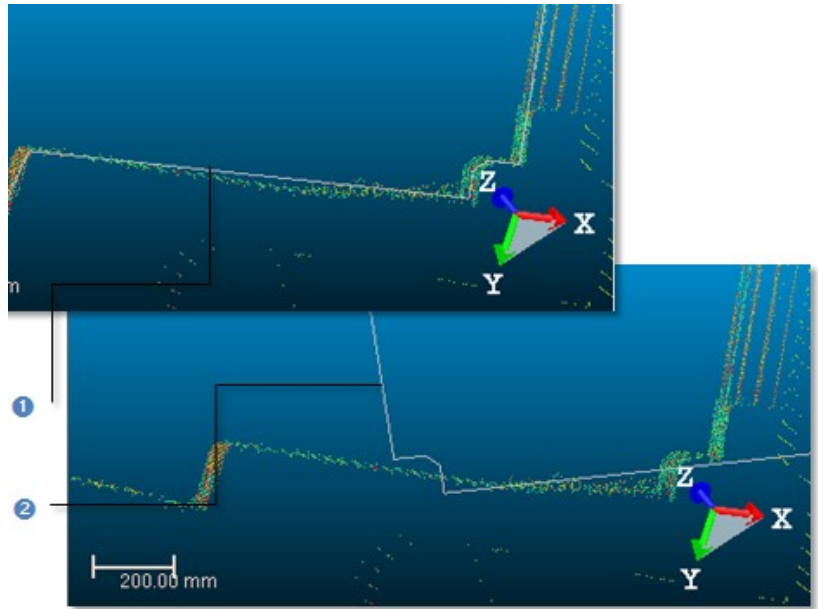
- 4 - The scale factor before scaling down
5 - The scale factor after scaling down

Tip: You can also click in the 3D-locked viewer and use the **+** and **-** keys to scale the profile **Up** and **Down**.

Reversing the Profile

To Reverse the Profile:

- Click **Flip Polyline on Plane** 



1 - Before reversing the profile

2 - After reversing the profile

Tip: You can also right-click in the 3D-locked viewer and select **Flip Polyline on Plane** from the pop-up menu.

Creating the Profile

When you are satisfied with the newly defined profile's position, orientation and scale in the 3D scene, you can create it as a persistent object in the **RealWorks** database. The object is of polyline type and has the same shape as the selected one (the one required to activate the tool). You can create as many profiles as you need without leaving the tool.

To Create the Profile:

1. Click **Create**.
2. Click **Close**.


EasyProfile Tool

The idea behind this new feature is to allow you to easily extract profiles along curbs, pavements, rail lines, cuttings, natural features etc. Profiles are determined by tracking a pre-determined section. A section can be of segment and circle arc based shape. This tool requires a point cloud selection to be able to be activated.

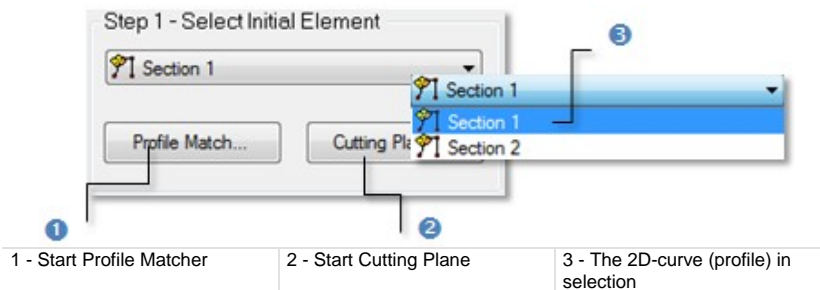
Opening the Tool

A section can be of segment and/or circle arc shape. A section shape is defined based on a 2D-curve (profile). If the project you load contains at least one 2D-curve (profile), you can then select it for defining a section shape. In that case, both the **Profile Matcher** and **Cutting Plane** buttons in **Step 1** of the **EasyProfile™** dialog are available. This means that you can position the existing 2D-curve (profile) within a 3D scene or create a new one. If any 2D-curve (profile) is available, only the **Cutting Plane** button is available. You should then define one. Note that a 2D-curve (profile) - mainly composed of several segments and curve parts, connected or not - can be one imported from AutoCAD or can be previously generated within **RealWorks**.

To Open the Tool:

1. Select an object with cloud property from the **Project Tree**.
2. In the **OfficeSurvey** menu, select **EasyProfile Tool** . The **EasyProfile** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window and is composed of three parts. Each part corresponds to one step in the **EasyProfile** procedure.

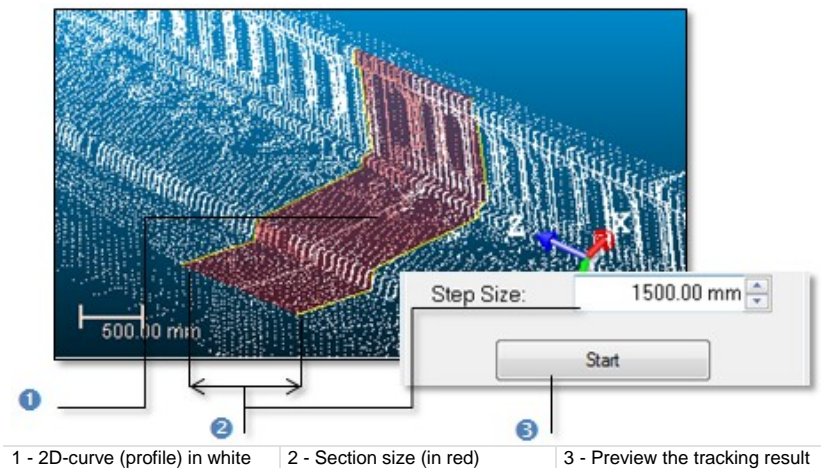


Caution: You can select several point clouds as input of the tool but one of them should not be the **Project Cloud**.

Selecting an Existing Profile

To Select an Existing Profile:

1. Click on the pull-down arrow.
2. Select an existing 2D-curve (profile) from the drop-down list.



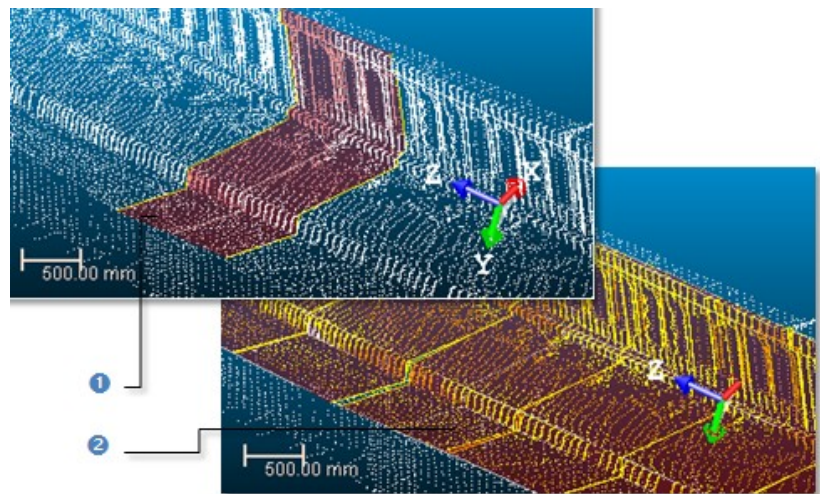
Setting the Section Size

After defining a section shape, you need to define its size. Profile tracking will start from the defined section shape with the specified step size value (that we call **Element**) and will consist of building and propagating in both directions a series of consecutive constrained **Elements** (all based on the first **Element** and all ball-jointed at a pivot point). The tracking will stop of its own when the fitting error between the current (last) **Element** and its points is too large or the number of points in the immediate neighborhood is too small.



To Set the Section Size:

1. Enter a value* in the **Step Size** field.
2. Or use the **Up**  and **Down**  buttons to select a value.
3. Click **Start**.

Points inside each **Element** (shown in yellow) can be hidden by un-checking the **Display Used Points** option. Those outside the **Elements** can be also hidden by un-checking the **Display Remaining Points** option. Once the tracking ends, profiles are then computed from the sequence of built **Elements** and the **start** button becomes dimmed.



1 - First Elements (in red) with profile (in white) | 2 - Propagation of consecutive Elements

Note: The **Delete Elements**  and **Pick to Continue**  buttons are not available before clicking **Start**.


Caution: You cannot enter a value equal to zero or negative in the **Step Size** field.

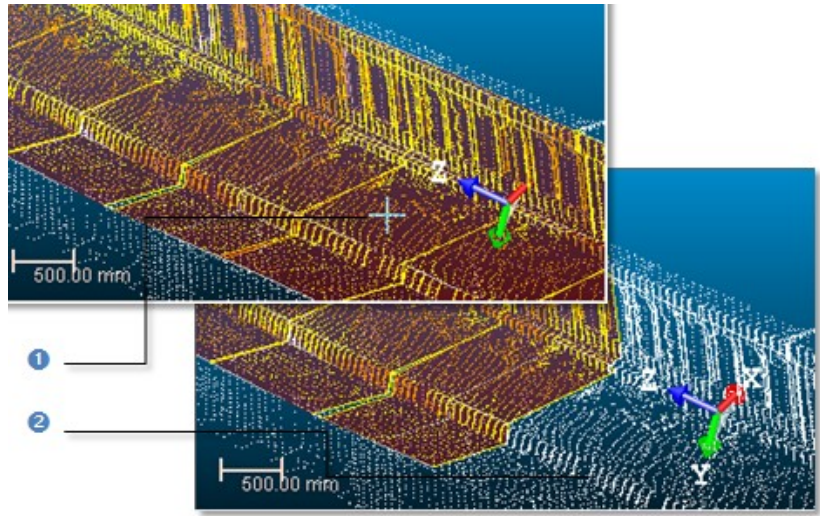
Modifying Built Elements

If the last **Elements** are not correctly fitted, you may decide to delete them. You do this by picking an **Element** in the **3D View**. Those that are after the picked **Element** are deleted. You can decide to jump to a position by picking a point on the selected point cloud. Once a point has been picked, the tracking propagation will start by its own from that point onwards. The side on which the new sequence is appended to the previous sequence is chosen automatically.

Deleting Sections

To Delete Sections:

1. Click **Delete Elements** . The mouse cursor becomes as shown below.
2. Pick an **Element** in the **3D View** window.



1 - The picked Element


2 - The picked Element and those follow are deleted

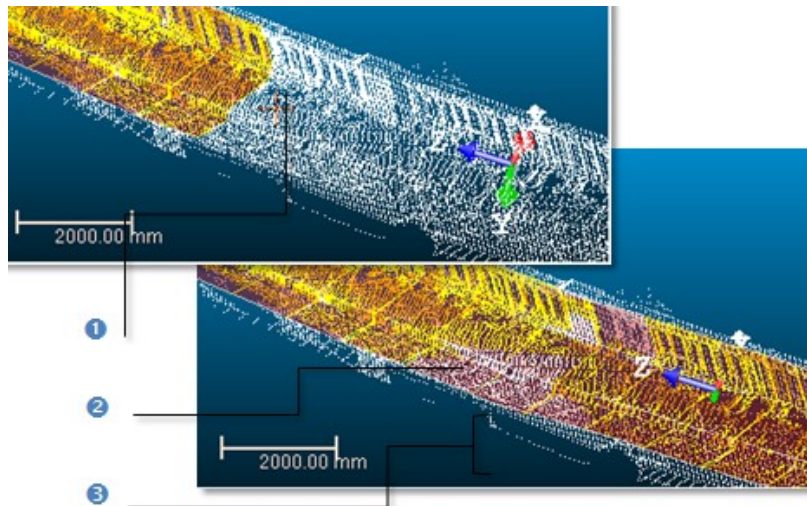
Notes:

- You can undo the deletion by selecting **Undo Delete Elements** from the pop-up menu or by using the following short-cut **Ctrl + Z**.
- You cannot delete the first **Element**; the one which contains the selected 2D-curve and is used for tracking.

Continuing Tracking

To Continue Tracking:

1. Click **Pick to Continue** . The mouse cursor becomes as shown below.
2. If required, change the **Step Size** value.
3. Pick one point on the working cloud.



1 - The picked point

2 - The old sequence is appended to the new sequence

3 - Propagation of another sequence of Elements from the picked point

Note: You can undo continuing tracking by selecting **Undo Continue** from the pop-up menu or by using the following short-cut **Ctrl + Z**.

Creating Profiles

Once you are satisfied with the tracking result, you can save it in the database. A new folder is created and rooted under the current project. This folder contains all computed profiles and a cloud. A profile is always named Prof-xxx where xxx is its order. The cloud - always named **EasyProf-Cloud** - contains points inside the fitted **Elements** used for tracking profiles.

To Create Profiles:

1. Click **Create**.
2. Click **Close**.

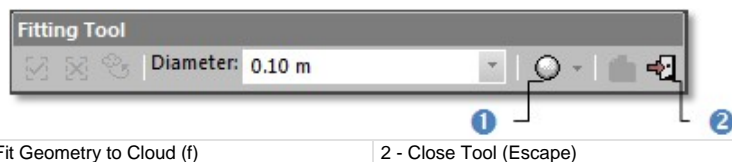
Fitting Tool

This tool is used for fitting a geometry to a set of points. The geometry can be a plane, a sphere or a cylinder. Creating a plane (or a cylinder) can be useful when you need to compare a surface to a geometric model in the **Surface-to-Model Inspection Tool**.

Opening the Tool

To Open the Tool:

1. Select and display one point cloud (or more*) from the **Project Tree**.
2. From the **OfficeSurvey** menu, select **Fitting Tool**. The **Fitting Tool** toolbar appears as well as an information window at the top right corner of the **3D View**.



The information window displays the total number of points in the selected point cloud (**Right Number**) and the number of points after defining a region for fitting (**Left Number**). Before fencing, the **Right Number** and the **Left Number** are both equal.

Tip: You can also click on the **Fitting Tool** icon in the **Tools** toolbar.

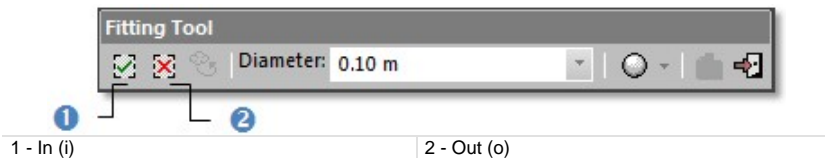
Note: You can fit the whole selected point cloud without fencing as **Fit Geometry to Cloud** is active. If no fence has been defined, a geometry also appears when clicking **Sphere**, **Cylinder**, **Vertical Cylinder**, **Plane** or **Horizontal Plane**. In this case, the geometry fits all points of the selected point cloud and the two numbers of points in the information window remain unchanged.

Caution: (*) You can select several point clouds as input of the tool but one of them should not be the **Project Cloud**.

Fencing a Set of Points

To Fence a Set of Points:

1. Navigate through the 3D scene to find a set of points for which you want to fit with a geometry.
2. Fence this set of points by drawing a polygonal fence.
3. Right-click anywhere in the **3D View**.
4. Select **End Fence** from the pop-up menu. The **In** and **Out** icons become active.



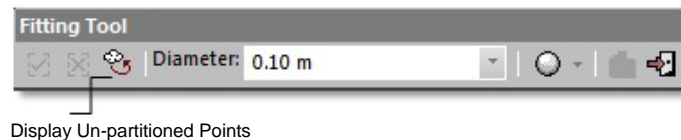
5. Select **In** to keep points inside the fence.
6. Select **Out** to keep points outside the fence.
 - The number of points in the selected point cloud will be diminished from the amount of points used for fitting (in the information window and in the **3D View**).

Tips:

- You can also select **In** (or **Out**) from the pop-up menu or use the related short-cut key **I** (or **O**).
- Instead of selecting **End Fence** from the pop-up menu, press the **Space Bar**.

Notes:

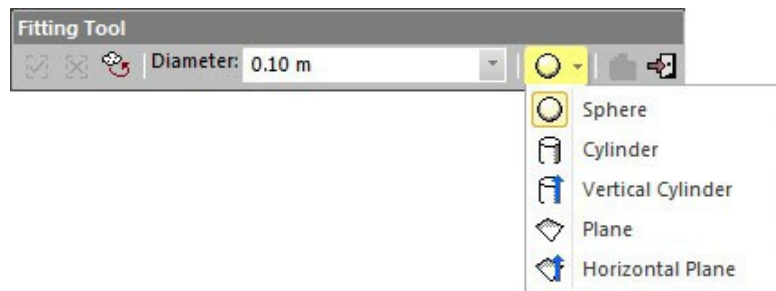
- Pressing **Esc** will undo a closed fence (validated) or a fence in progress (still to be validated).
- After fencing, the **Display Un-partitioned Points** becomes enabled. Clicking on it will reload all points of the selected point cloud.



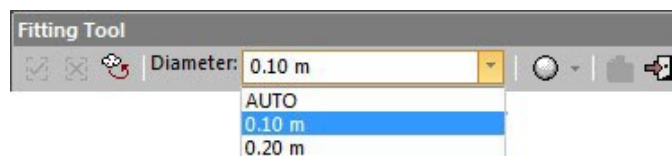
Fitting With a Geometry

To Fit With a Geometry:

1. Click on the **Fit Geometry to Cloud** pull down arrow.
2. Choose a geometry type from the drop-down list. If **Cylinder** (or **Plane**) has been chosen, the **Fitting Tool** toolbar looks as shown below.



3. Click **Cylinder**, **Vertical Cylinder**, **Plane** or **Horizontal Plane**.
 - A **Cylinder**, **Vertical Cylinder**, **Plane** or **Horizontal Plane** appears so that it fits all points inside the fence.
 - If **Vertical Cylinder** has been chosen, the **Cylinder** has a direction of axis parallel to the **Z-Axis**.
 - If **Horizontal Plane** has been chosen, the **Plane** has a normal direction parallel to the **Z-Axis**.
4. If **Sphere** has been chosen, the **Fitting Tool** toolbar looks as shown below. The **Diameter** field becomes enabled.



5. Click on the **Diameter** pull down arrow.
6. Choose between **Auto** and **100 mm***.
7. Or give a diameter value**.
8. Click **Sphere**.
 - If **Auto** has been chosen, a **Sphere** appears so that it fits all points inside the fence.

- If 100 mm (or a user-defined value) has been chosen, a **Sphere** whose diameter is constrained by the chosen (or defined) value appears.

Tips:

- You can fit points directly inside a drawn fence without using **In** (or **Out**).
- You can use the **F** short-cut key instead of clicking the **Fit Geometry to Cloud** icon. The geometry type used for fitting will be the current one.

Notes:

- After fitting, the **Display Unpartitioned Points** icon becomes unavailable.
- Pressing **Esc** will undo the geometry fitting as well as the drawn fence. The fitted geometry disappears from the **3D View**.
- (*) This will correspond to a Trimble **Spherical Target**.
- The information window at the top right corner of the **3D View** displays the **RMS Deviation** value after fitting with a **Cylinder**, **Vertical Cylinder**, **Plane**, **Horizontal Plane** or **Sphere**.
- (**) The value will not be kept anymore if no fitting has been performed.

Creating a Fitted Geometry

If you are satisfied with the fitting result; you can save it as a persistent object in the database. The **Created Fitted Geometry** icon enables to create a fitted entity under the current group behind other objects. The **Create In** feature also creates a fitted entity and puts it under the model group you have to choose from the drop-down pop-up submenu.

To Create a Fitted Geometry:

1. Click **Create Fitted Geometry**.
 - A **Fitted Object** is then created and rooted in the **Models Tree** under the current project and shown in the **3D View**.
 - The selected **Point Cloud** recovers its total number of points (in the information window and in the **3D View**).
2. Click **Close Tool**.

Notes:

- Before leaving the **Fitting Tool**, be sure to create the newly fitted geometry in the database; otherwise it will be lost.
- Pressing **Esc** will close the **Fitting Tool** but will not cancel the created geometry.
- The **Create In** feature can only be selected from the pop-up menu. It is available only if there is at least one group of models under the selection (required to open the **Fitting Tool**). Otherwise, it won't appear in the pop-up menu.


Tip: Instead of clicking on a button in the **Fitting Tool** toolbar, you can also select its corresponding command from the pop-up menu.

3D Inspection Tool

The main function of the tool is to calculate the distance between each point of the compared cloud to a reference surface. The result is a colored cloud where colors stand for a distance (e.g. blue for the closest points and red for furthest points, according to a predefined **ColorBar**).

Opening the Tool

To open the Tool:

1. Select two items from the **Project Tree**.
2. In the **OfficeSurvey** menu, select **3D Inspection Tool** . The **3D Inspection** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window. The first item (by order of selection) is called **Reference** and is displayed in red in the **3D View**. The second item (by order of selection) called **Comparison** is green. You can display (or hide) each of them in the **3D View** by checking (or un-checking) the **Display Reference** (or **Display Comparison**).

You use the **Segmentation** (or **Sampling**) sub-tool for defining a region (or for reducing the number of points) on each of the two selected items (keep checked either **Display Reference** or **Display Comparison**) or on both of them (keep both checked). **Reload Points** will reload points of the item for which the display option has been checked.

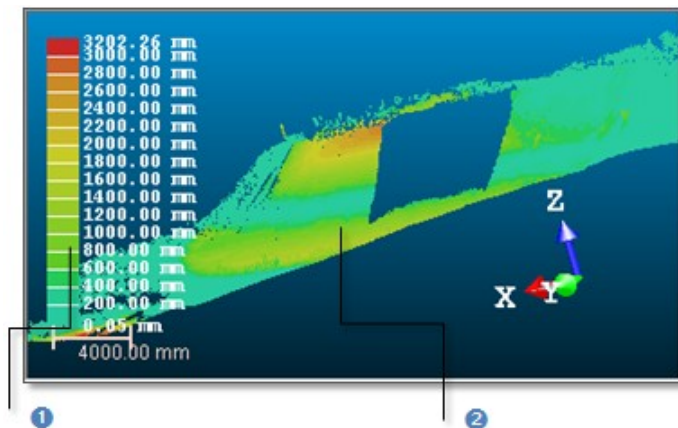
The **Min. Distance** and the **Max. Distance** are both set as "Undefined" in **Step 1**.

Tip: The **3D Inspection Tool** can also be selected from the pop-up menu.

Previewing a 3D Inspection Cloud


To Preview a 3D Inspection Cloud:

1. Click **Preview**.
 - The inspection is done by comparing the **Reference** with the **Comparison**. The result is a 3D inspection cloud inside which points are colored according to the difference (expressed in terms of elevation) between the **Reference** and the **Comparison**. A **ColorBar** located at the right side of the 3D inspection cloud is a scale of elevation values and each color corresponds to an elevation value.
 - The **Min. Distance** and the **Max. Distance** become enabled and display in text the values.
 - The **Preview** becomes dimmed and **Step 2** is enabled.



1 - The ColorBar

2 - The 3D inspection cloud

2. If required, click **Swap Surfaces** .
 - The **Reference** becomes a surface to inspect (**Comparison**) and the **Comparison** swaps for **Reference**.
 - The **Preview** becomes enabled again.

Note: Reload Points will reload the **Comparison**.

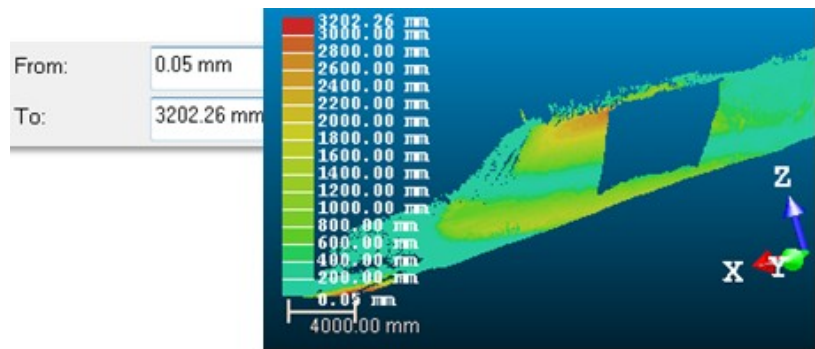
Filtering a 3D Inspection Cloud

The **Min. Distance** and **Max Distance** values of the computed 3D inspection cloud are by-default set as the **From** and **To** values in **Step 2** in the **3D Inspection** dialog. The **From** and **To** values both define a filtering range. This means that points (of the computed 3D inspection cloud) out of the defined range will not be taken into account.

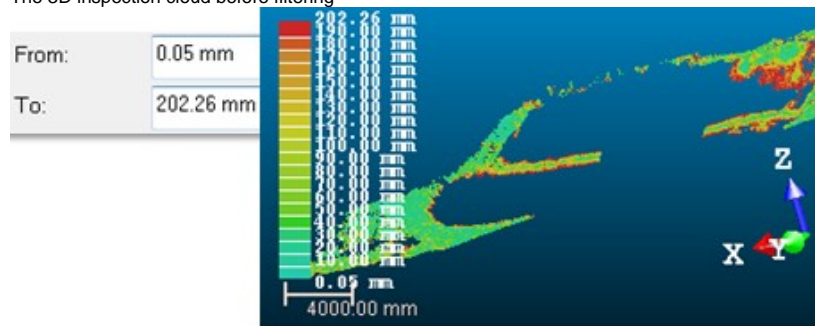
To Filter a 3D Inspection Cloud:

1. Enter a distance value in the **From** field and press **Enter**.
2. Enter a distance value in the **To** field and press **Enter**.

The 3D inspection cloud will be updated according to the defined range as well as its related **ColorBar**.



The 3D inspection cloud before filtering



The 3D inspection cloud after filtering

Notes:

- The unit of measurement for **From** (or **To**) is set to **Meter**, you do not need to enter "m" after the value. You can change the unit of measurement in **Preferences**.
- The **From** value should be less than the **To** value and reversely.

Tip: You can undo the filtering by selecting e.g. **Undo** from the **Edit** menu. Doing that will reload the computed 3D inspection cloud as well as its related **ColorBar**.

Creating a 3D Inspection Cloud

To Create the 3D Inspection Cloud:

- Click **Create**. The **3D Inspection** dialog closes.

An object of **3D Inspection Cloud** type named **Inspection** is created and rooted under the **Models Tree**. This object is by-default selected so that you can launch directly the **3D Inspection Analyzer Tool**; and the two items required for the comparison are both unselected.

You can select a folder from the **Project Tree** under which you want to put the created **3D Inspection Cloud**. You only need to do that before clicking **Create**.

Tips:


- **Close** can also be selected from the pop-up menu.
- If required, select the **Cloud Color** rendering from the **3D View > Rendering** menu to be able to view the 3D inspection cloud with color information.

3D Inspection Analyzer Tool

The **3D Inspection Analyzer Tool** enables to filter a 3D inspection cloud by keeping points between two defined values or to extract connected cloud(s) as individual component(s). This tool can only be used as a standalone tool after selecting a computed 3D inspection cloud.

Opening the Tool

To Open the Tool:

1. Select a 3D inspection cloud from the **Project Tree**.
2. In the **OfficeSurvey** menu, select **3D Inspection Analyzer Tool** .

The **3D Inspection Analyzer** dialog opens as the third tab of the **WorkSpace** window. The selected 3D inspection cloud is by-default displayed in the **3D View**. The number of points inside the 3D inspection cloud is displayed in **Step 1 - Extract Cloud by Range** in the **3D Inspection Analyzer** dialog.





3. Edit a **ColorBar**,
4. Or extract a cloud from the 3D inspection cloud,
5. Or auto-Split a 3D inspection cloud in a cluster of clouds.
6. Or hide (or display) the 3D inspection cloud by checking (or un-checking) the **Display Inspection Cloud** option.

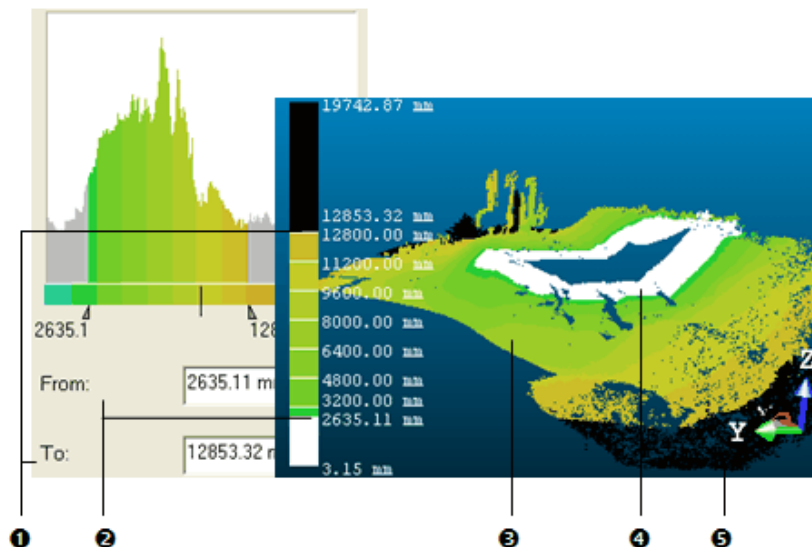
Tip: The **3D Inspection Analyzer Tool** can also be selected from the **Tools** toolbar.

Extracting Clouds From 3D Inspection Clouds


Extracting a cloud from a 3D inspection cloud consists in defining a range (**From** and **To** values). Inspected points inside the defined range will be then kept.

To Extract a Cloud From a 3D Inspection Cloud:

1. Do one of the following:
 - Define the **From** value as follows:
 - Drag and drop the **From** slider.
 - Or key a value in the **From** filed and press **Enter**.
 - Or use the **Up**  (or **Down** ) button.
 - Define the **To** value as follows:
 - Drag and drop the **To** slider.
 - Or key a value in the **From** filed and press **Enter**.
 - Or use the **Up**  (or **Down** ) button.




- | | |
|---|--|
| 1 - The "To" slider and the "To" value in the ColorBar
2 - The "From" slider and the "From" value in the ColorBar
3 - The 3D inspection cloud | 4 - Points of the 3D inspection cloud below the "To" value
5 - Points of the 3D inspection cloud above the "From" value |
|---|--|

The 3D inspection cloud's **Number of Points** (see **Step 1**) as well as its aspect and the color associated to it (see **3D View**) will be then updated. Kept points are those in the defined range; they remain with their own color. The unkept points are those out of the defined range. All are colored in black when they are above the **From** value, and in white when below the **To** value. The **Reload Points**  becomes enabled.

2. Click **Extract**. Unkept points are removed from the **3D View**.



3. If required, define a new range. The **Extract** button becomes again enabled.
4. If required, click **Reload Points** . The 3D inspection cloud takes its initial state (number of points and distances **From** and **To**) as well as the **ColorBar** even if you have done several extractions.

Notes:

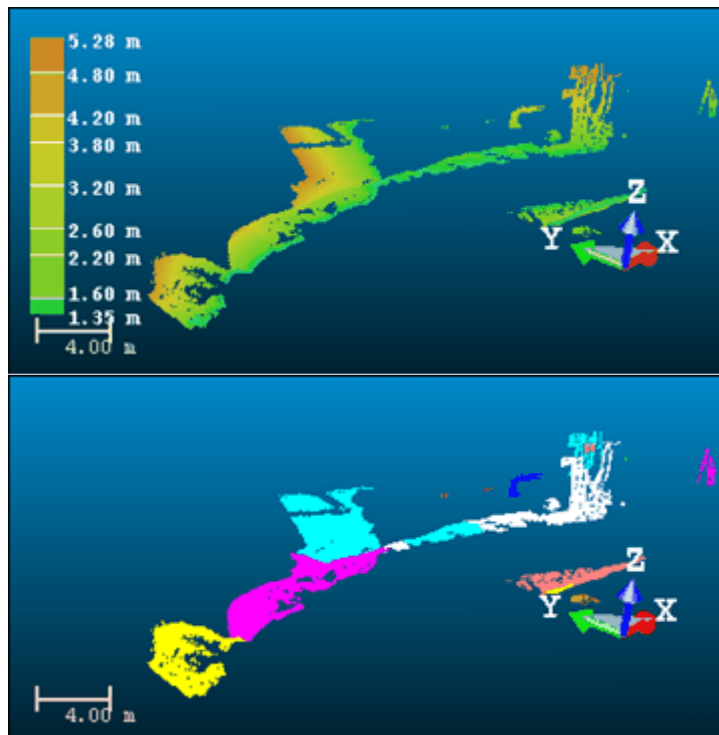
- The unit of measurement for **From** (or **To**) is set to **Meter**, you do not need to enter "m" after the value. You can change the unit of measurement in **Preferences**.
- The **From** value cannot be greater than the **To** value, and reversely.
- The **Extract** button becomes dimmed.
- The **Display Inspection Cloud** option when checked (or unchecked) enables to display (or hide) the cloud in display in the **3D View**. This cloud can be the initial 3D inspection cloud or an extracted cloud.

Auto-Splitting a 3D Inspection Cloud in a Cluster of Clouds

This step is an option. You can split the selected 3D inspection cloud or a cloud extracted from it. If the selected 3D inspection cloud is composed of connected components, this step can help you in splitting these connected components into individual components. Sometimes, you need to first remove connected part(s) from the 3D inspection cloud by extracting (see [Step 1](#)). No parameter is required.

To Auto-Split a 3D Inspection Cloud in a Cluster of Clouds:

1. Click on the **Split to Clusters** button. A dialog displaying the number of components found appears.
2. Click **OK**. The dialog closes.



- Each component is a **Cloud** and each has its own color and displayed in the **3D View**.
- The **Split to Clusters** button becomes dimmed.
- The 3D inspection cloud (filtered or not) is hidden. The **Display Inspection Cloud** option is unchecked by its own.

Tip: If required, apply the **Cloud Color** rendering to be able to view each split component with its own color.

Note: You can split a 3D inspection cloud without extracting a cloud from it. To do that, bypass the **Step 1** in the **3D Inspection Analyzer** dialog.

Creating the Extracted Cloud(s)

To Create the Extracted Cloud(s):

1. Click **Create**. The **3D Inspection Analyzer** dialog remains open.
 - If the 3D inspection cloud has not been split but only filtered, a lonely cloud named as follows **Insp3D-Cloud From value - To value** is created.
 - If the 3D inspection cloud has been split; each component is created as a **Cloud**. Each has its own color and all are put under a folder called **Extracted Clouds**.
2. If required, use the extracted cloud as input for a new extraction (or splitting).
3. Click **Close**. The **3D Inspection Analyzer** dialog closes.

Tip: **Close** can also be selected from the pop-up menu.

CHAPTER 12

Modeling

When you load a file (saved in the **RealWorks** format and in the **Modeling** processing mode) that file will be opened with that processing mode set. When you are outside this processing mode and you need to use it, you have to click on the pull-down arrow in the **Tools** toolbar and choose **Modeling**.

The **Modeling** processing mode is only present in **RealWorks Advanced - Modeler** and **Advanced - Plant**.

Note: The **Export Pipe Center Lines** feature is not present in the **Modeling** processing mode in **RealWorks Advanced - Modeler**.


Geometry Creator Tool

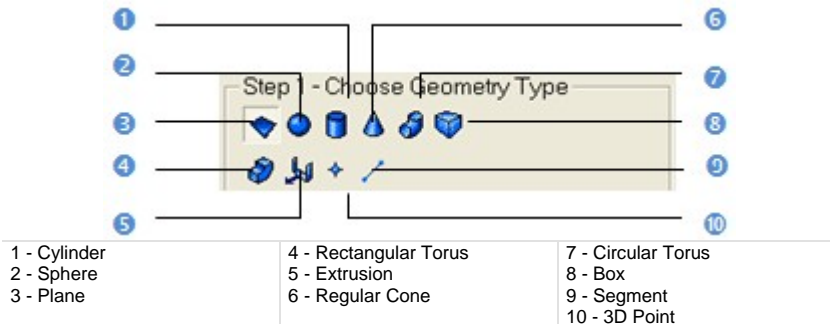
The **Geometry Creator Tool** is for creating a geometry. Ten types are available. You can create a geometry by editing known parameters, picking points or picking entities within displayed objects. All construction methods inside the tool are pure; they are in opposition to those based on point cloud fitting. A created geometry can be used as an entry for the other tools of **RealWorks** like e.g. the **Surface-to-Model Inspection Tool** where models of tunnel are required for comparison.

Opening the Tool

No selection is required to open the **Geometry Creator Tool**. Inside each creation mode, picking a point (free or constrained) does not need to be on displayed objects. The **Picking Parameters** toolbar opens in the 3D constraint mode. When you swap a creation mode for another, a message appears and prompts you to save or not the current geometry (default or drawn one) except for **Plane**, **Circular Torus**, **Extrusion** and **3D Point**.

To Open the Tool:

- From the **Modeling** menu, select **Geometry Creator Tool** . The **Geometry Creator** dialog opens.




This dialog opens as the third tab of the **Workspace** window. The **Plane** type is set by default and the **3D Plane Tool** information box appears at the top right corner of the **3D View**.

Tips:

- You can click the **Geometry Creator Tool** icon in the **Tools** toolbar.
- You can press the **Esc** key to leave the **Geometry Creator Tool**.

Defining a Plane

To Define a Plane:

1. Click the **Plane**  icon. Step 2 of the **Geometry Creator** dialog becomes as shown in [A] in **Examiner/Walkthrough** and in [B] in **Station-Based** mode.



2. Do one of the following:
 - Define a 3D plane (in all navigation modes (**Examiner**, **Walkthrough** and **Station-Based**)).
 - Define a 3D plane in the **Station-Based** mode.
 - Edit parameters.
3. Click **Create**.
4. Click **Close**.

A plane whose name is **OBJECTX** is created and put under the current **Sub-Project** in the **Models Tree**. X is its order. With the **Property** window open, you can edit manually the plane parameters like its **Center** and **Direction of Axis**.

General	
Type	Plane
Name	OBJECT3429
Geometry	
Color of Geometry	RGB(192,192,192)
Center	-6205.71 mm; -2459.39 mm; -6841.61 mm
Direction of Normal	1.00; 0.00; 0.00
Bounds	
N° Holes	0

1 - Editable parameters


2 - Un-editable parameters

Defining a 3D Plane (in all Navigation Modes (Examiner, Walkthrough and Station-Based))


There are three methods available in all navigation modes for interactively defining a plane's orientation and position: **Pick two Screen Points**, **Pick Three Points** and **Pick Two Points**. For the first method, the plane will pass through the line defined by these two points and will be perpendicular to the screen plane. For the second method, the plane will pass through these three points. The third method is to pick two points which define a vector. As a plane is defined by two vectors. Selecting this method will orientate the plane so that the second vectors is parallel to the **Z Axis** (or **Elevation Axis**) of the active coordinate frame.

There are two methods for precisely defining the orientation of a plane: **Axis** and **Pick Entity with Direction**. For the first method, the plane becomes perpendicular to an axis of the active frame. For the second method, the plane's orientation will be aligned to the axis of the picked entity.



To Pick Three Points:

1. Click **Pick three Points** . The **Picking Parameters** toolbar opens in the 3D constraint mode.
2. Pick three points (free or constrained) on displayed objects or not.

To Pick Two Screen Points:

1. Click **Pick two Screen Points** .
2. Pick two points on displayed objects or not.

To Pick Two Points:

1. Click **Pick Two Points**  (in the **X, Y and Z Coordinate System**).
2. Or click  in the **North, East and Elevation Coordinate System**).
3. Pick two points. No need to pick on displayed objects.

To Pick an Entity With a Direction:

1. Click **Pick Entity with Direction** .
2. Pick an entity with direction.

To Make Perpendicular to an Axis:


1. Choose among **X Axis** , **Y Axis**  and **Z Axis**  (in the **X, Y and Z Coordinate System**).

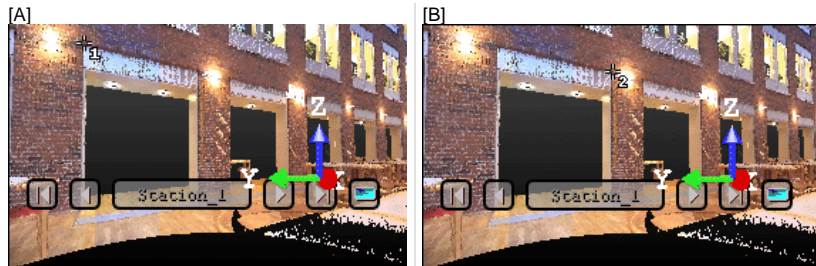
2. Or choose among North Axis N , East Axis E and Elevation Axis El (in the North, East and Elevation Coordinate System).

Defining a 3D Plane in the Station-Based Navigation Mode

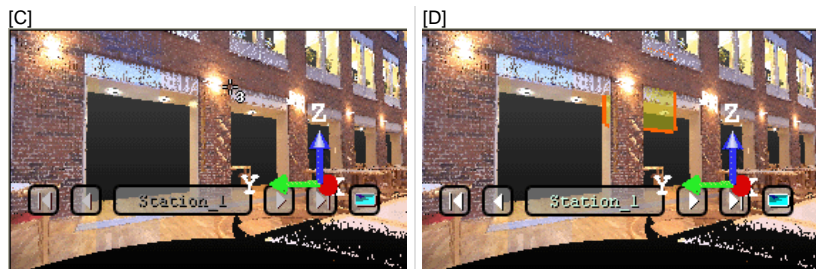
There are ten methods available in the **Station-Based** mode for defining a 3D plane. Three are specific to that mode: two are based on two screen points and a 3D point and one on three screen points and a 3D point. The other methods are already described in the **Examiner** (or **Walkthrough**) mode.

To Define a Vertical Plane by Picking two Screen Points and one 3D Point:


1. Click on the  button. The cursor becomes as shown in [A].
2. Pick a point anywhere on the screen. The cursor becomes as shown in [B].



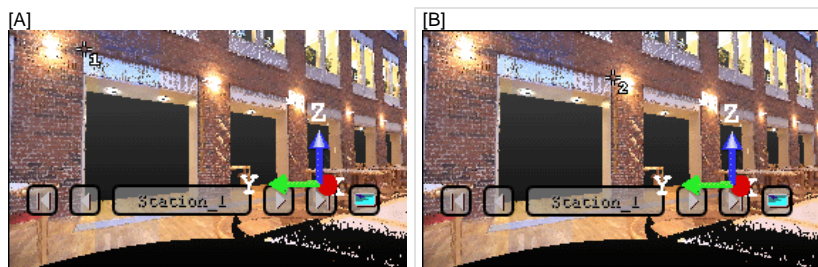
3. Pick another point anywhere in the **3D View** (on the displayed point cloud or not). The cursor becomes as shown in [C] and the **Picking Parameters** toolbar appears in 3D constraint mode.
4. Pick the last point anywhere in the **3D View** (only on the displayed point cloud). A vertical plane appears with the third picked point as center.



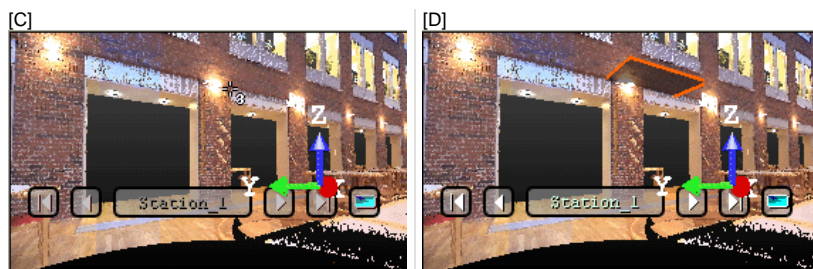
To Define a Horizontal Plane by Picking Two Screen Points and one 3D Point:

1. Click on the  button. The cursor becomes as shown in [A].


- Pick a point anywhere on the screen. The cursor becomes as shown in [B].



- Pick another point on the screen. These two points will define the orientation of the first axis of the horizontal plane's frame. The cursor becomes as shown in [C] and the **Picking Parameters** toolbar appears in 3D constraint mode.
- Pick the last point in 3D (on a cloud point, a measured point or a geometry). This point defines the height of the vertical plane. A horizontal plane appears with the third picked point as center.

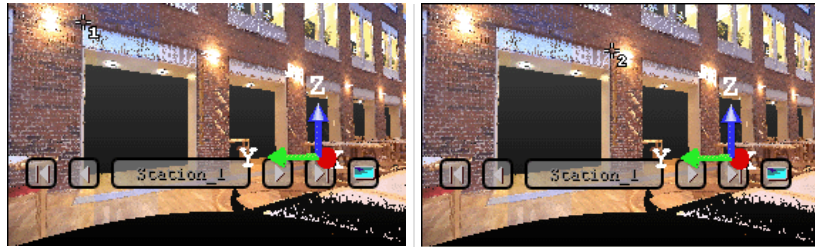


To Define a Plane by Picking Three Screen Points and one 3D Point (this Enables to Define a Plane with any Orientation):

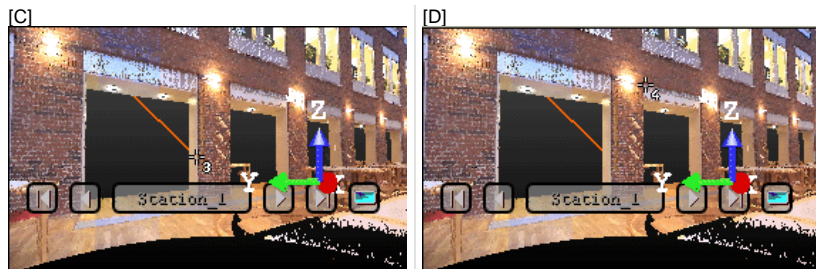
- Click on the  button. The cursor becomes as shown in [A].
- Pick a point anywhere on the screen. The cursor becomes as shown in [B].

[A]

[B]






3. Pick another point on the screen so that the two points represent a horizontal segment in the 3D space. These two points define the orientation of a horizontal segment drawn on the final plane. The cursor becomes as shown in [C].
4. Pick another on the screen so that the previous point and this new one represent the steepest slope direction of the final plane. The cursor becomes as shown in [D] and the **Picking Parameters** toolbar appears in 3D constraint mode.



5. Pick the last point in the **3D View** (only on the displayed point cloud). The three first picked points - which are not collinear (not lying on the same line) - draw a 3D plane; the fourth picked point is its center.



Tip: ,  and  can be selected from the pop-up menu.



Editing Parameters

To Edit Parameters:


1. Enter a direction in the **Normal** field.
2. Enter a point's position in the **Position** field.

Defining a Sphere

To Define a Sphere:

1. Click the **Sphere**  icon. **Pick Two Points**  is set by default, the **Picking Parameters** toolbar appears in the 3D constraint mode and the cursor is in the picking mode.
2. Do one of the following:
 - Pick two points*.
 - Edit parameters.
3. Click **Create**.
4. Click **Close**.

A sphere whose name is **OBJECTX** is created and put under the current **Sub-Project** in the **Models Tree**. **X** is its order. With the **Property** window open, you can edit manually the sphere parameters like its **Center**, **Diameter**, **Direction of Axis** and **Distance Between Extremities**.

General	
Type	Sphere
Name	OBJECT227
Geometry	
Color of Geometry	 RGB(192,192,192)
Center	16408.29 m; -88716.95 m; 182.44 m
Diameter	2.00 m
Bounds	
Direction of Axis	0.00; 0.00; 1.00
Distance between Extremities	2.00 m
Extremity 1	16408.29 m; -88716.95 m; 181.44 m
Extremity 2	16408.29 m; -88716.95 m; 183.44 m

1

2

3


1 - Auto-computed parameters
2 - Editable parameters
3 - Un-editable parameters

Notes:

- The **Direction of Axis** is assumed to be aligned with the **Z-Axis** of the active coordinate frame.
- (*) In the picking mode, pressing **Esc** (or selecting **Cancel Picking** from the pop-up menu) first leaves that mode and then makes appeared a sphere at the middle of the **3D View**. Its parameters are displayed in the dialog. If you choose another geometry type, a dialog appears and prompts you to create the current geometry (or not).

Picking Two Points

To Pick Two Points:

1. If required, click **Pick Two Points** . The **Picking Parameters** toolbar opens in 3D constraint mode.
2. Pick a point. This point will be the first extremity of a sphere.
3. Pick another point. This point will be the second extremity of a sphere.



Editing Parameters

To Edit Parameters:


1. Enter a 3D position in the **Center** field.
2. Enter a value in the **Radius** field.

Defining a Cylinder

To Create a Cylinder:

1. Click **Cylinder** . **Pick Three Points** * is set by default, the **Picking Parameters** toolbar appears in the 3D constraint mode and the cursor is in the picking mode.
2. Do one of the following:
 - Pick three points*.
 - Edit parameters.
3. Click **Create**.
4. Click **Close**.

A cylinder whose name is **OBJECTX** is created and put under the current **Sub-Project** in the **Models Tree**. **X** is its order. With the **Property** window open, you can edit manually the cylinder parameters like its **Center**, **Pipe Diameter**, **Length**, **Direction of Axis**, **Direction of Bound 1** and **Direction of Bound 2**.

General	
Type	Cylinder
Name	OBJECT225
Geometry	
Color of Geometry	 RGB(192,192,192)
Center	16399.23 m; -88708.65 m; 203.28 m
Pipe Diameter	2.00 m
Length	2.00 m
Direction of Axis	0.41; 0.88; -0.22
Bounds	
Direction of Bound 1	-0.41; -0.88; 0.22
Direction of Bound 2	0.41; 0.88; -0.22
Extremity 1	16398.82 m; -88709.53 m; 203.50 m
Extremity 2	16399.64 m; -88707.77 m; 203.06 m

1 - Editable parameters

2 - Un-editable parameters


Notes:

- The **Direction of Bound 1** and **Direction of Bound 2** are aligned with the **Direction of Axis**.

- (*) In the picking mode, pressing **Esc** (or selecting **Cancel Picking** from the pop-up menu) first leaves that mode and then makes appeared a vertical cylinder at the middle of the **3D View**. Its parameters are displayed in the dialog. If you choose another geometry type, a dialog appears and prompts you to create the current geometry (or not).

Picking Three Points

To Pick Three Points:

1. If required, click the **Pick Three Points**  icon. The **Picking Parameters** toolbar opens in 3D constraint mode.
2. Pick a point. This point will be the first extremity of a cylinder.
3. Pick another point. This point will be the second extremity of a cylinder.
4. Pick a third point. This point will define the pipe radius.



Editing Parameters

To Edit Parameters:


1. Click on the pull down arrow.
2. Choose between **Two Points and Radius** and **Point, Direction and Length**.
 - If **Two Points and Radius** has been chosen:
 - a) Enter a point's coordinates in the **First Point** field.
 - b) Enter another point's coordinates in the **Last Point** field.
 - c) Enter a distance value in the **Radius** field.
 - If **Point, Direction and Length** has been chosen:
 - a) Enter a direction in the **Direction** field.
 - b) Give a point's position in the **Position** field.
 - c) Give a distance value in the **Radius** field.

Defining a Regular Cone

To Define a Regular Cone:

1. Click the **Regular Cone**  icon. **Pick Three Points**  is set by default, the **Picking Parameters** toolbar appears in the 3D constraint mode and the cursor is in the picking mode.
2. Do one of the following:
 - Pick three points*.
 - Edit parameters.
3. Click **Create**.
4. Click **Close**.

A regular cone whose name is **OBJECTX** is created and put under the current **Sub-Project** in the **Models Tree**. **X** is its order. With the **Property** window open, you can edit manually the regular cone parameters like its **Center**, **Diameter at Base**, **Diameter at Top**, **Distance Between Extremities** and **Direction of Axis**.

General	
Type	Regular Cone
Name	OBJECT224
Geometry	
Color of Geometry	 RGB(192,192,192)
Center	16418.60 m; -78710.21 m; 179.60 m
Diameter at Base	4.00 m
Diameter at Top	1.00 m
Distance between Extremities	3.00 m
Direction of Axis	0.42; 0.80; 0.44
Bounds	
Extremity at Base	16419.23 m; -78709.02 m; 180.26 m
Extremity at Top	16417.97 m; -78711.40 m; 178.94 m


1 - Auto-computed parameters
2 - Editable parameters
3 - Un-editable parameters

Notes:

- The **Center** and **Direction of Axis** (automatically computed based-on the other parameters) can be modified.
- (*) In the picking mode, pressing **Esc** (or selecting **Cancel Picking** from the pop-up menu) first leaves that mode and then makes appeared a vertical regular cone at the middle of the **3D View**. Its parameters are displayed in the dialog. If you choose another geometry type, a dialog appears and prompts you to create the current geometry (or not).

Picking Three Points

To Pick Three Points:

1. If required, click the **Pick Three Points**  icon.
2. Pick a point. This point will be the base extremity of a regular cone.
3. Pick another point. This point will be the top extremity of a regular cone.
4. Pick a third point. This point will define the top radius.

The drawn regular cone parameters are displayed in the dialog.


Editing Parameters

To Edit Parameters:


1. Enter a 3D position in the **First Point** field.
2. Enter another 3D position in the **Last Point** field.
3. Give a radius for the first point in the **Top Radius** field.
4. Give a radius for the last point in the **Base Radius** field.

Defining a Circular Torus

To Define a Circular Torus:

1. Click the **Circular Torus**  icon.
2. Do one of the following:
 - Pick two points(1).
 - Align to join an existing cylinder(2).
 - Align to join to two secant cylinders of same radius(3).
 - Edit parameters.
3. Click **Create**.
4. Click **Close**.

A circular torus whose name is **OBJECTX** is created and put under the current project in the **Models Tree**. **X** is its order. With the **Property** window open, you can edit manually the circular torus parameters like its **Center**, **Pipe Diameter**, **Center Line Diameter**, **Direction of Axis** and **Bend Angle**.

General	
Type	Circular Torus
Name	OBJECT3419
Geometry	
Color of Geometry	 RGB(192,192,192)
Center	-7506.58 mm; 141.34 mm; 171.29 mm
Pipe Diameter	0.27 m
Center Line Diameter	0.54 m
Direction of Axis	0.00; 0.06; 1.00
Bounds	
Bend Angle	90.00 °
Extremity 1	-7238.79 mm; 141.06 mm; 171.19 mm
Extremity 2	-7506.30 mm; 408.72 mm; 156.45 mm

1 - Editable parameters


2 - Un-editable parameters

Notes:

- (1) With no constraint applied, the created circular torus is of closed shape (the **Bend Angle** equal to 360 degrees).
- (2) With one constraint applied, the created circular torus is an open regular torus, its **Bend Angle** is equal to 90 degrees.
- (3) With two constraints applied, the created circular torus is an open regular torus, its **Bend Angle** is equal or less than 90 degrees.

Picking Two Points


To Pick Two Points:

1. Click **Pick Two Points** . The **Picking Parameters** toolbar opens in 3D constraint mode.
2. Pick a point. This point will be the center of a torus.
3. Pick another point. This point will define the **Pipe Radius + Center Line Radius** distance.

Note: Press **Esc** or select **Cancel Picking** from the pop-up menu to leave the **Pick Two Points** mode. A 3D torus appears at the middle of the **3D View** and its parameters are shown in the **Geometry Creator** dialog.


Aligning to Joint to an Existing Cylinder

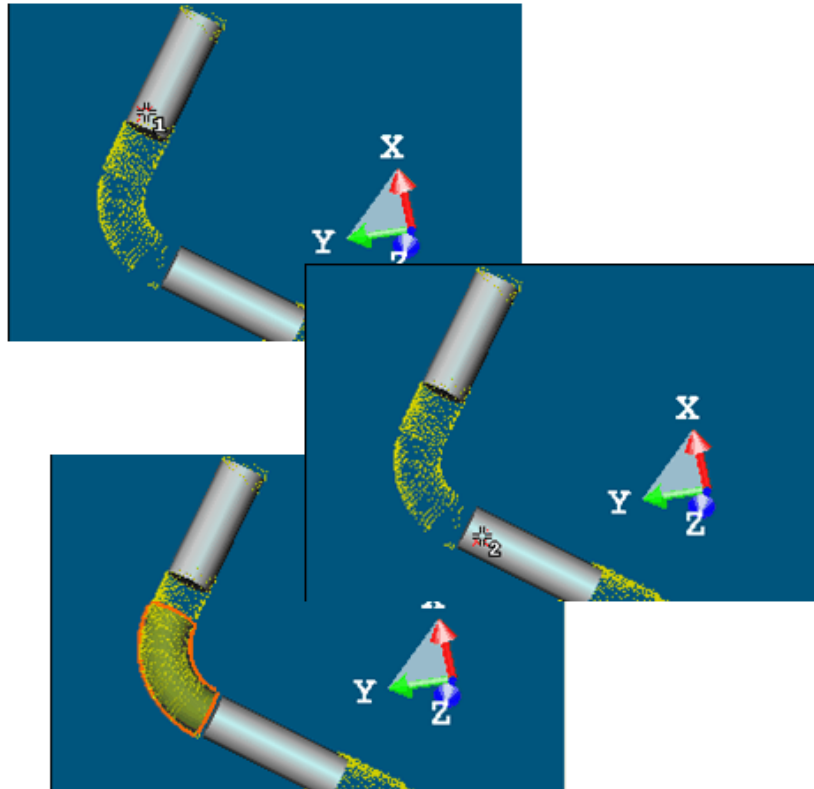
To Align to Join to an Existing Cylinder:

1. Click the **Align to Join to an Existing Cylinder**  icon.
2. Pick an existing cylinder.

Aligning to Join to two Secant Cylinders of Same Radius

To Align to Join to two Secant Cylinders of Same Radius:


1. Click the **Align to Join to Two Secant Cylinders of Same Radius**  icon.
2. Pick an existing cylinder.
3. Pick another existing cylinder. The result is null if the two picked cylinders are not secant or do not have the same diameter.



Note: A warning appears if the two picked cylinder axes are parallel.

Defining a 3D Point

To Define a 3D Point:



1. Click the **3D Point**  icon. A **3D Point** at the middle of the **3D View**.
2. Do one of the following:
 - Pick a point*
 - Pick three secant planes*.
 - Pick a plane and a segment*.
 - Pick an entity with center*.
 - Project a 3D Point on a plane*.
 - Edit parameters.
3. Click **Create**.
4. Click **Close**.

A **3D Point** whose name is **OBJECTX** is created and put under the current project in the **Models Tree**. **X** is its order. With the **Property** window open, you can only edit manually the **3D Point's Center**.

Note: (*) In the picking mode, pressing **Esc** (or selecting **Cancel Picking** from the pop-up menu) first leaves that mode and then makes appeared a **3D Point** at the middle of the **3D View**. Its parameters are displayed in the dialog. If you choose another geometry type, a dialog appears and prompts you to create the current geometry (or not).

Picking a Point


To Pick a Point:

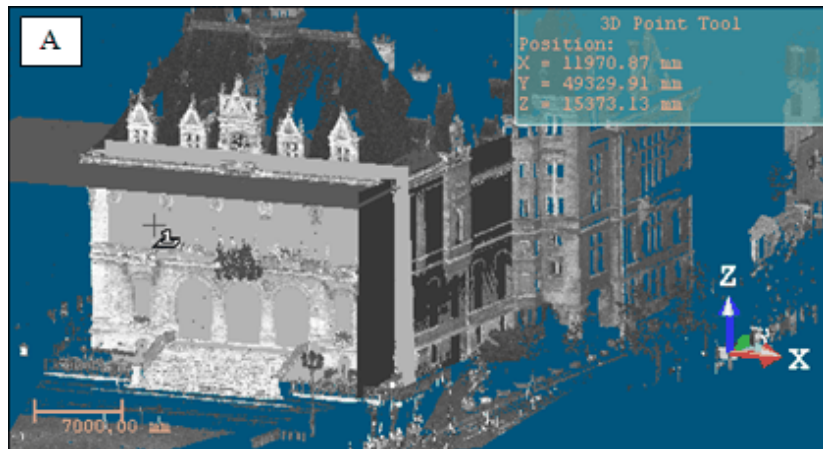
1. Click the **Pick Point**  icon. The cursor takes the following shape , the initial **3D Point** is hidden and the **Picking Parameters** toolbar opens in the 3D constraint mode.
2. Pick a point (free or constrained) on displayed objects (or not).

Picking Three Planes

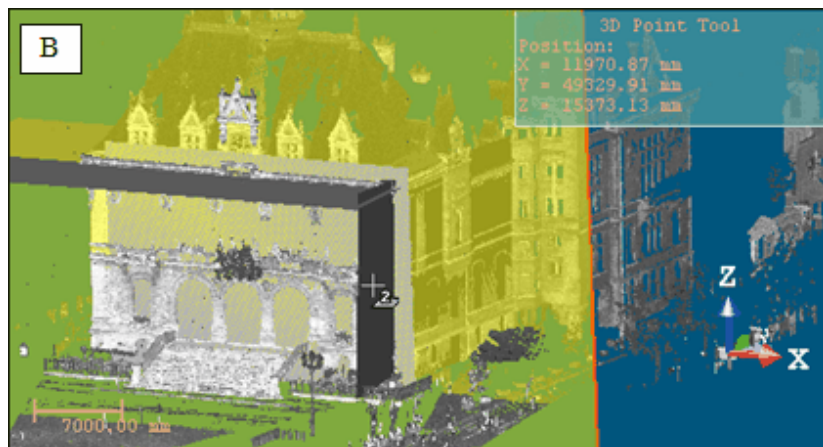
If three planes are not parallel two-by-two (with no coplanar normals), then they will intersect (cross over) somewhere at a point.

To Pick Three Planes:

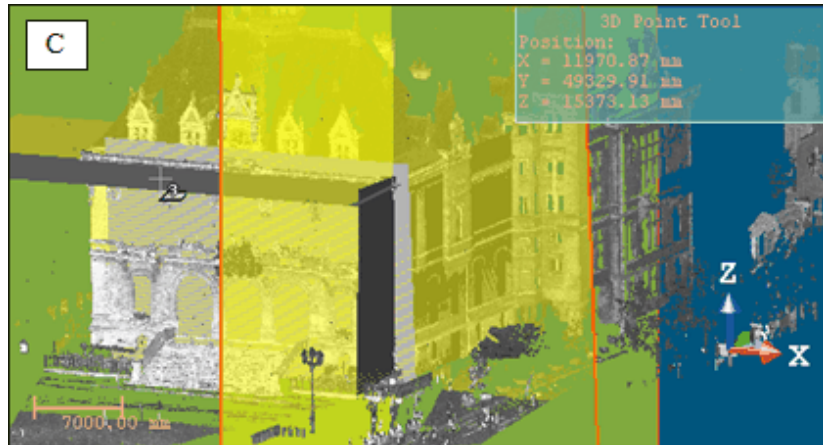
1. Click the **Pick Three Planes**  icon. The initial **3D Point** is hidden in the **3D View** and the cursor becomes as shown in [A].



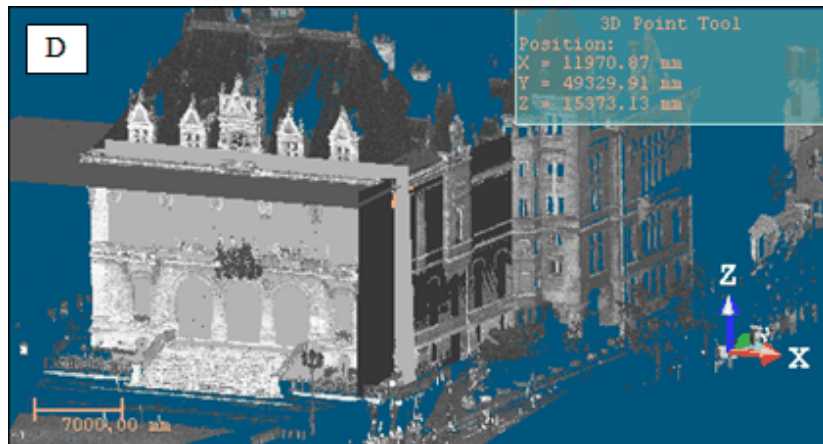
2. Pick a plane. A red frame with a yellow background appears upon the picked plane. The cursor becomes as shown in [B].



- Pick another plane. Another red frame with a yellow background appears upon the picked plane. The cursor becomes as shown [C].




- Pick again another plane. A 3D Point (the intersection of the three picked planes) appears. [D]

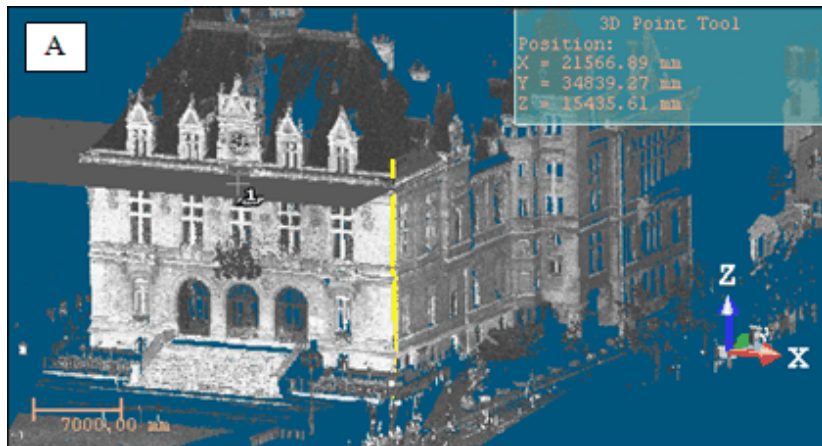


Picking a Plane and a Segment

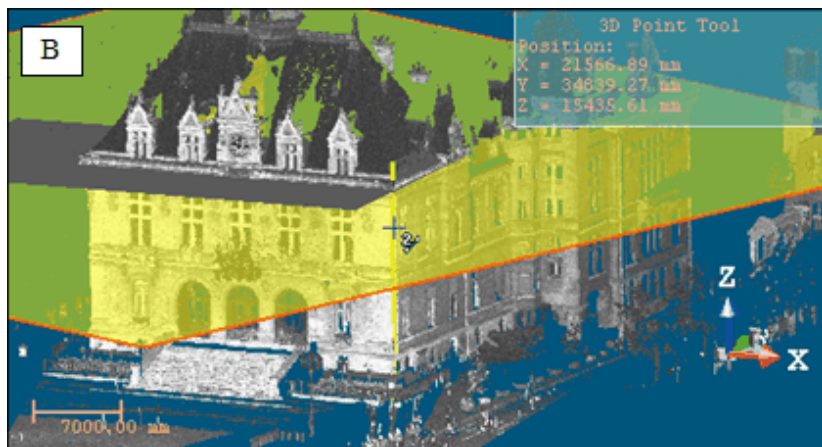
In this creation mode, you need to have a plane and a segment - both intersected at a point.

To Pick a Plane and a Segment:

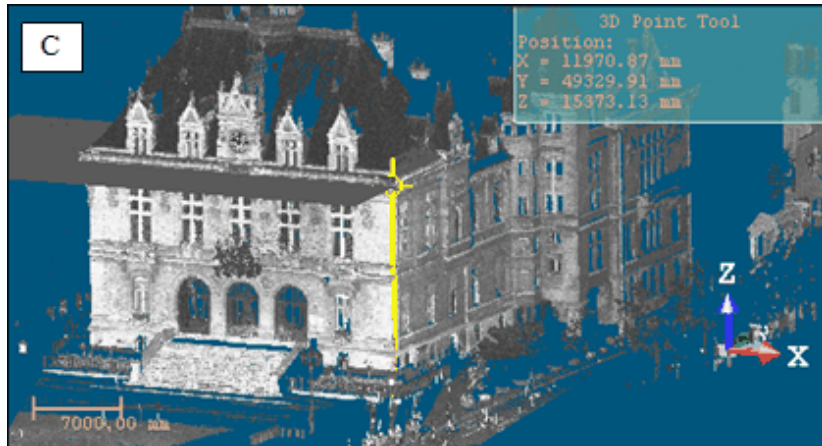
1. Click the **Pick Plane and Segment**  icon. The initial **3D Point** disappears from the **3D View** and the cursor takes the shape shown in [A].



2. Pick a plane. A red frame with a yellow background upon the picked plane appears. The cursor takes the shape shown in [B].





3. Pick a segment. The plane and the segment intersect at a 3D Point [C].



Picking an Entity with Center

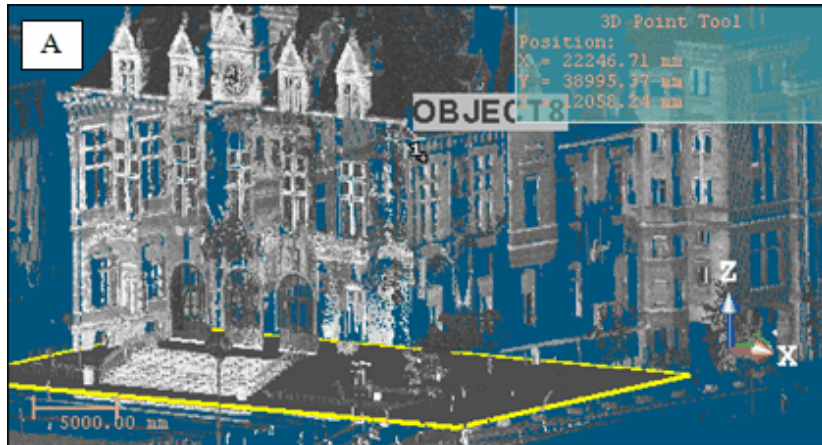
To Pick an Entity With Center:

1. Click the **Pick Entity with Center**  icon. The cursor takes the following shape . The initial **3D Point** disappears from the **3D View**.
2. Pick an object of any type having a center (except point cloud, mesh, etc.).

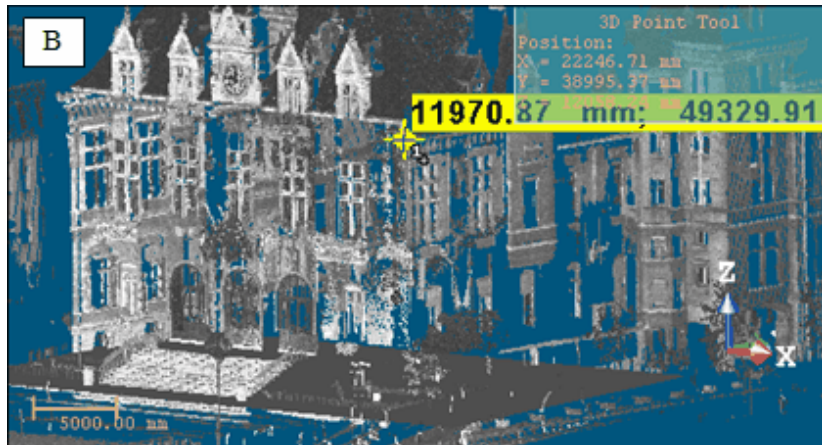
Projecting a 3D Point on a Plane

To Project a 3D Point on a Plane:

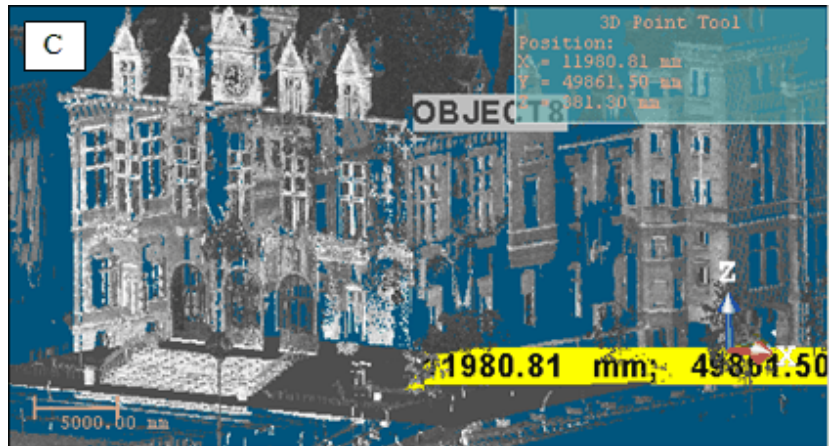
1. Click the **Project 3D Point on Plane**  icon. The initial 3D point disappears from the **3D View** and the cursor takes the shape shown in [A].



2. Pick a **3D Point**. A yellow label with the following text “**Point to Project on Plane**” appears over the picked point [B].




3. Pick a plane. A 3D point appears on that plane [C].



Picking Two Axial Entities

To Pick Two Axial Entities:

1. Click the **Pick Two Axial Entities** .
2. Pick an axial geometry in the **3D View**.
3. Pick another axial geometry in the **3D View**.

A 3D Point in yellow appears in the **3D View**. Its parameter (**Center**) is updated in the dialog in the **Position** field.

Note: A warning message appears if no intersection is found.


Editing Parameters

To Edit Parameters:

- Enter a point position in the **Position** field.

Define a Segment

To Define a Segment:


1. Click the **Segment**  icon.
2. Do one of the following:
 - Pick two points*,
 - Pick two planes*,
 - Pick an axial geometry*,
 - Edit parameters.
3. Click **Create**.
4. Click **Close**.

A segment whose name is **OBJECTX** is created and put under the current **Sub-Project** in the **Models Tree**. **X** is its order. With the **Property** window open, you can edit manually the segment parameters like its **Center**, **Direction of Axis** and **Length**.

Note: (*) In the picking mode, pressing **Esc** (or selecting **Cancel Picking** from the pop-up menu) first leaves that mode and then makes appeared a vertical segment at the middle of the **3D View**. Its parameters are displayed in the dialog. If you choose another geometry type, a dialog appears and prompts you to create the current geometry (or not).

Picking Two Points

To Pick Two Points:


1. Click the **Pick Two Points**  icon. The **Picking Parameters** toolbar opens in the 3D constraint mode and the cursor is in the picking mode.
2. Pick a point. This point will be the first extremity of a segment.
3. Pick another point. This point will be the second extremity of a segment.

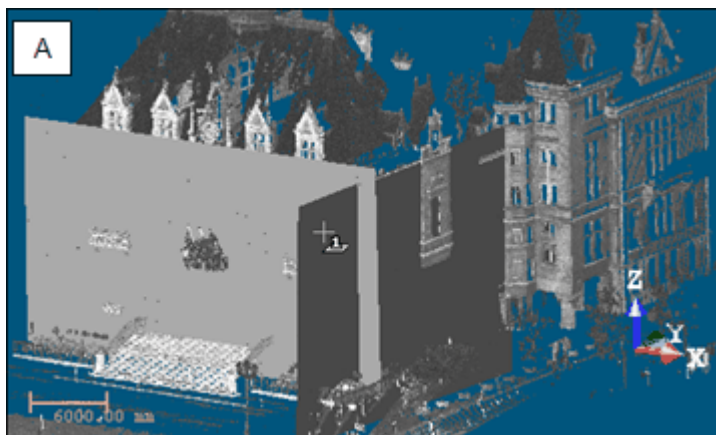
The drawn segment parameters are displayed in the dialog.

Picking Two Planes

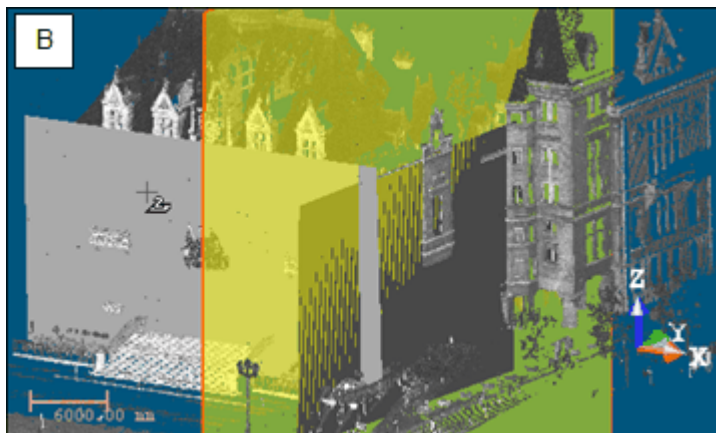
If two planes are not parallel, then they will intersect (cross over) each other somewhere at a line.

To Pick Two Planes:

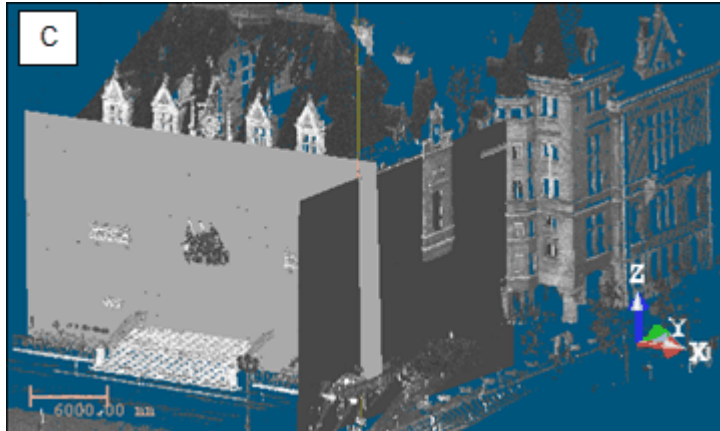
1. Click the **Pick Two Planes**  icon. The **Picking Parameters** toolbar opens in 3D constraint mode. The cursor becomes as shown in [A].



2. Pick a plane. A red frame with a yellow background upon the picked plane appears. The cursor takes the shape shown in [B].




3. Pick another plane.



A segment in yellow appears in the **3D View**. Its parameters (**Direction of Axis** and **Center**) are updated in the dialog respectively in the **Direction** and **Point** fields.

Picking an Axial Geometry

To Pick an Axial Geometry:

1. Click the **Pick Axial Geometry**  icon.
2. Pick an axial geometry in the **3D View**.

A segment in yellow appears in the **3D View**. Its parameters (**Direction of Axis** and **Center**) are updated in the dialog respectively in the **Direction** and **Point** fields.

Editing Parameters


To Edit Parameters:

1. Click on the pull down arrow.
2. Choose between **Two Points** and **Point, Direction and Length**.
 - If **Two Points** has been chosen:
 - a) Enter a 3D position in the **First Point** field.
 - b) Enter another 3D position in the **Last Point** field.
 - If **Point, Direction and Length** has been chosen:
 - a) Enter a direction in the **Direction** field.
 - b) Enter a 3D position in the **Point** field.
 - c) Enter a distance value in the **Length** field.


Creating an Extruded Entity

An **Extrusion** is a tool for creating a three-dimensional geometry of free shape from 2D profiles.

To Create an Extruded Entity:

1. Click the **Extrusion**  icon. The **Drawing Tool** and **Picking Parameters** (in 3D constraint mode) toolbars appear.
2. Draw or select a polyline.
3. Enter a distance value in the **Length** field.
4. Click **Create**.
5. Click **Close**.

An **Extrusion** whose name is **OBJECTX** is created and put in the **Models Tree**. **X** is its order. With the **Property** window open, you can edit manually the segment parameters like its **Center**, **Length**, **Direction of Axis**, **Direction of Bound 1** and **Direction of Bound 2**.

General	
Type	Extrusion
Name	OBJECT13
Geometry	
Color of Geometry	 RGB(192,192,192)
Center	6715.97 mm; 50507.18 mm; 12329.92 mm
Length	1000.00 mm
Direction of Axis	0.67; -0.73; 0.13
Bounds	
Direction of Bound 1	-0.67; 0.73; -0.13
Direction of Bound 2	0.67; -0.73; 0.13
Extremity 1	6381.46 mm; 50872.91 mm; 12263.97 mm
Extremity 2	7050.47 mm; 50141.45 mm; 12395.87 mm

1
2

1 - Editable parameters
2 - Un-editable parameters

Creating a Box

To Create a Box:

1. Click the **Box**  icon. **Step 2** of the **Geometry Creator** dialog becomes as shown in [A] in **Examiner/Walkthrough** and in [B] in **Station-Based** mode.



2. In **Examiner** (or **Walkthrough**) mode, do one of the following:
 - Pick an entity with direction,
 - Project a box on a plane,
 - Pick the bottom left corner of a box,
 - Pick four screen points,
 - Edit parameters.
3. In **Station-Based** mode - in addition to the five construction methods above - do one of the following:
 - Define a horizontal face by picking one 3D point, then four screen points (two horizontal directions and depth),
 - Define a vertical face by picking one 3D point, the four screen points (horizontal direction, vertical direction and depth).
4. Click **Create**.
5. Click **Close**.

A box whose name is **OBJECTX** is created and put under the current project in the **Models Tree**. **X** is its order. With the **Property** window open, you can edit manually the box parameters like its **Center**, **Width**, **Depth**, **Height**, etc.


General	
Type	Box
Name	OBJECT3419
Geometry	
Color of Geometry	■ RGB(192,192,192)
Center	-15397.53 mm; 3825.42 mm; -7494.35 mm
Width	400.00 mm
Depth	200.00 mm
Height	100.00 mm
Direction of Width	1.00; 0.01; -0.00
Direction of Depth	-0.01; 1.00; 0.00
Direction of Height	0.00; -0.00; 1.00
Bounds	
Extremity 1 for width	-15597.52 mm; 3823.40 mm; -7493.79 mm
Extremity 2 for width	-15197.55 mm; 3827.43 mm; -7494.91 mm
Extremity 1 for depth	-15396.53 mm; 3725.42 mm; -7494.62 mm
Extremity 2 for depth	-15398.54 mm; 3925.41 mm; -7494.09 mm
Extremity 1 for height	-15397.68 mm; 3825.55 mm; -7544.35 mm
Extremity 2 for height	-15397.39 mm; 3825.29 mm; -7444.35 mm

1 - Editable parameters (in black)

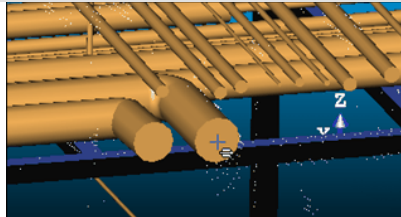
2 - Un-editable parameters (in gray)

Picking an Entity With a Direction

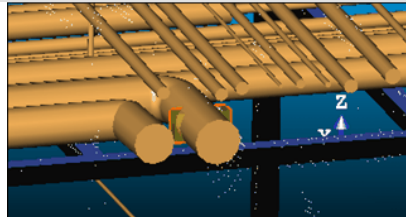
To Pick an Entity With a Direction:

1. Click **Pick Entity With Direction** . The cursor becomes as shown in [A].
2. Pick an entity. A 3D box appears [B].

[A]



[B]




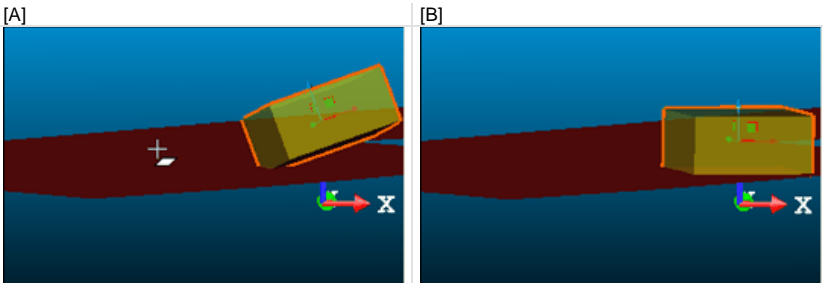
- It has the picked entity's **Direction of Axis** as **Normal Z** direction (also called **Direction of Height** in the **Property** window),
- Its center is the same as the picked entity's.

Note: Press **Esc** (or select **Cancel Picking** from the pop-up menu) to leave the picking mode.

Projecting a Box onto a Plane

To Project a Box in a Plane:

1. Click **Stick to Plane** . The cursor becomes as shown in [A].
2. Pick a plane. The selected box is projected on the picked plane [B]. The bottom side (of the box) lies on the plane.



The **Direction of Normal** (of the plane) [C] and the **Direction of the Height** (also called the **Normal Z*** of the box) [D] are parallel but not necessarily in the same direction. In the example below, both are opposite.



[C]	[D]
Properties	Properties
<div> <div>General</div> <div>TypePlane - Fitted</div> <div>NameOBJECT498</div> </div> <div> <div>Geometry</div> <div>Color of Geometry■ RGB(255,0,0)</div> <div>Center84.95 m; 206.89 m; -</div> <div>Direction of Normal-0.01; 0.01; -1.00</div> </div> <div> <div>Bounds</div> <div>N° Holes0</div> </div>	<div> <div>General</div> <div>TypeBox</div> <div>NameOBJECT1979</div> </div> <div> <div>Geometry</div> <div>Color of Geometry■ RGB(192,192,192)</div> <div>Center77.40 m; 180.76 m;</div> <div>Width8.94 m</div> <div>Height4.27 m</div> <div>Depth8.01 m</div> <div>Direction of Width0.92; 0.39; -0.00</div> <div>Direction of Height0.01; -0.01; 1.00</div> <div>Direction of Depth-0.39; 0.92; 0.01</div> </div> <div> <div>Bounds</div> </div>

Notes:

- Press **Esc** (or select **Cancel Picking** from the pop-up menu) to leave the picking mode.
- (*) In the **X, Y, Z Coordinate System**.

Picking the Bottom Left Corner of a Box

To Pick The Bottom Left Corner of a Box:


1. Click **Pick Bottom Left Corner of Box** . The cursor becomes as shown  and the **Picking Parameters** toolbar in 3D constraint mode opens.
2. Pick a point. A 3D Box appears. Its bottom left corner is the point you picked.

Notes:

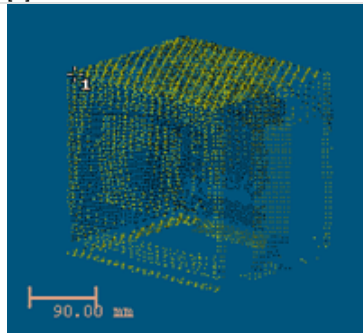
- Picking should be on displayed objects.
- Press **Esc** (or select **Cancel Picking** from the pop-up menu) to leave the picking mode.

Picking Four Screen Points

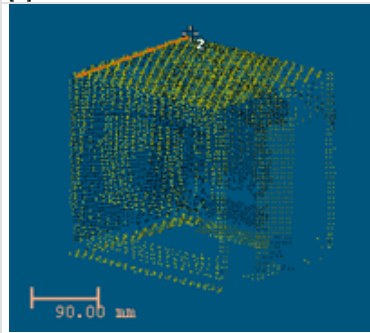
To Pick Four Screen Points:

1. Click the **Pick Four Screen Points**  icon. The cursor becomes as shown in [A] and the **Picking Parameters** toolbar in 3D constraint mode opens.
2. Pick the first screen point*. The cursor takes the shape shown in [B].
3. Move your mouse. A segment in orange links the first screen point to the cursor.

[A]

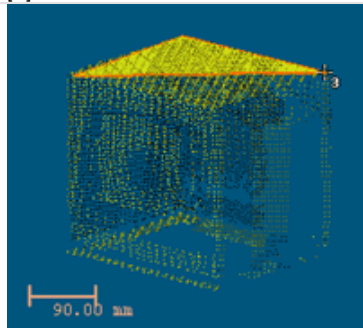


[B]

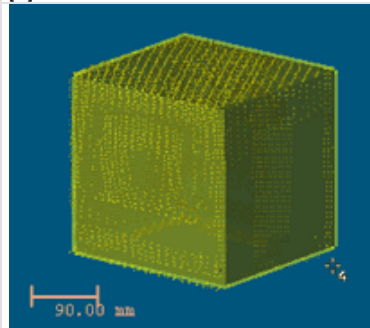


4. Pick the second screen point*. The cursor then takes the shape shown in [C].
5. Move your mouse. Two other segments in orange link the first and second screen points previously picked to the cursor. The two picked points and the cursor draw a triangular plane.
6. Pick the third screen point*. The cursor then takes the shape shown in [D] and the triangular plane changes to a rectangular plane.

[C]



[D]




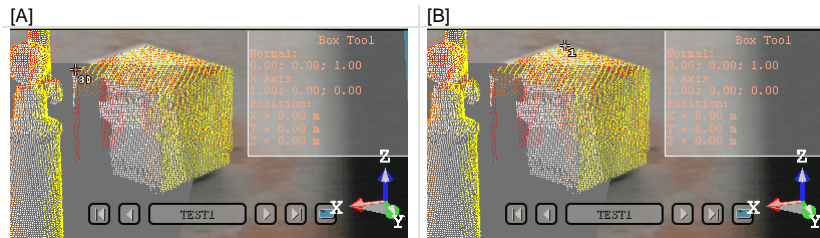
7. Move again your mouse (from Up to Down or vice versa). A 3D Box (with an orange frame) appears.
8. Pick the fourth screen point* to complete the 3D Box.

Note: Picking should be on displayed points.

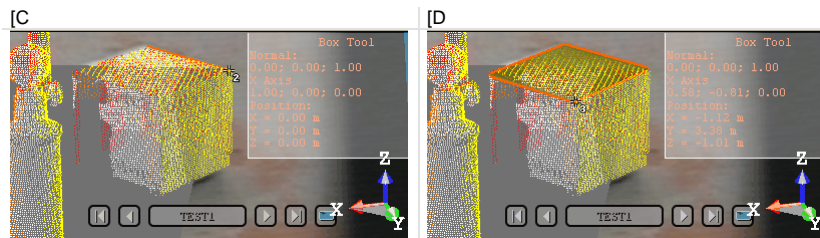
Defining Horizontal Face By Picking One 3D Point, Then Four Screen Points (Two Horizontal Directions and Depth)

To Define a Horizontal Face by Picking One 3D Point, then Four Screen Points (Two Horizontal Directions and Depth):

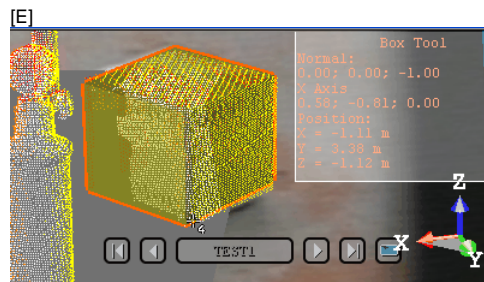
1. Click the **Define Horizontal Face By Picking One 3D Point, Then Four Screen Points (Two Horizontal Directions and Depth)**  icon. The cursor becomes as shown in [A].
2. Pick a 3D Point. Picking should be on displayed points. The cursor takes then the shape shown in [B].
3. Pick the first screen point. Picking doesn't need to be on displayed points.



4. Move your mouse. The cursor then becomes as shown in [C]. A segment in orange links the first screen point to the cursor. This segment can be vertical or horizontal.
5. Pick the second screen point, not necessary on displayed points.
6. Move your mouse again. The cursor then becomes as shown in [D]. A horizontal plane (with an orange frame) appears.
7. Pick the third screen point, not necessarily on displayed points.




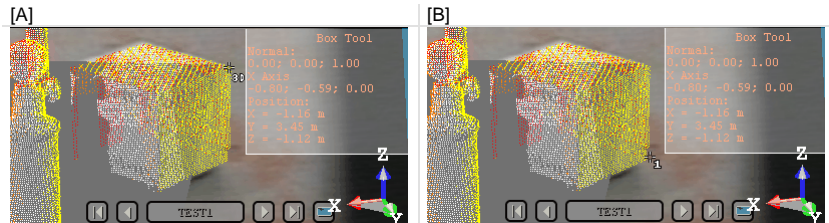
8. Move your mouse again. The cursor then becomes as shown in [E]. A 3D box (with an orange frame) appears.
9. Pick the fourth screen point to complete the 3D box.



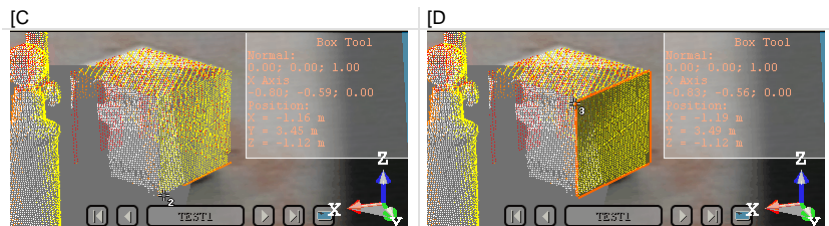
Defining Vertical Face By Picking One 3D Point, Then Four Screen Points (Horizontal Direction, Vertical Direction and Depth)

To Define a Vertical Face by Picking one 3D Point, then Four Screen Points (Horizontal Direction, Vertical Direction and Depth):

1. Click the **Define Vertical Face By Picking One 3D Point, Then Four Screen Points (Horizontal Direction, Vertical Direction and Depth)** . The cursor becomes as shown in [A].
2. Pick a point. Picking should be on displayed points. The cursor then takes the shape shown in [B].
3. Pick the first screen point. Picking doesn't need to be on displayed points.

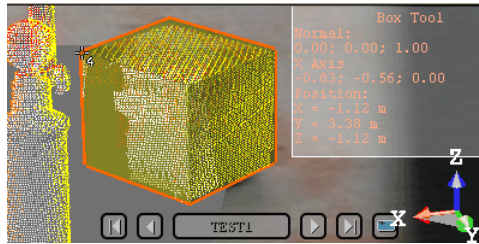


4. Move your mouse. The cursor then becomes as shown in [C]. A segment in orange links the first screen point to the cursor. This segment should not be vertical but horizontal.
5. Pick the second screen point, not necessary on displayed points.
6. Move your mouse again. The cursor then becomes as shown in [D]. A vertical plane (with an orange frame) appears.
7. Pick the third screen point, not necessarily on displayed points.



8. Move again your mouse. The cursor becomes then as shown in [E]. A 3D box (with an orange frame) appears.
9. Pick the fourth screen point to complete the 3D box.

[E]



Editing Parameters


To Edit Parameters:

1. Enter a point position in the **Center** field.
2. Enter a direction in the **Normal Z** field.
3. Enter a direction in the **Vector X** field.
4. Enter a distance value in the **Depth (X)** field.
5. Enter a distance value in the **Width (Y)** field.
6. Enter a distance value in the **Height (Z)** field.


Note: After updating a parameter, remember to press **Enter**. The current box (the one in display in the **3D**) will change its shape according to the updated parameter.

Creating a Rectangular Torus

To Create a Rectangular Torus:

1. Click the **Rectangular Torus**  icon.
2. Do one of the following:
 - Align to join to two existing secant boxes of same section,
 - Edit parameters.
3. Click **Create**.
4. Click **Close**.

A rectangular torus whose name is **OBJECTX** is created and put under the current **Sub-Project** in the **Models Tree**. **X** is its order. With the **Property** window open, you can edit manually the rectangular torus parameters like its **Center**, **Width**, **Height**, etc.

General	
Type	Rectangular Torus
Name	OBJECT3435
Geometry	
Color of Geometry	 RGB(192,192,192)
Center	-7519.56 mm; 2264.20 mm; -3876.14 mm
Width	100.00 mm
Height	100.00 mm
Center Line Diameter	2.00 m
Inner Diameter	0.95 m
Outer Diameter	1.05 m
Direction of Axis	0.00; 0.00; 1.00
Bounds	
Bend Angle	360.00 °
Extremity 1	-6519.56 mm; 2264.20 mm; -3876.14 mm
Extremity 2	-6519.56 mm; 2264.20 mm; -3876.14 mm

1


2

1 - Editable parameters (in black)2 - Un-editable parameters (in gray)

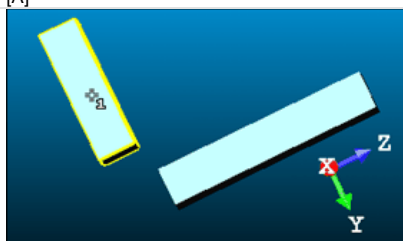
Note: If no constraints have been applied; the created rectangular torus is of open shape (the **Bend Angle** is less than 360 degrees). If no constraint has been applied, the created rectangular torus is of closed shape (the **Bend angle** is equal to 360 degrees).

Aligning to Join Two Existing Secant Boxes of Same Section

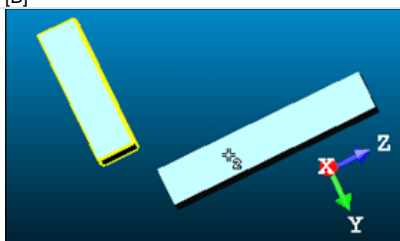
To Align to Join to Two Existing Secant Boxes of Same Section:

1. Click the **Align to Join to Two Existing Secant Boxes of Same Section**  icon. The cursor becomes as shown in [A].
2. Pick a first box. The cursor takes the shape shown in [B].

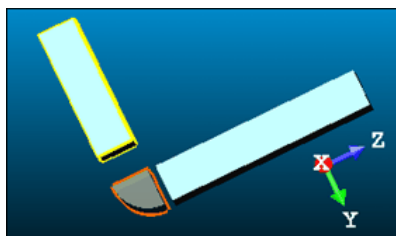
[A]



[B]



3. Pick another box. If the two boxes are secant and have the same section, a rectangular torus appears.



- Its **Direction of Normal** (also called **Direction of Axis**) is parallel to the two boxes' **Direction of Normal** (also called **Direction of Width**).
 - Its **Bend Angle** is equal to the angle drawn by the two boxes' **Direction of Height**.
 - Its **Outer Diameter** is equal to the two boxes' **Depth**.
4. If the two picked boxes are not secant; the "This constraint cannot be activated because the two boxes are not secant" warning message appears. Click **OK**. The warning message closes and the **Align to Join to Two Existing Secant Boxes of Same Section** constraint is left.

5. If the two picked boxes do not have the same section, the "There is too much indetermination to activate this constraint: two boxes are identical, one of them is a cube or they have no common face" warning message appears. Click **OK**. The warning message closes and the **Align to Join to Two Existing Secant Boxes of Same Section** constraint is left.

Tips:

- If required, make the two boxes secant using the **Make Secant to a Box (With Same Section)** constraint in the **Geometry Modifier Tool**.
- If required, modify manually the two boxes' parameters (like **Center**, **Width**, **Height**, **Depth**, **Directions**, etc.) in the **Property** window to make sure that both are secant and have the same section.

Editing Parameters

To Edit Parameters:

1. Give a direction in the **Normal** field.
2. Enter a 3D position in the **Center** field.
3. Enter a distance in the **Center Line Radius** field.
4. Enter a distance in the **Width** field.
5. Enter a distance in the **Height** field.

Note: The **Normal**'s direction is called **Direction of Axis** in the **Property** window.


Cloud-Based Modeler Tool

The **Cloud-Based Modeler Tool** allows you to model geometries of the following shapes: **Plane**, **Sphere**, **Cylinder**, etc. Modeling can only be cloud-based. In this case, a selection as input data (**Project Cloud** or pure point cloud) is required and the modeling is done by fitting points.

Opening the Tool

An object having the point cloud and geometry representations cannot be selected as entry for the **Cloud-Based Modeler Tool**; you need to first delete the geometry representation from that object. The selection can be multiple.

To Open the Tool:

1. Select one point cloud* (or more*) from the **Project Tree**.
2. From the **Modeling** menu, select **Cloud-Based Modeler Tool** . The **Cloud-Based Modeler** dialog opens.

This dialog opens as the third tab of the **WorkSpace** window. The **Segmentation Tool** is set by default and its toolbar appears. The input point cloud is called **Cloud Data**. The information box at the top right corner of the **3D View** displays the **RMS Deviate** and **Number of Points** information (both are initially "Undefined") for the **Cloud-Based Modeler Tool** and the **Number of Points** information for the **Segmentation Tool**. The **RMS (Root Mean Square) Deviate** corresponds to the standard deviation between points used for fitting and the fitted geometry.

Tip: You can also click **Cloud-Based Modeler Tool** in the **Tools** toolbar.



Note: (*) If the selected point cloud is **On** before starting the tool, it automatically tilts to **Off**. We advise you to maintain it **Off**.

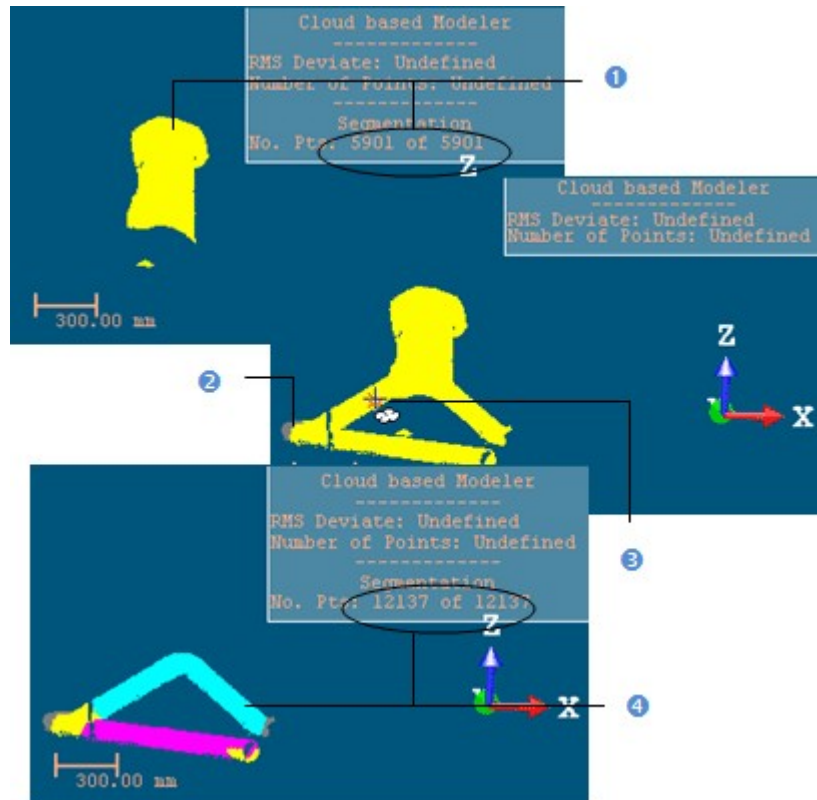
Caution: (**) You can select several point clouds as input of the tool but one of them should not be the **Project Cloud**.

Selecting a New Cloud Data

The **Set New Cloud Data** is for swapping the default **Cloud Data** (not necessary the one selected before starting the **Cloud-Based Modeler Tool**) for another one. You cannot choose and set a subset of the default **Cloud Data** as the new **Cloud Data**; you need to choose a different point cloud.

To Select a New Cloud Data:

1. Select another point cloud from the **Project Tree**.
2. Display the selected point cloud in the **3D View**.
3. If required, hide the default **Cloud Data** (by clicking  in the **Cloud-Based Modeler** dialog).
4. Click the **Set New Cloud Data**  icon. The cursor becomes as shown below and the information box related to the **Segmentation Tool** disappears from the **3D View**.
5. Pick a point on the selected point cloud. It becomes the new **Cloud Data**. The **Segmentation** information box appears again with the new cloud data number of points.



1 - The initial Cloud Data
2 - The newly selected Point Cloud

3 - The cursor in the Set New Cloud Data mode
4 - The new Cloud Data

Note: (*)The **Hide Cloud** icon becomes **Display Cloud** after clicking on it.

Defining a Set of Points on the Cloud Data

Frequently, the **Cloud Data** contains many points; you need to decimate them before doing the fitting. You may also decide to fit a geometry just on a part of it. To do these, you can use the **Segmentation** and the **Sampling** sub-tools.



1 - The Segmentation Tool

2 - The Sampling Tool

After segmenting/sampling the **Cloud Data**, **Keep Only Displayed Cloud in Cloud Data** and **Delete Displayed Cloud from Cloud Data** (respectively for keeping/deleting points in/from the current **Cloud Data** (after decimating/fencing) and **Reload Points** become active.





1 - Reload Points

2 - Keep Only Displayed Cloud in Cloud Data

3 - Delete Displayed cloud from Cloud Data

Keeping Only Displayed Cloud

To Keep Only Displayed Data:

1. Draw a fence on the **Cloud Data**.
2. Use the **In**  (or **Out** )* icon to keep points inside (or outside) the fence.
3. Or sample the **Cloud Data**.
4. Click the **Keep Only Displayed Cloud in Cloud Data**  icon. Points displayed in the **3D View** inside are kept.




Notes:

- **Reload Points** is only for reloading points of the current **Cloud Data** after sampling or segmenting.
- (*) You can skip the step of keeping **In** (or **Out**). In that case, points inside the fence are kept.

Tip: Outside the segmentation mode, you can select the **Segmentation Tool** from the pop-up menu or use its related short-cut key (**S**).

Deleting Displayed Cloud

To Delete the Displayed Cloud:

1. Draw a fence on the **Cloud Data**.
2. Use the **In**  (or **Out** )* icon to keep points inside (or outside) the fence.
3. Or sample the **Cloud Data**.
4. Click the **Delete Displayed Cloud from Cloud Data**  icon. Points displayed in the **3D View** are unkept (removed from the **Cloud Data**).

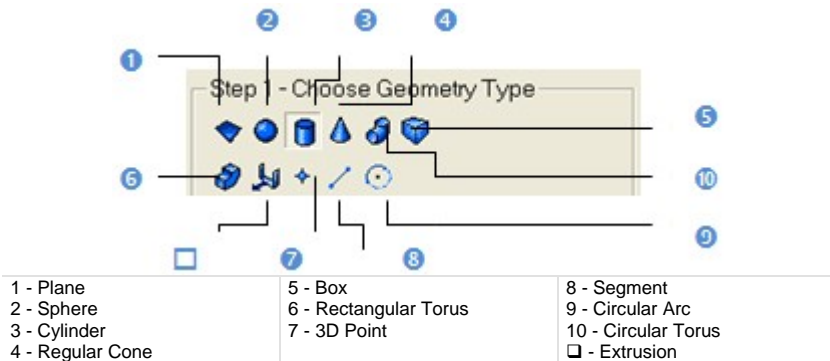
Notes:

- **Reload Points** is only for reloading points of the current **Cloud Data** after sampling or segmenting.
- (*) You can skip the step of keeping **In** (or **Out**). In that case, points inside the fence are unkept.

Tip: Outside the segmentation mode, you can select the **Segmentation Tool** from the pop-up menu or use its related short-cut key (**S**).

Choosing a Geometry Type

There are ten geometry types and one construction method (**Extrusion**). When the **Cloud-Based Modeler** dialog appears, the type which comes first is the one you have selected during the previous use of that tool. To change the geometry type, click an icon in the **Cloud-Based Modeler** dialog. **Extrusion** is a tool for creating a three-dimensional geometry of free shape from 2D profiles.









Tip: You can also select a geometry type from the pop-up menu.

Using Constraints

This optional step is for applying constraints to objects under construction. Check the **Use Constraint** option to make this step appear. Constraints can be assumed as limits imposed on objects and they vary according to the geometry type selected in **Step 1**. You cannot apply two constraints of the same type. The applied constraint you find in the constraint list - see hereafter - is always the last applied one.

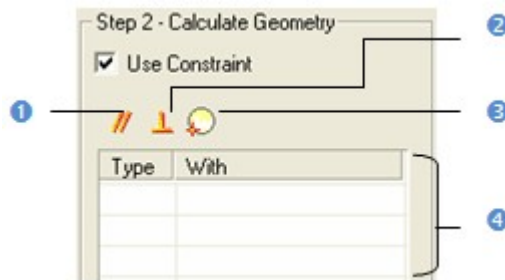
Plane

Three types of constraint are available (see [A]). The constraint list (with two columns **Type** and **With**) is empty before applying a constraint. A constraint (once applied) is put in the constraint list and each is selected by default (box checked). You can apply one constraint of the same type at once. You can mix a constraint type with another; but you have some restrictions in combining constraint types (see the table).

			
	No	No	Yes
	No	No	Yes
	Yes	Yes	No

If you mix incompatible constraints together; a warning message appears. For those you do not want anymore, you can de-select them by un-checking their check box. To remove all created constraints, click on the **Clear** button. After applying constraints, the constraint list looks as shown in the table.

[A]



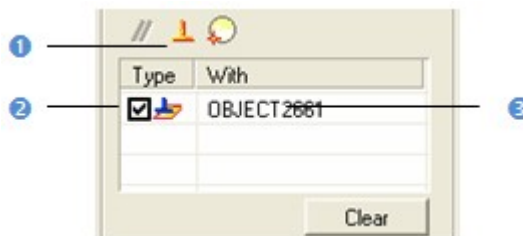
1 - Make Parallel

2 - Make Perpendicular

3 - Pass Through Point

4 - Constraint list

[B]



1 - Constraint type

2 - Check box

3 - Entity used as constraint

To Use a Constraint to Calculate a Plane:


1. Constrain a plane parallel/perpendicular to an entity.
2. Or constrain a plane passing through a point.
3. Un-checking a defined constraint in the constraint list will free the related constraint type for use.

Tips:

- You can also select an applied constraint from the constraint list and use the **Del** to clear it.
- All constraints can be selected from the pop-up menu.


Making Parallel

To Make Parallel:

1. Click the **Make Parallel**  icon. The **Make Parallel** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Making Perpendicular

To Make Perpendicular:

1. Click the **Make Perpendicular**  icon. The **Make Perpendicular** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).







Passing Through a Point

To Pass Through a Point:

1. Click the **Pass through Point**  icon. The **3D Point Tool** toolbar and its information box at the top right corner of the **3D View**.
2. Define and validate a 3D point.

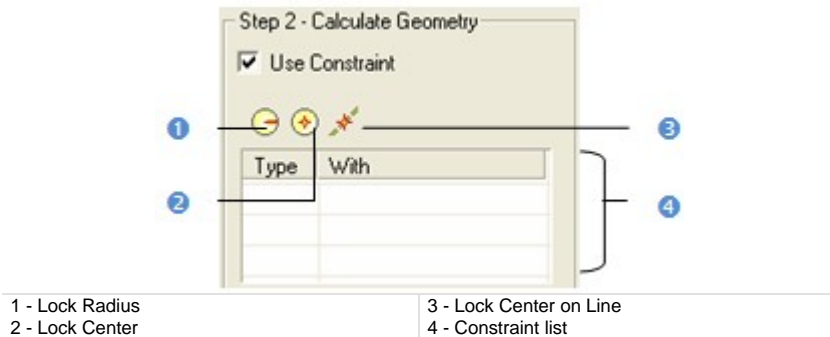
Sphere

Three types of constraint are available (see [A]). The constraint list (with two columns **Type** and **With**) is empty before applying constraints. A constraint (when applied) is put in the constraint list and each is selected by default (box checked). You can apply one constraint of the same type at once. You can mix a constraint type with another; but you have some restrictions in combining constraint types (see the table).

			
	No	No	Yes
	No	No	No
	Yes	No	No

If you mix incompatible constraints together; a warning message appears. For those you do not want anymore, you can de-select them by un-checking their check box. To remove all created constraints, click on the **Clear** button. After applying constraints, the constraint list looks as shown in [B].

[A]



[B]



To Use a Constraint to Calculate a Sphere:


1. Lock a sphere center.
2. Or lock a sphere radius.
3. Or lock a sphere center on a line.
4. Un-check a defined constraint in the constraint list to free it.

Tips:

- You can also select an applied constraint from the constraint list and use the **Del** to clear it.
- All constraints can be selected from the pop-up menu.


Locking a Radius

To Lock a Radius:

1. Click the **Lock Radius**  icon. The **3D Radius Tool** toolbar opens as well as its information box.
2. Do one of the following:
 - Pick a radial entity,
 - Pick two points,
 - Pick an axis and a point,
 - Edit parameters.
3. Validate the radius.

Locking a Center

To Lock a Center:

1. Click the **Lock Center**  icon. The **3D Point Tool** toolbar opens as well as its information box.
2. Define and validate a 3D point.

Locking Center on Line

To Lock a Center on a Line:

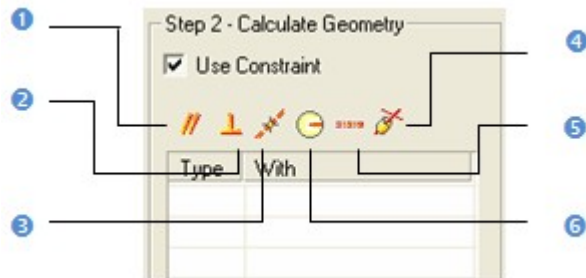
1. Click on the **Lock Center on Line**  icon. The **3D Axis Tool** toolbar opens.
2. Define and validate a 3D axis.

Cylinder

Six types of constraint are available (see [A]). The constraint list (with two columns **Type** and **With**) is empty before applying constraints. All constraints cannot be mixed together; refer to the table below to check how a constraint is compatible with the others. If you mix incompatible constraints together; a warning message appears. After applying constraints, the constraint list looks as shown in [B].

		//	⊥	↗	⊖	⊙				
							↗	⊖	⊙	⊙
//		No	No	Yes	Yes	No	Yes	Yes	No	No
⊥		No	No	Yes	Yes	No	Yes	Yes	No	Yes
↗		Yes	Yes	No	Yes	No	No	Yes	No	Yes
⊖		Yes	Yes	Yes	No	Yes	Yes	Yes	Yes	Yes
⊙		No	No	No	Yes	No	No	Yes	No	No
⊙	↗	Yes	Yes	No	Yes	No	No	Yes	Yes	Yes
	⊖	Yes	Yes	Yes	Yes	Yes	Yes	No	Yes	Yes
	⊙	No	No	No	Yes	No	Yes	Yes	No	No
	⊙	No	Yes	Yes	Yes	No	Yes	Yes	No	No

[A]



- 1 - Make Parallel
- 2 - Make Perpendicular
- 3 - Pass Axis Through Point

- 4 - Make Secant to Cylinder
- 5 - Fit to Axis
- 6 - Lock Radius

[B]



1 - Constraint type

2 - Check box

3 - Entity used as constraint

To Apply Constraints to a Cylinder:


1. Make a cylinder parallel/perpendicular to an entity/plane/direction.
2. Or lock a cylinder radius.
3. Or pass a cylinder axis through a point.
4. Or fit a cylinder axis.
5. Or constraint secant to a cylinder.
6. Un-check a defined constraint in the constraint list to free it.
7. Or click **Clear** to remove all constraints from the list.

Tips:

- You can also select an applied constraint from the constraint list and use the **Del** to clear it.
- All constraints can be selected from the pop-up menu.


Making Parallel

To Make Parallel:

1. Click the **Make Parallel**  icon. The **Make Parallel** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Making Perpendicular

To Make Perpendicular:

1. Click the **Make Perpendicular**  icon. The **Make Perpendicular** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Passing Axis Through Point

To Pass an Axis Through a Point:

1. Click the **Pass Axis Through Point**  icon. The **3D Point Tool** toolbar opens with the **Pick Point** mode is set by default.
2. Define and validate a 3D point.


Locking a Radius

To Lock a Radius:

1. Click the **Lock Radius**  icon. The **3D Radius Tool** toolbar opens as well as its information box.
2. Do one of the following:
 - Pick a radial entity,
 - Pick two points,
 - Pick an axis and a point,
 - Edit parameters.
3. Validate the radius.


Fixing to an Axis

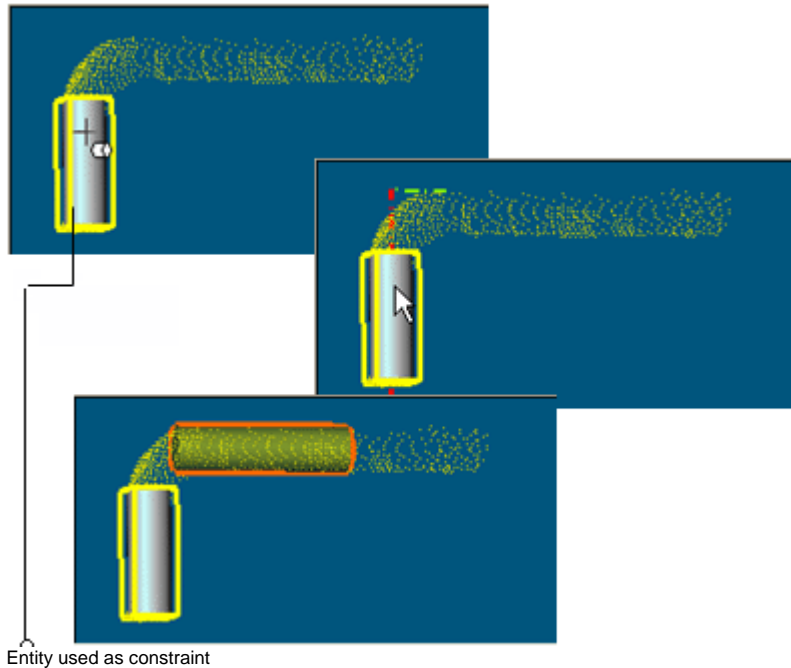
To Fix to an Axis:

1. Click on the **Fix to Axis**  icon. The **3D Axis Tool** toolbar opens.
2. Define and validate a 3D axis.

Making Secant to a Cylinder

To Make Secant to a Cylinder:

1. Click the **Make Secant to Cylinder**  icon. The **3D Secant Tool** dialog as well as its information box appears.
2. Pick a cylinder.
3. If required, check **Use Same Radius** to set the same radius than the picked cylinder.
4. If required, check **Use Given Angle** and give a secant angle.
5. If required, click **Perpendicular** to have a 90° secant angle.
6. Click **OK**.



Note that the **Make Secant to Cylinder** constraint type generates four sub-constraint types according to the option(s) checked.

- If only **Use Same Radius** has been checked, you have the two following constraint types: **Make Axis Secant to Axis** and **Lock Radius**.
- If only **Use Given Angle** has been checked and the given angle value is different to 90° and 270°, you have the two following constraint types: **Make Axis Secant to Axis** and **Lock Angle with Direction**.

- If only **Use Given Angle** has been checked and the given angle value is equal to 90° and 270° , you have the two following constraint types: **Make Axis Secant to Axis** and **Make Perpendicular to Direction**.
- If only **Use Given Angle** has been checked and Perpendicular pressed-on, you have the two following constraint types: **Make Axis Secant to Axis** and **Make Perpendicular to Direction**.
- If the two options have been checked with an angle other than 90° or 270° , you have the three following constraint types: **Make Axis Secant to Axis**, **Lock Radius** and **Lock Angle with Direction**.
- If the two options have been checked with an angle equal to 90° or 270° , you have the three following constraint types: **Make Axis Secant to Axis**, **Lock Radius** and **Make Perpendicular to Direction**.
- If the two options are kept unchecked, you have the **Make Axis Secant to Axis** constraint type.

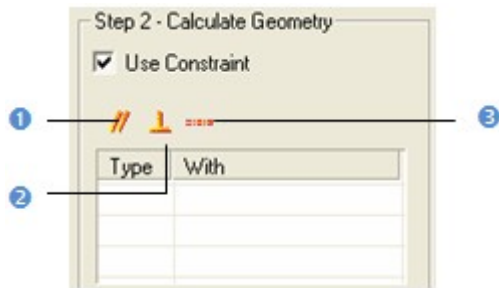
Regular Cone

Three types of constraint are available (see [A]). The constraint list with (two columns **Type** and **With**) is empty before applying constraints. All constraints cannot be mixed together; refer to the table below to check how a constraint is compatible with the others.

	//	⊥	≡
//	No	No	No
⊥	No	No	No
≡	No	No	No

If you mix incompatible constraints together; a warning message appears. After applying constraints, the constraint list looks as shown in [B].

[A]

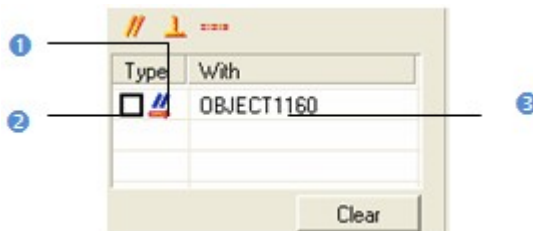


1 - Make Parallel

2 - Make Perpendicular

3 - Fit to Axis

[B]



1 - Constraint type

2 - Check box

3 - Entity used as constraint

To Apply Constraints to a Regular Cone:


1. Make a cone parallel/perpendicular to an entity/plane/direction.
2. Or fit a cone axis.
3. Un-check a defined constraint in the constraint list to free it.
4. Or click **Clear** to remove all constraints from the list.

Tips:

- You can also select an applied constraint from the constraint list and use the **Del** to clear it.
- All constraints can be selected from the pop-up menu.


Making Parallel

To Make Parallel:

1. Click the **Make Parallel**  icon. The **Make Parallel** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Making Perpendicular

To Make Perpendicular:

1. Click the **Make Perpendicular**  icon. The **Make Perpendicular** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).

Fixing to an Axis

To Fix to an Axis:

1. Click on the **Fix to Axis**  icon. The **3D Axis Tool** toolbar opens.
2. Define and validate a 3D axis.

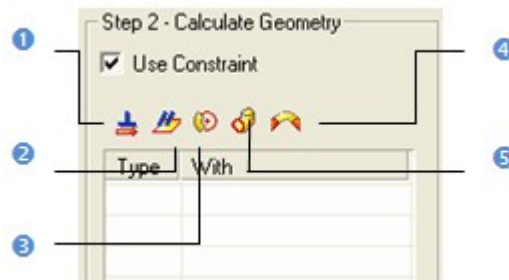
Circular Torus

Five types of constraint are available (see [A]). The constraint list (with two columns **Type** and **With**) is empty before applying constraints. All constraints cannot be mixed together; refer to the table below to know on how a constraint is compatible with the other(s).

					
	No	No	Yes	Yes	No
	No	No	Yes	Yes	No
	Yes	Yes	No	Yes	No
	Yes	Yes	Yes	No	No
	No	No	No	No	No

If you mix incompatible constraints together; a warning message appears. After applying constraints, the constraint list looks as shown in [B].

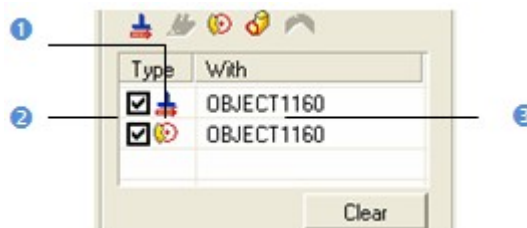
[A]



1 - Make Perpendicular to Direction
2 - Make Parallel to Plane
3 - Lock Center Line Radius

4 - Align to Join Two Existing Secant Cylinders of Same Radius
5 - Lock Pipe Radius

[B]



1 - Constraint type

2 - Check box

3 - Entity used as constraint

To Apply Constraints to a Circular Torus:

1. Make a circular torus parallel/perpendicular to an entity/plane/direction.
2. Or fit a circular torus axis.
3. Or lock a circular torus's center line radius.
4. Or lock a circular torus's pipe radius.
5. Align and join a circular torus to two secant cylinders of the same radius.
6. Un-check a defined constraint in the constraint list to free it.
7. Or click **Clear** to remove all constraints from the list.


Note: Press **Esc** to leave the picking mode.

Tips:

- You can also select an applied constraint from the constraint list and use the **Del** to clear it.
- All constraints can be selected from the pop-up menu.

Making Perpendicular to a Direction

To Make Perpendicular to a Direction:

1. Click the **Make Perpendicular to a Direction**  icon. The **3D Direction Tool** toolbar opens.
2. Define and validate a 3D direction.


Making Parallel to a Plane

To Make Parallel to a Plane:

1. Click the **Make Parallel to a Plane**  icon. The **3D Plane Tool** toolbar opens.
2. Define and validate a 3D plane.


Locking Center Line Radius

To Lock a Center Line Radius:

1. Click the **Lock Center Line Radius**  icon. The **3D Radius Tool** toolbar opens as well as its information box.
2. Do one of the following:
 - Pick a radial entity,
 - Pick two points,
 - Pick an axis and a point,
 - Edit parameters.
3. Validate the radius.


Locking Pipe Radius

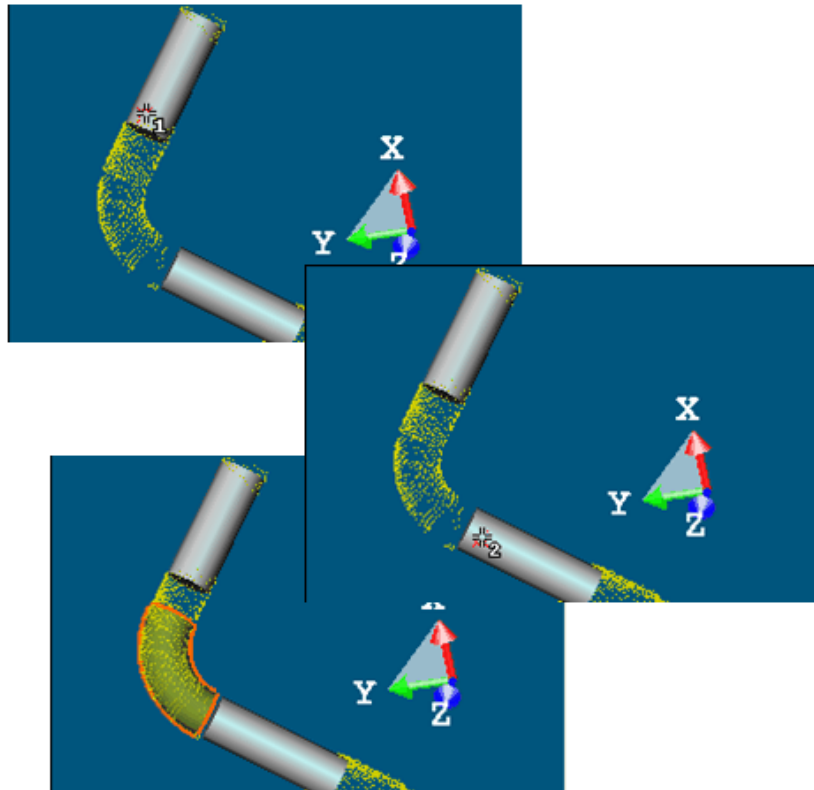
To Lock a Pipe Radius:

1. Click the **Lock Pipe Radius**  icon. The **3D Radius Tool** toolbar opens as well as its information box.
2. Do one of the following:
 - Pick a radial entity,
 - Pick two points,
 - Pick an axis and a point,
 - Edit parameters.
3. Validate the radius.

Aligning to Join to two Secant Cylinders of Same Radius

To Align to Join to two Secant Cylinders of Same Radius:





1. Click the **Align to Join to Two Secant Cylinders of Same Radius**  icon.
2. Pick an existing cylinder.
3. Pick another existing cylinder. The result is null if the two picked cylinders are not secant or do not have the same diameter.



Note: A warning appears if the two picked cylinder axes are parallel.

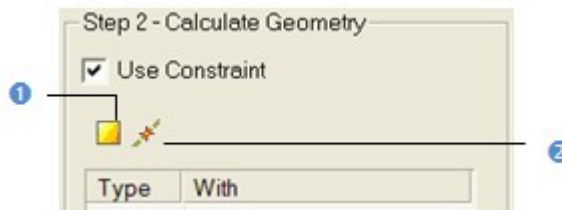
3D Point

Two types of constraint are available (see [A]). The constraint list (with two columns **Type** and **With**) is empty before applying constraints. These two constraints cannot be mixed together; refer to the table below to check how a constraint is compatible with the others.

		
	No	No
	No	No

If you mix incompatible constraints together; a warning message appears. After applying constraints, the constraint list looks as shown in [B].

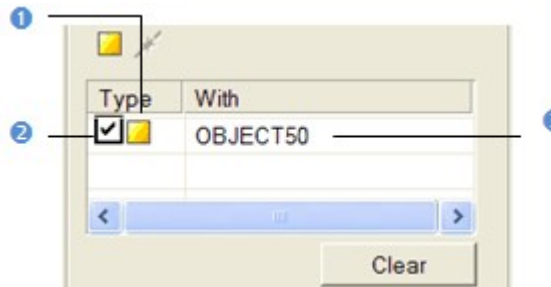
[A]



1 - Lock on Plane

2 - Lock to Line or Axis

[B]



1 - Constraint type

2 - Check box

3 - Entity used as constraint

To Use a Constraint to Calculate a 3D Point:


1. Lock a 3D point on a plane.
2. Or lock a 3D point on a line/axis.
3. Un-check a defined constraint in the constraint list to free it.
4. Or Click **Clear** to remove all constraints from the list.

Tips:

- You can also select an applied constraint from the constraint list and use the **Del** to clear it.
- All constraints can be selected from the pop-up menu.


Locking on Plane

To Lock on a Plane:

1. Click the **Lock on Plane**  icon. The **3D Plane Tool** toolbar opens with the **Pick Entity with Direction** mode is set by default.
2. Define and validate a 3D plane.







Locking to Line (or Axis)

To Lock on a Plane:

1. Click the **Lock to Line (or Axis)**  icon. The **3D Axis Tool** toolbar opens with the **Pick Axis Entity** mode is set by default.
2. Define and validate a 3D direction.

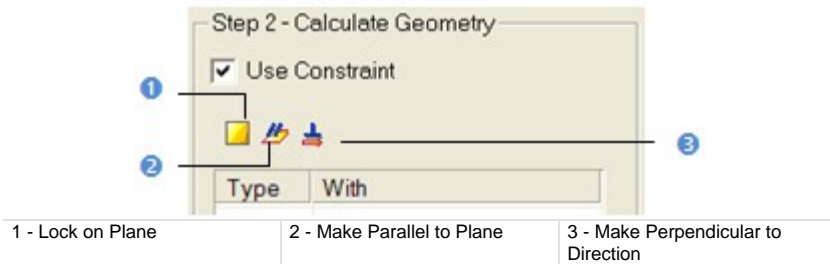
Circular Arc

Three types of constraint are available (see [A]). The constraint list (with two columns **Type** and **With**) is empty before applying constraints. **Lock on Plane**, **Make Parallel to Plane** and **Make Perpendicular to Direction** (already evoked) will detail here. All constraints cannot be mixed together; refer to the table below to check how a constraint is compatible with the others.

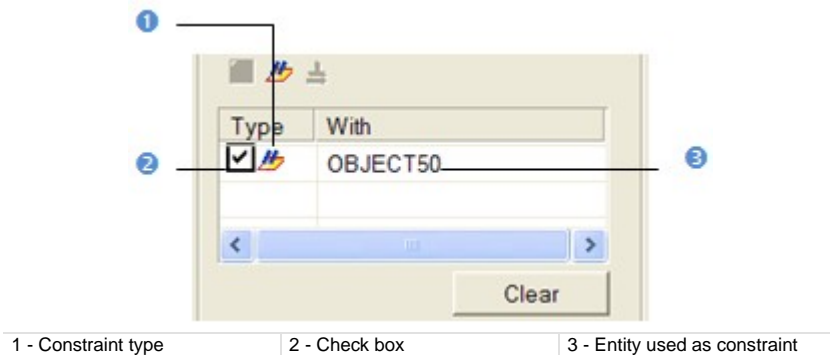
			
	No	No	No
	No	No	No
	No	No	No

If you mix incompatible constraints together; a warning message appears. After applying constraints, the constraint list looks as shown in [B].

[A]



[B]



To Apply Constraints to a Circular Arc:

1. Lock a circular arc on a plane (see the **3D Plane Tool**).


2. Or make a circular arc parallel to a plane (see the **3D Plane Tool**).
3. Or make a circular arc perpendicular to a direction (see the **3D Direction Tool**).
4. Un-check a defined constraint in the constraint list to free it.
5. Or click **Clear** to remove all constraints from the list.

Tips:

- You can also select an applied constraint from the constraint list and use the **Del** to clear it.
- All constraints can be selected from the pop-up menu.

Locking on Plane

To Lock on a Plane:

1. Click the **Lock on Plane**  icon. The **3D Plane Tool** toolbar opens with the **Pick Entity with Direction** mode is set by default.
2. Define and validate a 3D plane.


Making Parallel to a Plane

To Make Parallel to a Plane:

1. Click the **Make Parallel to a Plane**  icon. The **3D Plane Tool** toolbar opens.
2. Define and validate a 3D plane.






Making Perpendicular to a Direction

To Make Perpendicular to a Direction:

1. Click the **Make Perpendicular to a Direction**  icon. The **3D Direction Tool** toolbar opens.
2. Define and validate a 3D direction.

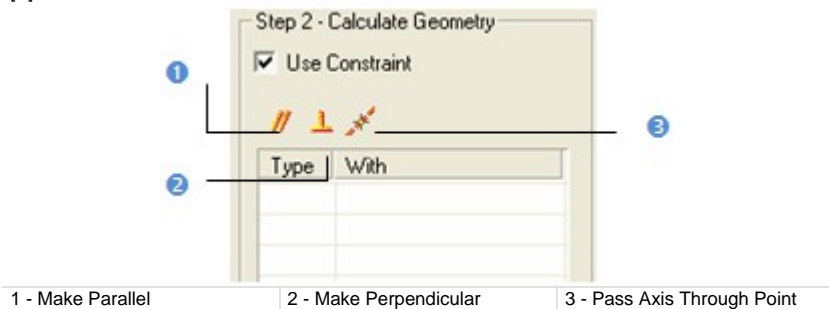
Segment

Three types of constraint are available (see [A]). The constraint list (with two columns **Type** and **With**) is empty before applying constraints. All constraints cannot be mixed together; refer to the table (click to see) to know on how a constraint is compatible with the other.

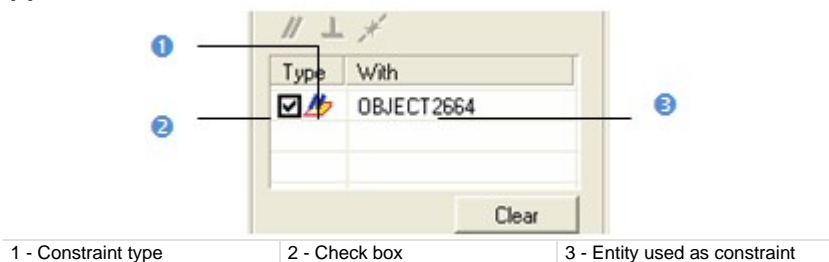
			
	NO	No	Yes
	NO	No	Yes
	Yes	Yes	No

If you mix incompatible constraints together; a warning message appears. After applying constraints, the constraint list looks as shown in [B].

[A]



[B]



To Use a Constraint to Calculate a Segment:

1. Make a segment parallel/perpendicular to an entity/plane/direction (see the [3D Picking Tool/3D Plane Tool/3D Direction Tool](#)).
2. Or pass a segment through a point (see the [3D Point Tool](#)).
3. Un-check a defined constraint in the constraint list to free it.


4. Or click **Clear** to remove all constraints from the list.

Tips:

- You can also select an applied constraint from the constraint list and use the **Del** to clear it.
- All constraints can be selected from the pop-up menu.


Making Parallel

To Make Parallel:

1. Click the **Make Parallel**  icon. The **Make Parallel** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Making Perpendicular

To Make Perpendicular:

1. Click the **Make Perpendicular**  icon. The **Make Perpendicular** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).

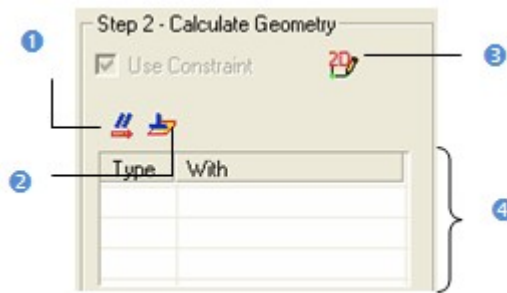
Passing Axis Through Point

To Pass an Axis Through a Point:

1. Click the **Pass Axis Through Point**  icon. The **3D Point Tool** toolbar opens with the **Pick Point** mode is set by default.
2. Define and validate a 3D point.

Extrusion

This step is not in option as for the other geometry types. The **Use Constraint** option is checked by default and cannot be unchecked. The **Drawing Tool** and **Picking Parameters** (in 3D constraint mode) toolbars appear and the cursor is in the drawing mode. Two types of constraint are available. The constraint list (with two columns **Type** and **With**) is empty before applying constraints.



1 - Make Parallel to Direction
2 - Make Perpendicular to Plane

3 - Launch 2D Drawing Tool
4 - Constraint list

Defining a Polyline

In the **Drawing Tool** toolbar, not only the **Change Mode** icon is available but also the **Draw Rectangle** and **Draw Circle** icons. This differs from the **Drawing Tool** toolbar in the **Polyline Drawing Tool**.

To Define a Polyline:

- If required, click the **Start 2D Drawing Tool**  icon.

Notes:

- After defining a polyline, all drawing modes (**Line**, **Arc**, **Rectangle** and **Circle**) are dimmed in the **Drawing Tool** toolbar.
- The **Walkthrough** navigation mode is forbidden. If you are in the **Walkthrough** mode, the navigation mode will swap of its own from that mode to **Examiner** after starting drawing.

Drawing in a Plane Parallel to the Screen View

To Draw a Polyline in a Plane Parallel to the Screen View:

1. Choose a drawing mode among **Line**, **Arc**, **Rectangle** and **Circle**.
2. Draw a polyline. The scene is locked on a plane parallel to the screen view with a 2D grid superimposed (if not hidden previously) and the **Picking Parameters** toolbar appears in the 2D constraint mode.
3. Validate the polyline. The scene is free from the 2D lock.

The **Lock 2D Curve** and **Make Parallel to Direction** constraints appear in the constraint list. The primitive to extrude will have an axis direction perpendicular to the screen view.




1 - Constraint type

2 - Check box

Drawing in a User-Defined Plane

To Draw a Polyline in a User-Defined Plane:

1. Click the **Start 3D Plane Tool**  icon. The **3D Plane Tool** toolbar appears.
2. Define and validate a 3D plane*. The scene is locked on the defined 3D plane with a 2D grid superimposed (if not hidden previously) and the **Picking Parameters** toolbar appears in the 2D constraint mode.
3. Choose a drawing mode among **Line**, **Arc**, **Rectangle** and **Circle**.
4. Draw and validate a polyline. The scene is free from the 2D lock.


By default, the defined polyline is a 2D polyline in the defined 3D plane which is brought parallel to the screen view. The **Lock 2D Curve** and **Make Parallel to Direction** constraints appear in the constraint list. The primitive to extrude will have an axis direction perpendicular to the screen view.



1 - Constraint type

2 - Check box



Notes:

- (*) Please, refer to the **3D Plane Tool** on how to define a 3D plane.
- The **Find Best Extrusion View**  is only present when using the **3D Plane Tool** in the **Cloud-Based Modeler Tool**. It enables to find the best cutting direction.

Selecting a Polyline

If there is a polyline within the project (or after drawing and creating one), you can set it as selected. Only a 2D polyline and 3D coplanar polyline (all nodes are in the same plane) can be selected.

To Select a Polyline:

1. Click the **Select Polyline**  icon. The cursor becomes as .
2. Pick a polyline. A polyline (in green) appears over the picked polyline. The scene is locked on the polyline's plane with a 2D grid superimposed (if not hidden previously) and the **Picking Parameters** toolbar appears in the 2D constraint mode.
3. Validate the polyline. It becomes red and the scene is free from the 2D lock.

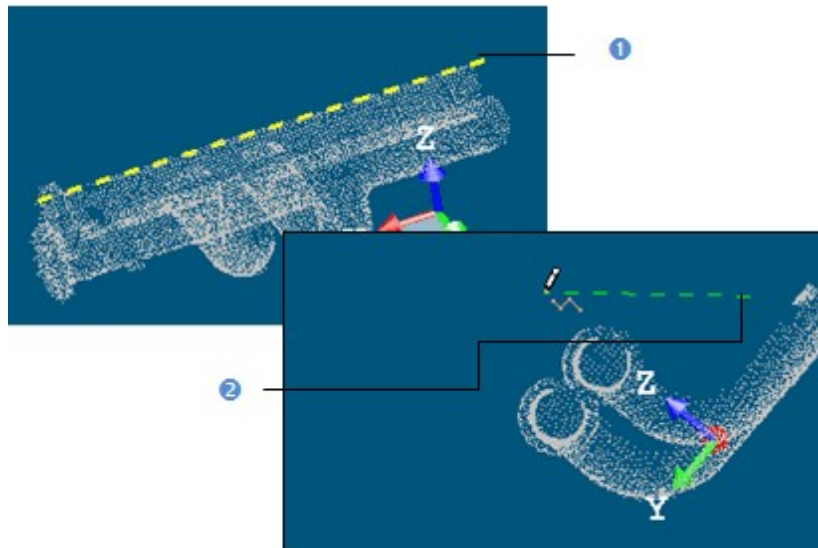
Tip: You can also check **Select Polyline** from the pop-up menu.

Making Parallel to a Direction

If you wish the primitive to extrude has an axis direction parallel to a defined direction, follow the steps below:

To Make Parallel to a Direction:

1. Click the **Make Parallel to Direction** icon. The **3D Direction Tool** toolbar opens.
2. Define and validate a direction. The **Make Parallel to Direction** constraint is put in the constraint list.
3. Draw* or select a polyline (if existing).
4. Validate the defined polyline.



1 - Defined direction

2 - First Picked point in the polyline drawing mode

Notes:


- (*) Picking a first point in the polyline drawing mode will bring the 3D scene locked to a view perpendicular to the defined direction.
- After validating the defined polyline, the **Lock 2D Curve** constraint is put in the constraint list under the **Make Parallel to Direction** constraint.

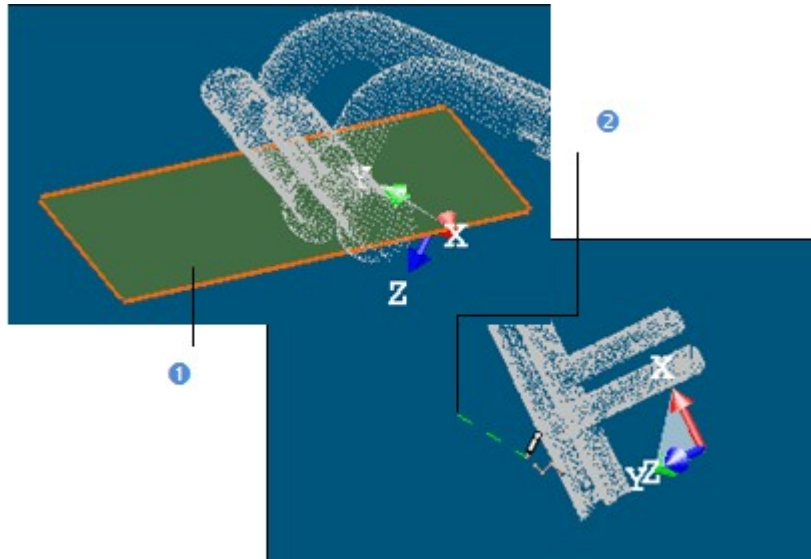
Tip: You can also select an applied constraint from the constraint list and use the **Del** to clear it.

Making Perpendicular to a Plane

If you wish the primitive to extrude has an axis direction parallel to the normal direction of a defined plane, follow the steps below:

To Make Perpendicular to a Plane:

1. Click the **Make Perpendicular to Plane**  icon. The **3D Plane Tool** toolbar opens.
2. Define and validate a plane. The **Make Perpendicular to Plane** constraint is put under the constraint list.
3. Draw* or select a polyline (if existing).
4. Validate the defined polyline.



1 - A defined plane

2 - The first picked point in the polyline drawing mode

Notes:

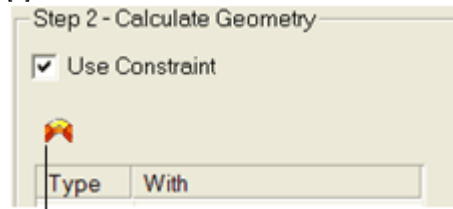
- (*) Picking a first point in the polyline drawing mode will bring the 3D scene locked to a view parallel to the defined plane with a 2D Grid (if not previously hidden).
- After validating the defined polyline, the **Lock 2D Curve** constraint is put in the constraint list under the **Make Perpendicular to Plane** constraint.

Tip: You can also select an applied constraint from the constraint list and use the **Del** to clear it.

Rectangular Torus

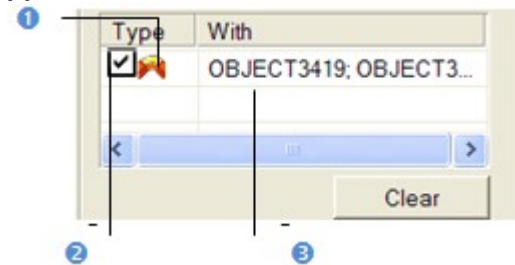
Only one constraint type is available (see [A]). The constraint list with (two columns **Type** and **With**) is empty before applying a constraint. After applying a constraint, the constraint list looks as shown in [B].

[A]



Align to Join to Two Existing Secant Boxes of Same Section

[B]



1 - Constraint type

2 - Check box

3 - Entity used as constraint

To Apply Constraints to a Rectangular Torus:

1. Align to join to two existing secant boxes of same section.
2. Un-check a defined constraint in the constraint list to free it.
3. Or click **Clear** to remove all constraints from the list.


Tips:

- You can also select an applied constraint from the constraint list and use **Del** to clear it.
- All constraints can be selected from the pop-up menu.

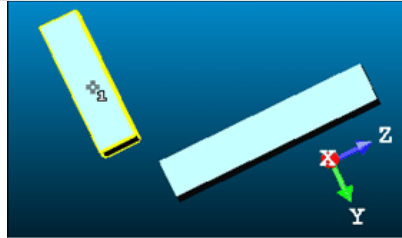
Note: The user should fit (or extract) a **Rectangular Torus** with constraint. Otherwise, the **Fit** and **Extract** buttons remain dimmed.

Aligning to Join Two Existing Secant Boxes of Same Section

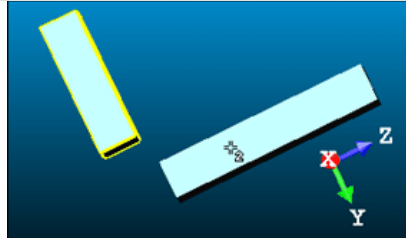
To Align to Join to Two Existing Secant Boxes of Same Section:

1. Click the **Align to Join to Two Existing Secant Boxes of Same Section**  icon. The cursor becomes as shown in [A].
2. Pick a first box. The cursor takes the shape shown in [B].

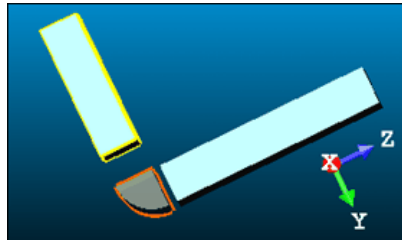
[A]



[B]



3. Pick another box. If the two boxes are secant and have the same section, a rectangular torus appears.









- Its **Direction of Normal** (also called **Direction of Axis**) is parallel to the two boxes' **Direction of Normal** (also called **Direction of Width**).
 - Its **Bend Angle** is equal to the angle drawn by the two boxes' **Direction of Height**.
 - Its **Outer Diameter** is equal to the two boxes' **Depth**.
4. If the two picked boxes are not secant; the "This constraint cannot be activated because the two boxes are not secant" warning message appears. Click OK. The warning message closes and the **Align to Join to Two Existing Secant Boxes of Same Section** constraint is left.
 5. If the two picked boxes do not have the same section, the "There is too much indetermination to activate this constraint: two boxes are identical, one of them is a cube or they have no common face" warning message appears. Click OK. The warning message closes and the **Align to Join to Two Existing Secant Boxes of Same Section** constraint is left.

Tips:

- If required, make the two boxes secant using the **Make Secant to a Box (With Same Section)** constraint in the **Geometry Modifier Tool**.
- If required, modify manually the two boxes' parameters (like **Center**, **Width**, **Height**, **Depth**, **Directions**, etc.) in the **Property** window to make sure that both are secant and have the same section.

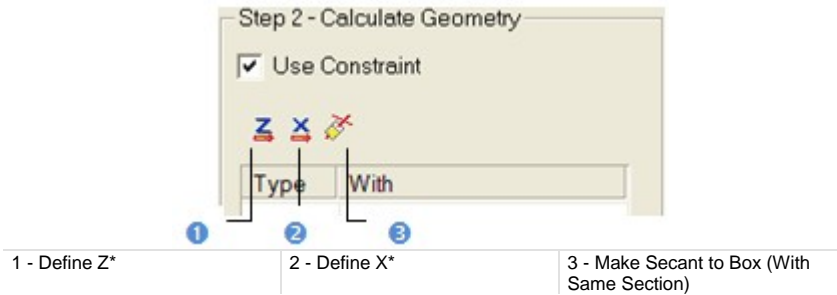
Box

Three types of constraint are available (see [A]). The constraint list with (two columns **Type** and **With**) is empty before applying constraints. All constraints cannot be mixed together, refer to the table below to check how a constraint is compatible with the others.

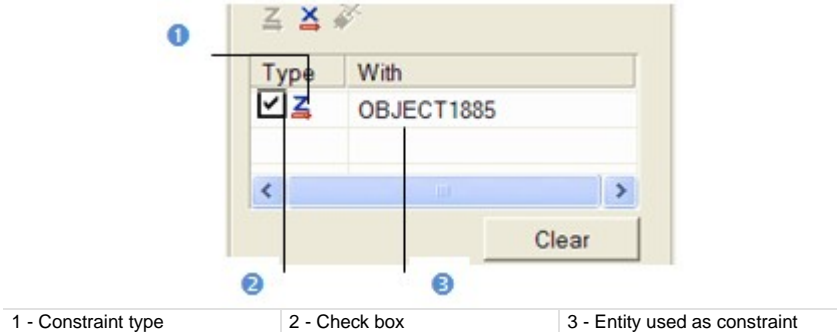
			
	No	Yes	No
	Yes	No	No
	No	No	Yes

If you mix incompatible constraints together; a warning message appears. After applying constraints, the constraint list looks as shown in [B].

[A]



[B]



To Use a Constraint to Calculate a Box:

1. Define the vector X direction,
2. Or define the normal Z direction,
3. Or make secant to a box (with same section).


4. Un-check a defined constraint in the constraint list to free it.
5. Or click **Clear** to remove all constraints from the list.

Tips:

- You can also select an applied constraint from the constraint list and use **Del** to clear it.
- All constraints can be selected from the pop-up menu.

Defining the Vector X Direction

To Define the Vector X Direction:


1. Click the **Define X**  icon. The **3D Direction Tool** toolbar opens with the **Pick an Entity With Direction** set by-default.
2. Define a direction using available tools.

The **Vector X** (also called **Direction of the Weight** in the **Property** window) of the box is parallel* to the defined direction.

Note: (*) But not necessary in the same direction.

Defining the Normal Z Direction

To Define the Normal Z Direction:

1. Click the **Define Z**  icon. The **3D Direction Tool** toolbar opens with the **Pick an Entity With Direction** set by-default.
2. Define a direction using available tools.

The **Normal Z** (also called **Direction of the Height** in the **Property** window) of the box is parallel** to the defined direction.

Note: (*) But not necessary in the same direction.

Making Secant to a Box (With Same Section)

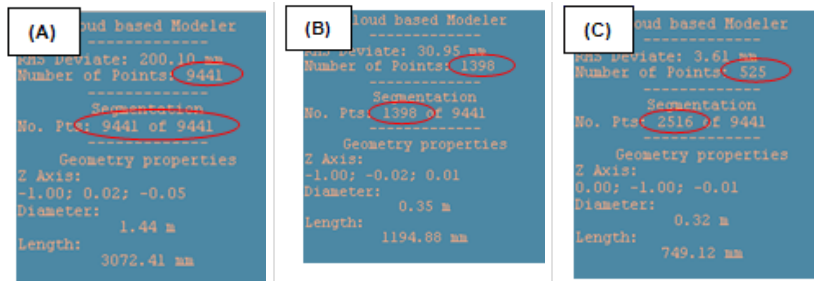
To Make Secant to a Box (With Same Section):

1. Click the **Make Secant to a Box (With Same Section)**  icon.
2. Pick a box.

Calculating a Geometry

This step can come after choosing a geometry type or after constraints have been applied. You have two ways for computing a geometry: **Fit** and **Extract**. **Fit** consists of adjusting the displayed/fenced point cloud with geometry. **Extract** consist in picking a point on the point cloud where the geometry should lie. The resulting geometry will be fitted to a subset of the point cloud in the neighborhood around the picked point. This is a faster way of defining a geometry; no fence is necessary.

If no sampling/segmentation has been done; the displayed cloud (current cloud data) will be used for fitting or for extraction. In (A), the number of points in the fitted geometry is equal to the number of points in the current cloud data. If a sampling/segmentation has been done, the number of points in the geometry is equal to the number of points after segmenting/sampling if **Fit** has been chosen (B) and different if **Extract** has been chosen (C). After fitting or extracting a geometry, its properties are shown in the **Cloud-Based Modeler** information box.



To Fit a Geometry on Points:

- Click **Fit**. A primitive of the type selected in **Step 1** appears.

To Extract a Geometry from a Picked Point:

- Click **Extract**.
- Pick a point. A primitive of the type selected in **Step 1** appears.

After fitting a geometry on points or extracting a geometry from a picked point, you can change the geometry shape by opening the **Geometry Modifier Tool**. Note that modifications on the geometry will create it as persistent object in the database. If you swap from a geometry type to another, the fitted/extracted geometry will be lost; and no warning message will appear. If constraints have been applied; clicking on the **Clear** button will also cancel the fitted or extracted geometry.

Tip: If a sampling/segmentation has been started; you can directly fit and create a geometry in the database without doing this in two steps. Choose for that the **Fit and Create** command from the pop-up menu or use its related short-cut key **Space bar**.

Notes:

- **Fit** and **Extract** are dimmed if **Extrusion** has been selected in **Step 1** and remains in this state as long as a polyline has not been drawn and validated.
- The **Extract** function will work well when the picked point is in a clear area on the model, i.e., where the model is only present in a large enough neighborhood around the point. When it is difficult to find a clear area, then it is better to first fence then fit

Creating a Geometry

If you are satisfied with the fitted/extracted geometry, you can create it as a persistent object in the **RealWorks** database by using the **Create** button in the dialog. You can create as many geometries as required without leaving the tool. When you need to leave the tool, just click on **Close**.

Tip: **Create** and **Close** can be selected from the pop-up menu.

Geometry Modifier Tool


Rotating or panning an object can be free (called **Standard Navigation**) or constrained. Constraints can be imposed (**Screen Rotation**, **Vertical Pan**, **Horizontal Pan**, etc.) or defined by the user. That's the reason of the **Geometry Modifier Tool** which enables to define these constraints by using manipulators in order to have a fine control in rotating or panning objects.

No selection is required to open the **Geometry Modifier Tool**. Once inside the tool, you need to do a selection - only geometries except polylines and meshes, measures, etc. - in one of these three windows (**3D View**, **List** and **Property**) and display the selection to be able to use a manipulator. Selection can only be single. In the **Modeling** processing mode, you can use all the complete manipulation modes of the tool.

Opening the Tool

The **Geometry Modifier Tool** has several aspects; this depends on the selection done not necessary before opening the tool but also inside the tool. In the **Geometry Modifier Tool**, manipulations are split into three groups: using manipulators for displacement or for changing shape, using picked geometries and using constraints. Constraints vary according the entity type selected before (or after) opening the tool.

To Open the Tool:

- From the **Modeling** menu, select **Geometry Modifier Tool** . The **Geometry Modifier Tool** toolbar appears.



Tip: You can also click on the **Geometry Modifier Tool** icon in the **Tools** toolbar.

Modifying a Shape


Modifying a geometry's shape means changing its dimensions with manipulators which change according to the geometry type.

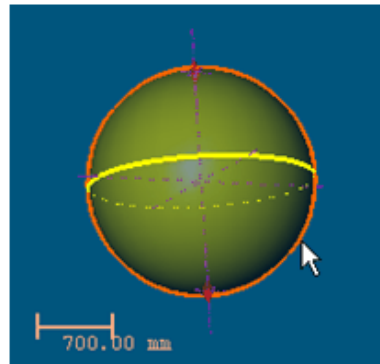
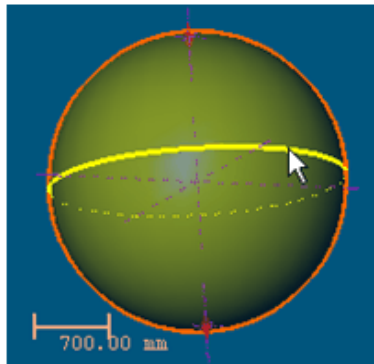
Notes:

- You cannot modify the shape of a geometry of **Plane** (or **3D Point**).
- You can select a geometry before or after choosing the **Modify Shape** command.
- The **Modify Shape** icon can be selected from the pop-up menu.

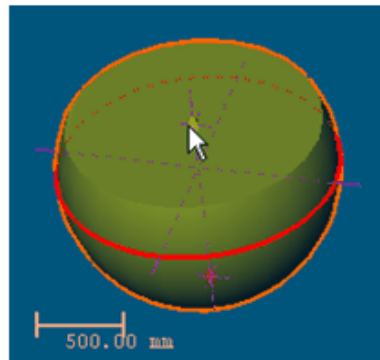
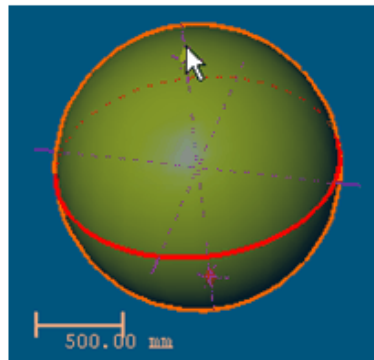
Sphere

To Modify a Sphere Shape:

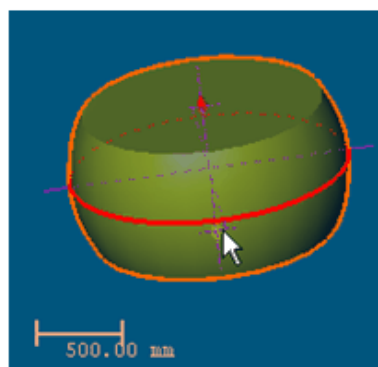
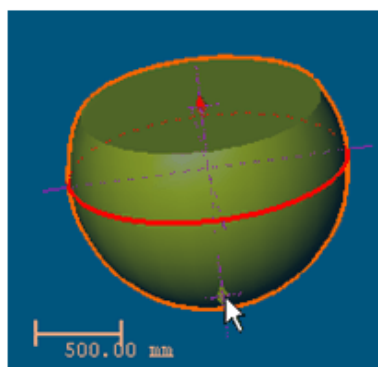
1. Select and display a sphere.
2. Click the **Modify Shape**  icon. The selected sphere is displayed with a manipulator (with one ring and two handles).
3. Pick on the ring to select it. It turns to yellow.
4. Drag to increase (or reduce) the sphere's diameter.



5. Pick on the top handle to select it. It turns to yellow.
6. Drag to shorten the selected sphere's axis by the top. It is like bounding the selected sphere with a plane. The result is a hemisphere.




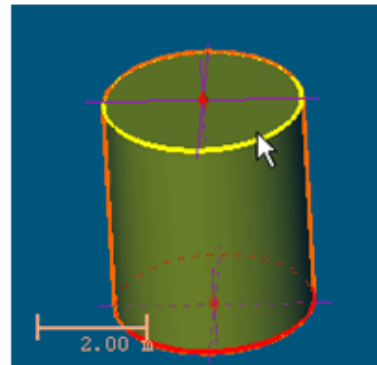
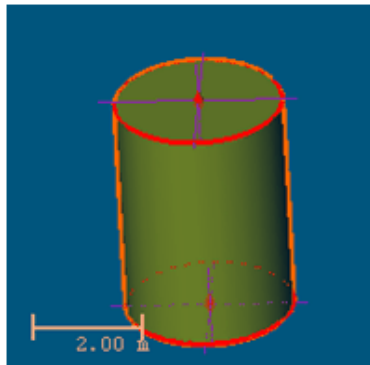
7. Pick on the bottom handle to select it. It turns to yellow.
8. Drag to shorten the selected sphere's axis by the bottom. It is like bounding the initial sphere with a plane. The result is a sliced sphere.



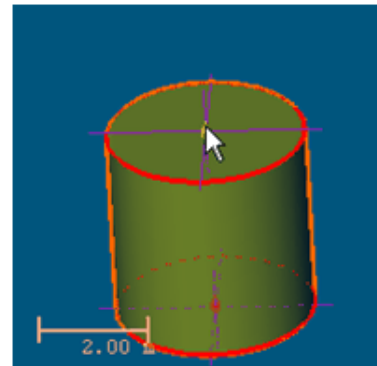
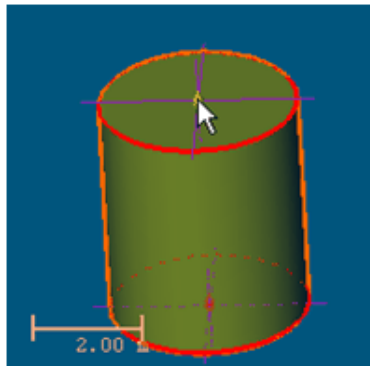
Cylinder

To Modify a Cylinder Shape:

1. Select and display a cylinder.
2. Click the **Modify Shape**  icon. The selected cylinder is displayed with a manipulator (with two rings and two handles).
3. Pick on a ring manipulator to select it. It turns to yellow.
4. Drag to increase or reduce the cylinder's diameter.




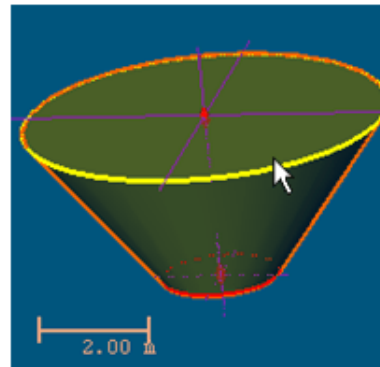
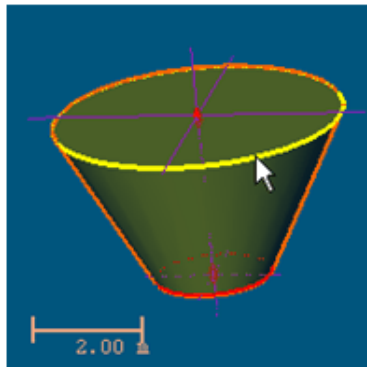
5. Pick on a handle to select it. It turns to yellow.
6. Drag to shorten/lengthen the selected cylinder's axis.



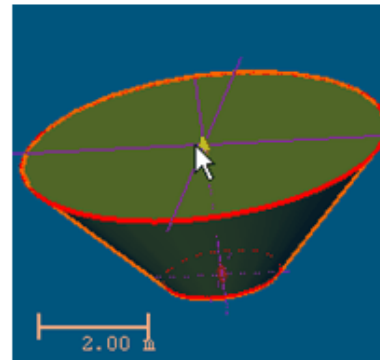
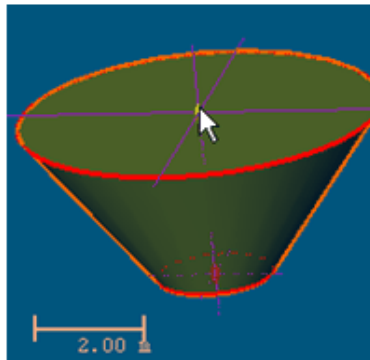
Cone

To Modify a Cone Shape:

1. Select and display a cone.
2. Click the **Modify Shape** . The selected cone is displayed with a manipulator (with two rings and two handles).
3. Pick e.g. the top ring manipulator to select it. It turns to yellow.
4. Drag to increase or reduce the cone's top diameter.




5. Pick e.g. the top handle to select it. It turns to yellow.
6. Drag to shorten the selected cone's axis by the top.

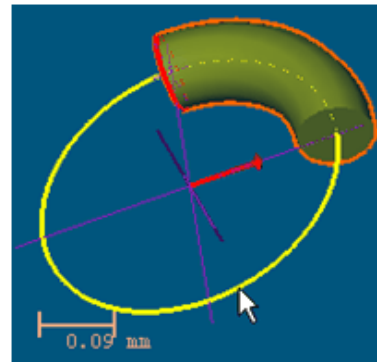
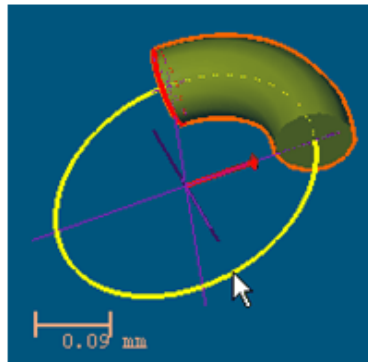


7. Do the same for the top handle (or ring) manipulator.

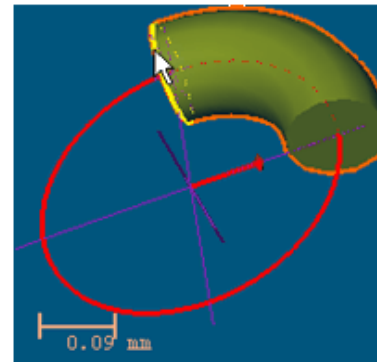
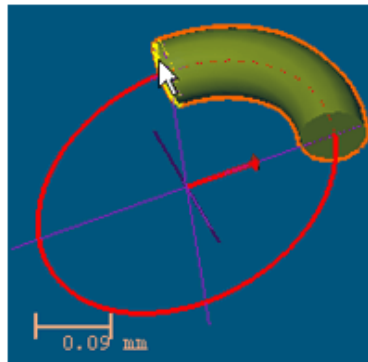
Circular Torus

To Modify a Circular Torus Shape:

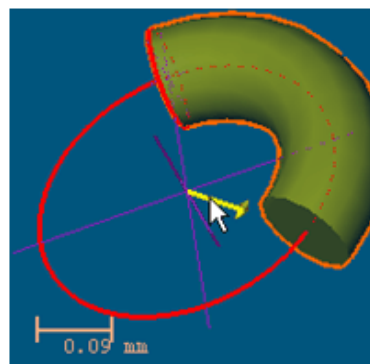
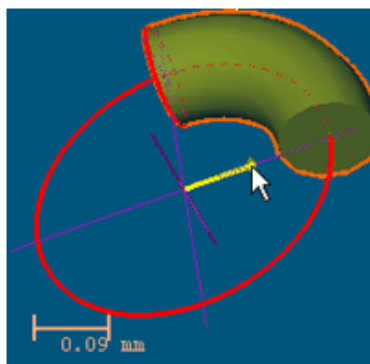
1. Select and display a circular torus.
2. Click the **Modify Shape**  icon. The selected circular torus is displayed with a manipulator (with two rings and one handle).
3. Pick the director radius ring manipulator to select it. It turns to yellow.
4. Drag to increase or reduce the selected circular torus director radius.



5. Pick the generator radius ring manipulator to select it. It turns to yellow.
6. Drag to increase or reduce the selected circular torus generator radius.



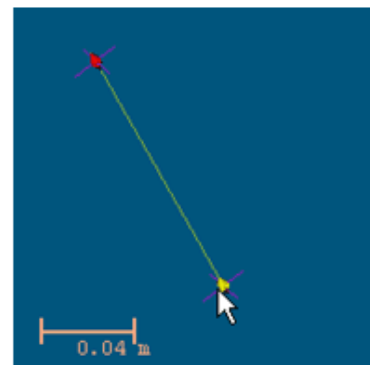
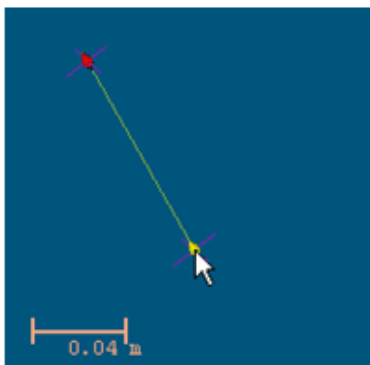
7. Pick on the handle to select it. It turns to yellow.
8. Drag to increase or reduce the selected circular torus angle.



Segment


To Modify a Segment Shape:

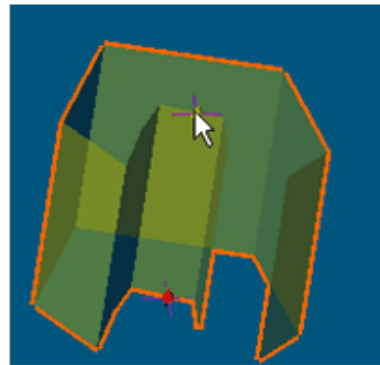
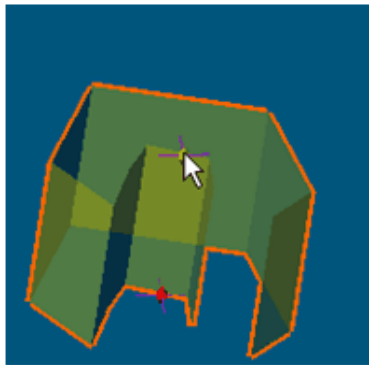
1. Select and display a segment.
2. Click the **Modify Shape** icon. The selected segment is displayed with a manipulator (with two handles).
3. Pick one of the two handles to select it. It turns to yellow.
4. Drag to increase or reduce the selected segment's length.



Extrusion


To Modify an Extrusion Shape:

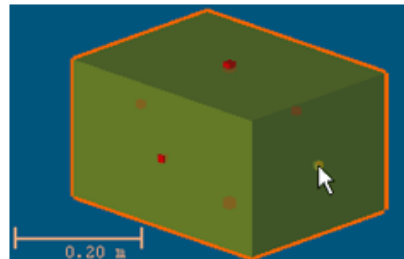
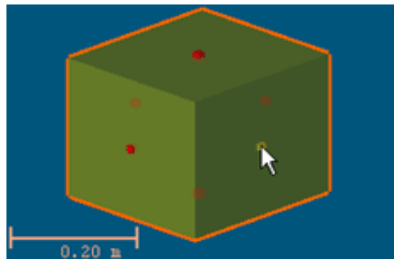
1. Select and display an extruded entity.
2. Click the **Modify Shape**  icon. The extruded entity in selection is displayed with a manipulator (with two handles).
3. Pick one of the two handles to select it. It turns to yellow.
4. Drag to increase or reduce the extruded entity's length.



Box


To Modify a Box Shape:

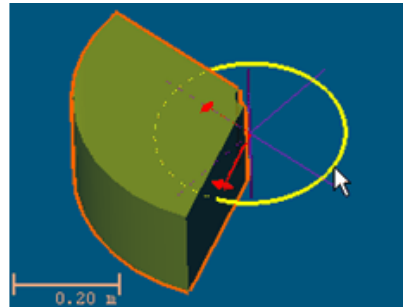
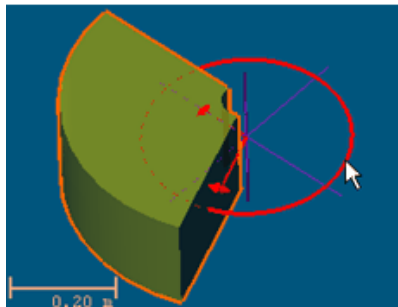
1. Select and display a box.
2. Click the **Modify Shape**  icon. The selected box is displayed with six manipulators, one on each face.
3. Pick on a manipulator to select it. It turns to yellow.
4. Drag to increase or reduce the box size (**Width**, **Height** or **Depth**).



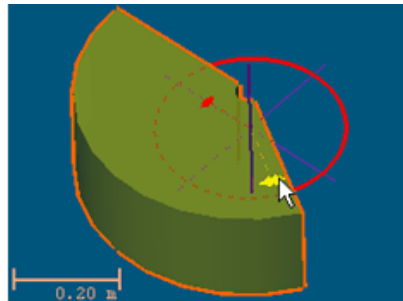
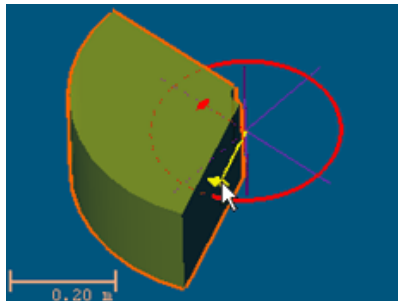
Rectangular Torus

To Modify a Rectangular Torus Shape:

1. Select and display a rectangular torus.
2. Click the **Modify Shape**  icon. The selected rectangular torus is displayed with manipulators (with one ring and two handles).
3. Pick the **Center Line Radius** ring manipulator to select it. It turns to yellow.
4. Drag to increase or reduce the selected rectangular torus **Center Line Radius**.



5. Pick on a handle manipulator to select it. It turns to yellow.
6. Drag to increase or reduce the selected circular torus angle.



Moving a Geometry

"Pan" a geometry means moving and positioning it at a desired location within the scene. "Rotate" a geometry means turning it around an axis or around a point.




1 - Pan Along Home Frame Axes
2 - Pan along Own Axes

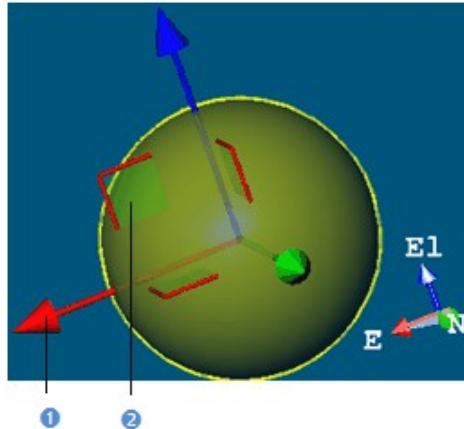
3 - Rotate

Note: You can select a geometry before or after choosing one of the above displacement mode.

Pan Along Home Frame Axes

To Pan Along the Home Frame Axes:

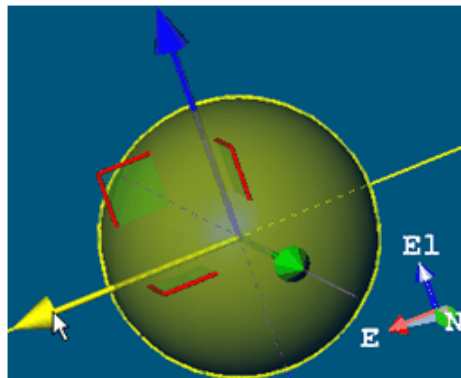
1. Select and display a geometry.
2. Click the **Pan Along Home Frame Axes**  icon. A manipulator (with three-axis handles and three-plane handles) appears. This manipulator has as origin the center of the selection.



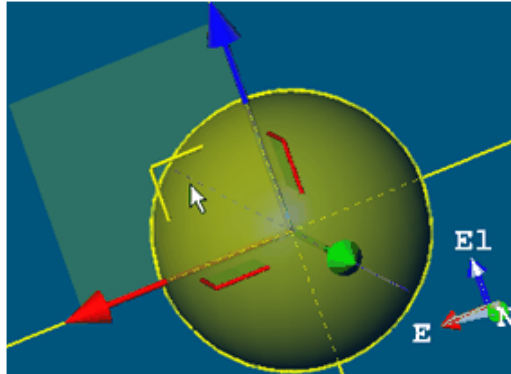
1 - An axis handle

2 - A plane handle

3. Select an axis handle by picking it; it turns to yellow. The direction along which you can pan the selection is highlighted in yellow and those for which you cannot are in mauve.
4. Move the selection along that direction.



5. Select a plane handle by picking it. It turns to yellow. The directions (two) along which you can pan the selection are highlighted in yellow and the one for which you cannot is in mauve.
6. Move the selection in that plane.




Tip: You can also select **Pan along Home Frame Axes** from the pop-up menu or use the **Ctrl + T** short-cut key to choose this manipulation mode.

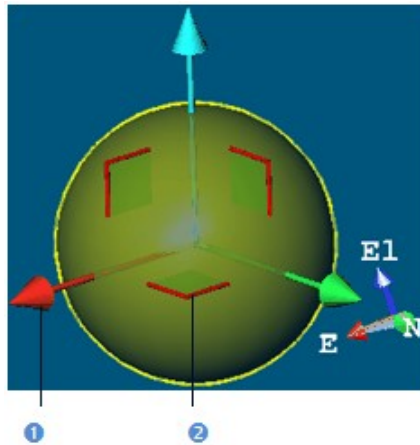
Notes:

- The manipulator (the three handles) has the same color as the global (active) coordinate frame.
- The manipulator will not appear if there is no selection.

Pan Along Own Axes

To Pan Along its Own Axes:

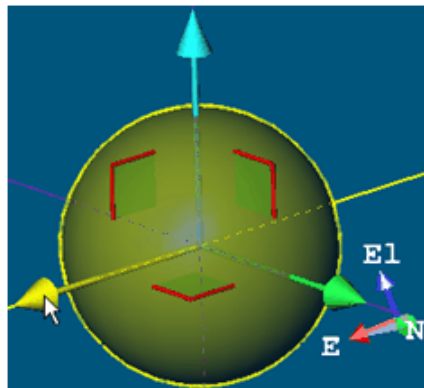
1. Select and display a geometry.
2. Click the **Pan Along Own Axes**  icon. A manipulator (with three-axis handles and three-plane handles) appears. This manipulator has the selected geometry center as origin.



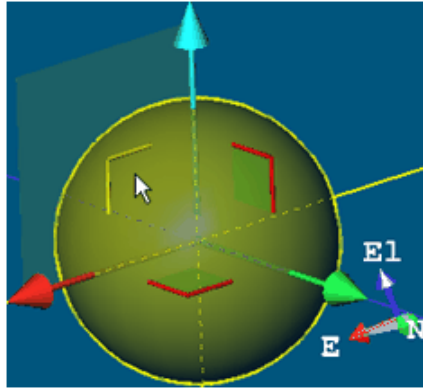
1 - An axis handle (x3)

2 - A plane handle (x3)

3. Select an axis handle by picking it; it turns to yellow. The direction along which you can pan the selected geometry is highlighted in yellow and those for which you cannot are in mauve.
4. Move the selected geometry along that direction.



5. Select a plane handle by picking it. It turns to yellow. The directions (two) along which you can pan the selection are highlighted in yellow and the one for which you cannot is in mauve.
6. Move the selection in that plane.




Tip: You can also select **Pan along Own Axes** from the pop-up menu or use the **T** short-cut key to choose this manipulation mode.

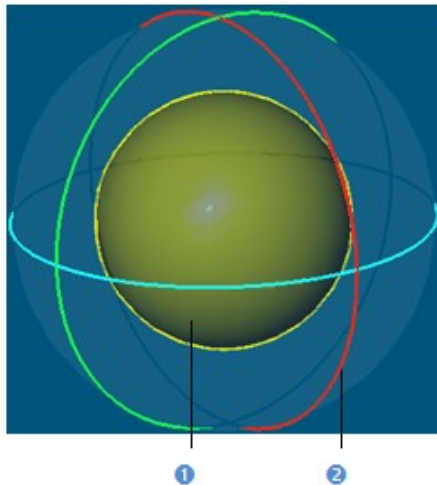
Notes:

- The manipulator (the three-axis handles) does not have the same color as the global (active) coordinate frame.
- The manipulator will not appear if there is no geometry selected.

Rotate

To Rotate a Geometry:

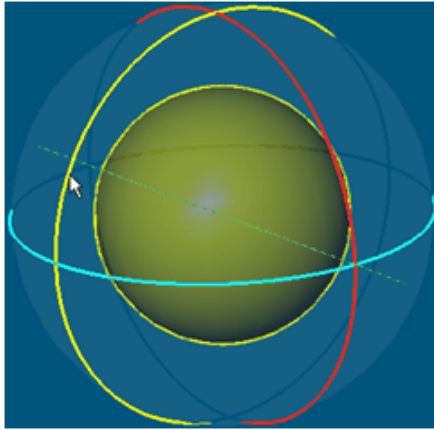
1. Select and display a geometry*.
2. Click the **Rotate**  icon. A manipulator (with three ring handles (red, light blue and green)) appears. This manipulator has the selected geometry center as origin.



1 - A selected object

2 - A ring handle (X3)

3. Select a ring handle by picking it. It turns to yellow. An axis passing the center of that ring handle and perpendicular to it appears. This axis has the color of the selected ring handle.
4. Drag the ring handle to rotate the selected geometry around the axis.

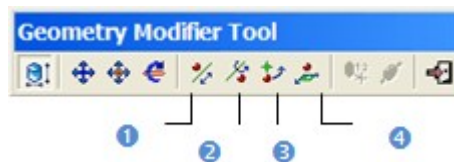


Tip: You can also select **Rotate** from the pop-up menu (or use the **R** short-cut key).

Note: The manipulator will not appear if there is no geometry selected.

Moving a Geometry by Picking Entities

You can pan (or rotate) a geometry along (or around) another geometry axis (or point).




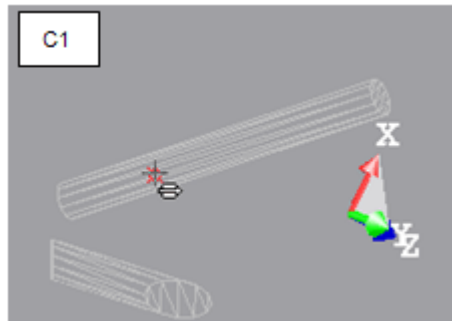
1 - Pick Axis From Other Geometry and Pan
2 - Pick Axis From Other Geometry and Rotate

3 - Pick Point from Other Geometry and Rotate
4 - Pick a Plane and Pan

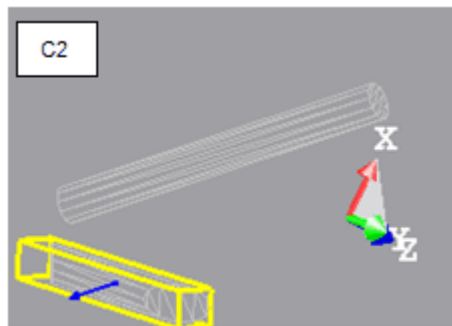
Pick an Axis from Other Geometry, then Pan

To Pick an Axis from Other Geometry, then Pan:

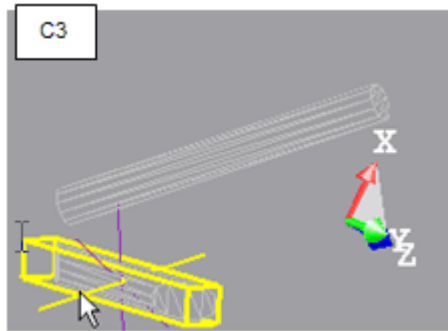
1. Click the **Pick an Axis From Other Geometry, then Pan**  icon. The cursor takes the shape shown in [C1]. This means that you are in the axis picking mode.
2. Place the cursor over an object [C1].



3. Pick the object. The cursor takes its default state (Arrow); this means that you leave the picking mode and you are in the selection mode.
4. Select another object. A position manipulator (in blue) (in the picked object's axis and having as origin its center) appears [C2].



5. Select the position manipulator. It turns to yellow. The direction along which you can pan the second object is in yellow and those (two) for which you cannot are in mauve.
6. Drag the position manipulator to pan the second object along the first object's axis [C3].


**Tips:**

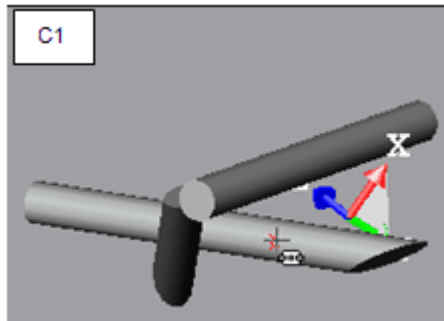
- You can also select **Pick Axis from Other Geometry and Pan** from the pop-up menu.
- With this tool, you can pan an object along its own axis or along another object's axis. Only objects having an axis can be picked for panning purposes.

Note: To leave the picking mode, press **Esc** (or select **Cancel Picking** from the pop-up menu).

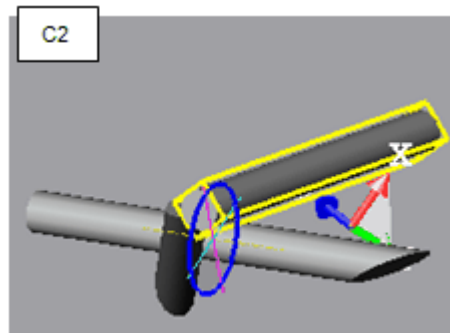
Pick an Axis from Other Geometry, then Rotate

To Pick an Axis From Other Geometry, then Rotate:

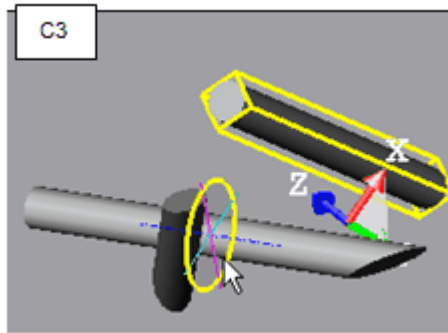
1. Click the **Pick an Axis from Other Geometry, then Rotate**  icon. The cursor takes the shape shown in [C1]. This means that you are in the picking mode.
2. Place the cursor over an object [C1].



3. Pick the object. A manipulator (perpendicular to the picked object's axis (in dashed yellow) and having as origin its center) appears [C2]. The cursor takes its default state (Arrow); this means that you leave the picking mode and you are in the selection mode.



4. Select an object other than the one for which you have picked the axis.
5. Select the ring handle (in deep blue) by picking it. It turns to yellow.
6. Drag the ring handle to rotate the selected object (second) around the axis of the picked object (first) [C3].


**Tips:**

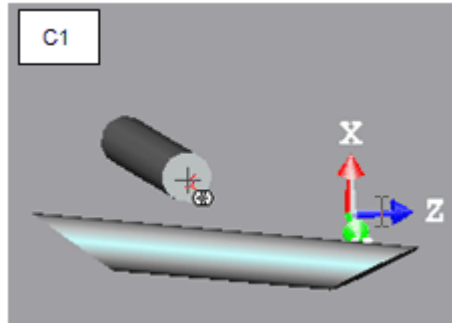
- You can also select the **Pick Axis from Other Geometry and Rotate** icon from the pop-up menu.
- In this manipulation mode, you can rotate an object around its own axis or around another object's axis. Only objects having an axis can be picked for rotation.

Note: To leave the picking mode, press **Esc** (or select **Cancel Picking** from the pop-up menu).

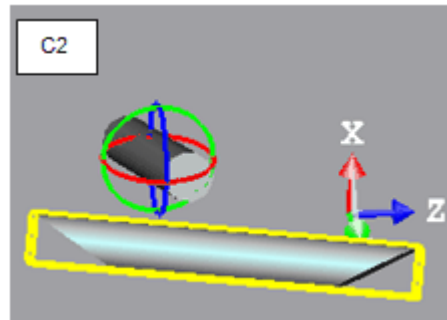
Pick a Point from Other Geometry, then Rotate

To Pick a Point From Other Geometry, Then Rotate:

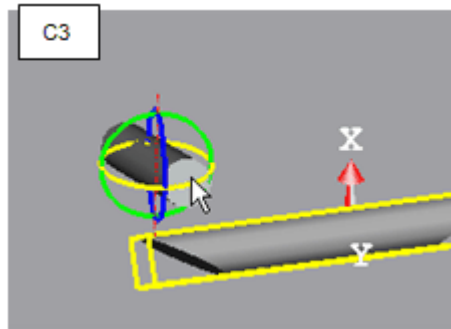
1. Click the **Pick a Point from Other Geometry, then Rotate**  icon. The cursor takes the shape shown in [C1]. This means that you are in the point picking mode.
2. Place the cursor over an object [C1].



3. Pick the object. The cursor takes its default state (Arrow); this means that you leave the picking mode and you are in the selection mode.
4. Select another object by picking it. A manipulator appears [C2]. This manipulator has three rings (red, light blue and green) and has as origin the center of the picked object.




5. Select a ring handle by picking it. It turns to yellow. An axis passing the center of that ring handle and perpendicular to it appears. This axis has the color of the selected ring handle.
6. Drag the ring handle to rotate the second object around the axis of the second object (C3).



Tip: You can also select the **Pick a Point from Other Geometry**, then **Rotate** from the pop-up menu.

Note: To leave the picking mode, press **Esc** (or select **Cancel Picking** from the pop-up menu).

Pick a Plane, and then Pan

After selecting **Pick Plane and Pan** , the cursor takes the shape shown in (A). This means that you are in the plane picking mode. After picking a point, it returns to its default shape (Arrow); this means that you are in the selection mode. With this tool, you can pan an object in a plane or along a plane's axis.

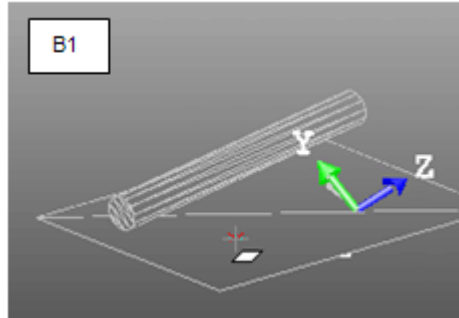
[A] 

Tip: You can also select **Pick Plane and Pan** from the pop-up menu.

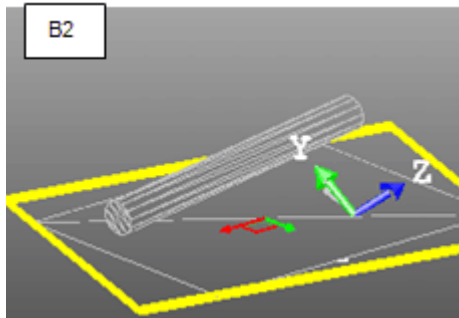
Note: To leave the picking mode, press **Esc** (or select **Cancel Picking** from the pop-up menu).

Picking and Panning a Plane

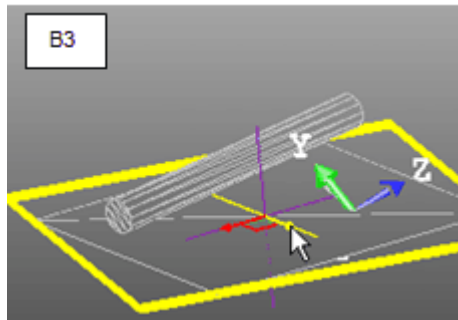
1. Place the cursor over a plane (B1).



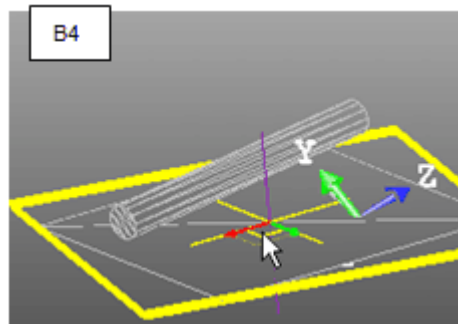
2. Pick the plane. The cursor takes its default state (Arrow); this means that you leave the picking mode and you are in the selection mode.
3. Select the same plane by picking it. A manipulator appears (B2). This manipulator has two axis handles (red and green) and a plane handle and has as origin the center of the picked plane.



4. Select an axis handle by picking it. It turns to yellow. The direction along which you can pan the plane is highlighted in yellow and those (two) for which you cannot are in mauve.
5. Drag the axis handle to pan the plane along the direction in yellow (B3).

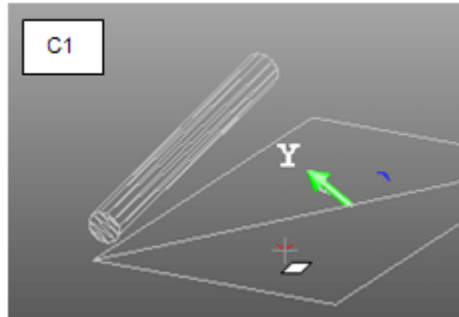


6. Select the plane handle by picking it. It turns to yellow. The directions (two) along which you can pan the object are highlighted in yellow and the one for which you cannot is in mauve.
7. Drag the plane handle to pan the plane along any of the two directions (B4).

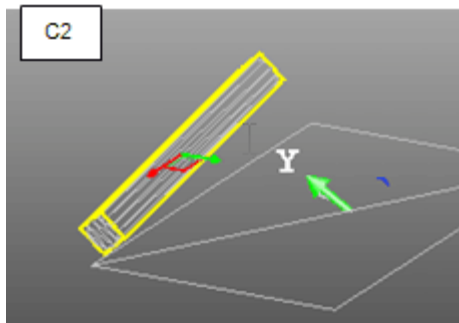


Picking a Plane and Selecting Another Object

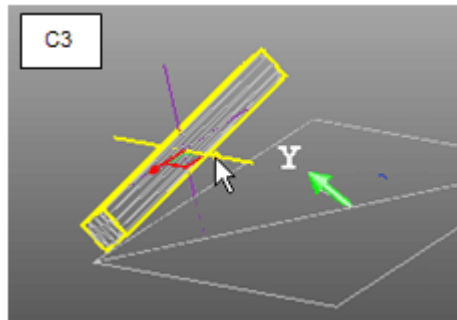
1. Place the cursor over a plane (C1).



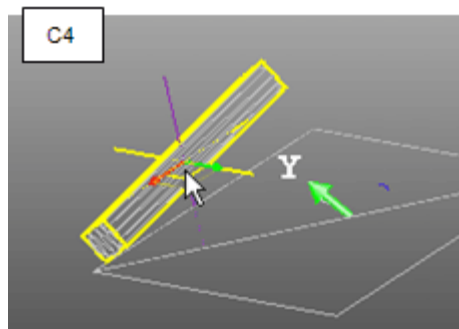
2. Pick the plane. The cursor takes its default state (Arrow); this means that you leave the picking mode and you are in the selection mode.
3. Select another object (plane or others) by picking it. A manipulator appears (C2). This manipulator has two axis handles (red and green) and a plane handle and has as origin the center of the picked object.



4. Select an axis handle by picking it. It turns to yellow. The direction along which you can pan the object is highlighted in yellow and those for which you cannot are in mauve.
5. Drag the axis handle to pan the object along the plane direction in yellow (C3).

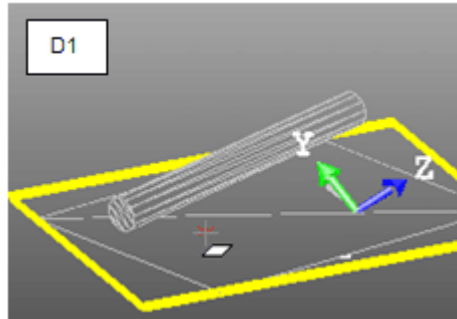


6. Select the plane handle by picking it. It turns to yellow. The directions (two) along which you can pan the object are highlighted in yellow and the one for which you cannot is in mauve.
7. Drag the plane handle to pan the object along any of the plane directions (C4).

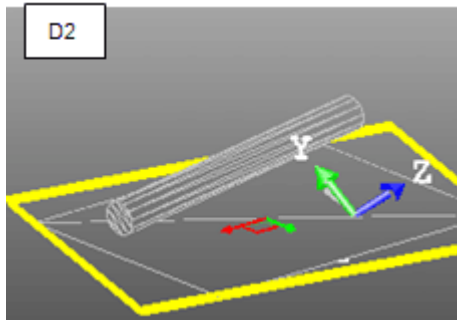


Picking and Selecting a Plane

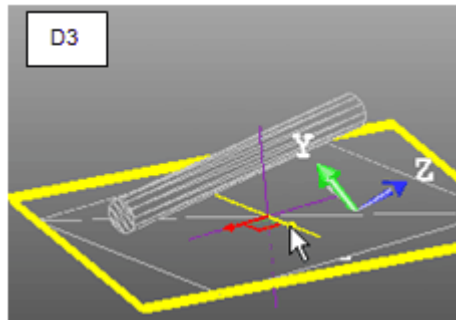
1. Place the cursor over the selected plane (D1).



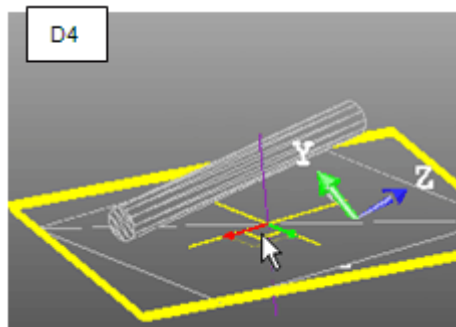
2. Pick the plane. A manipulator appears (D2). This manipulator has two axis handles (red and green) and a plane handle and has as origin the center of the picked plane.



3. Select an axis handle by picking it. It turns to yellow. The direction along which you can pan the plane is highlighted in yellow and those (two) for which you cannot are in mauve.
4. Drag the axis handle to pan the plane along the direction in yellow (D3).

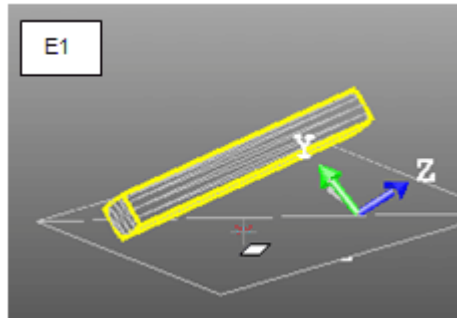


5. Select the plane handle by picking it. It turns to yellow. The directions (two) along which you can pan the object are highlighted in yellow and the one for which you cannot is in mauve.
6. Drag the plane handle to pan the plane along any of the two directions (D4).

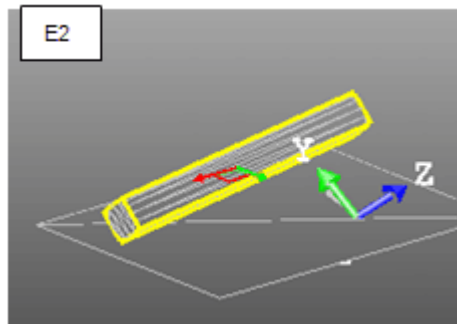


Picking a plane and selecting another object.

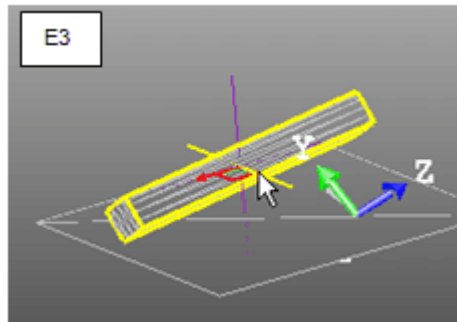
1. Place the cursor over a plane (E1).



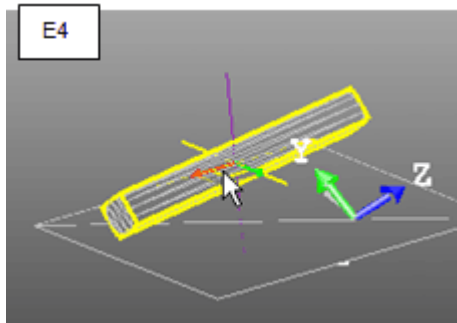
2. Pick the plane. A manipulator appears (E2). This manipulator has two axis handles (red and green) and a plane handle and has as origin the center of the picked object.



3. Select an axis handle by picking it. It turns to yellow. The direction along which you can pan the object is highlighted in yellow and those for which you cannot are in mauve.
4. Drag the axis handle to pan the object along the plane direction in yellow (E3).



5. Select the plane handle by picking it. It turns to yellow. The directions (two) along which you can pan the object are highlighted in yellow and the one for which you cannot is in mauve.
6. Drag the plane handle to pan the object along any of the plane directions (E4).



Moving a Geometry Along a User Defined Vector

You can move a geometry along a vector that you have to define by your own.






1 - Move Geometry Using 2-Point Defined Vector

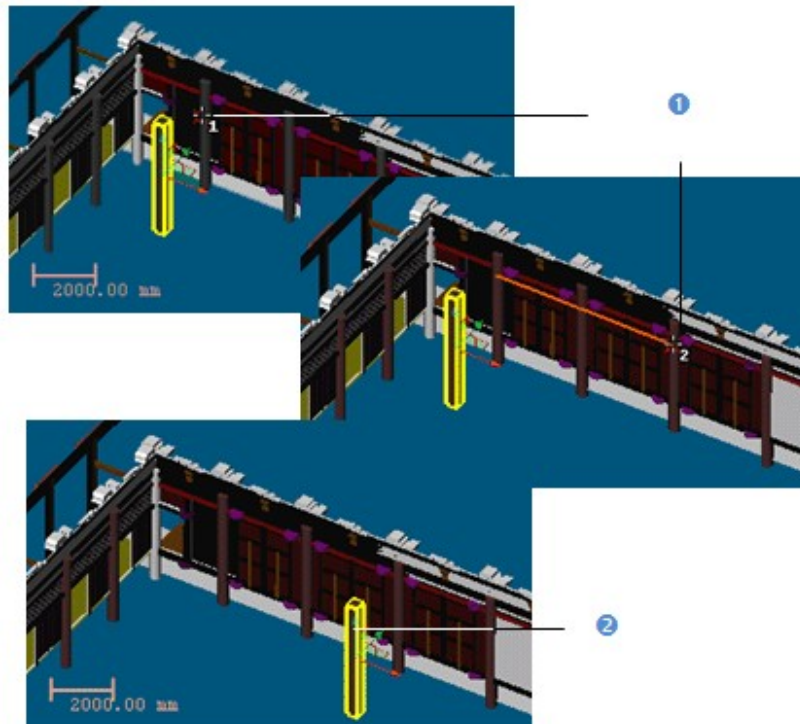
2 - Align Geometry (Z Axis) Along 2-Point Defined Vector

Note: You need select a geometry before choosing one of the above commands. Otherwise, they are dimmed. Each of the commands can be selected from the pop-up menu.

Moving a Geometry Using a 2-Point Defined Vector

After choosing the **Move Geometry using 2-Point Defined Vector**  icon, the cursor takes the shape shown in [A] and the **Picking Parameters** toolbar appears in 3D constraint mode. This means that you are in the point picking mode. After picking a point, it becomes as shown in [B]. This means that you need to pick to another point. After picking, it returns to its default shape (Arrow); this means that you are in the selection mode. With this tool, you can pan a geometry by picking two points.

[A]  [B] 






1 - The first and second picked points

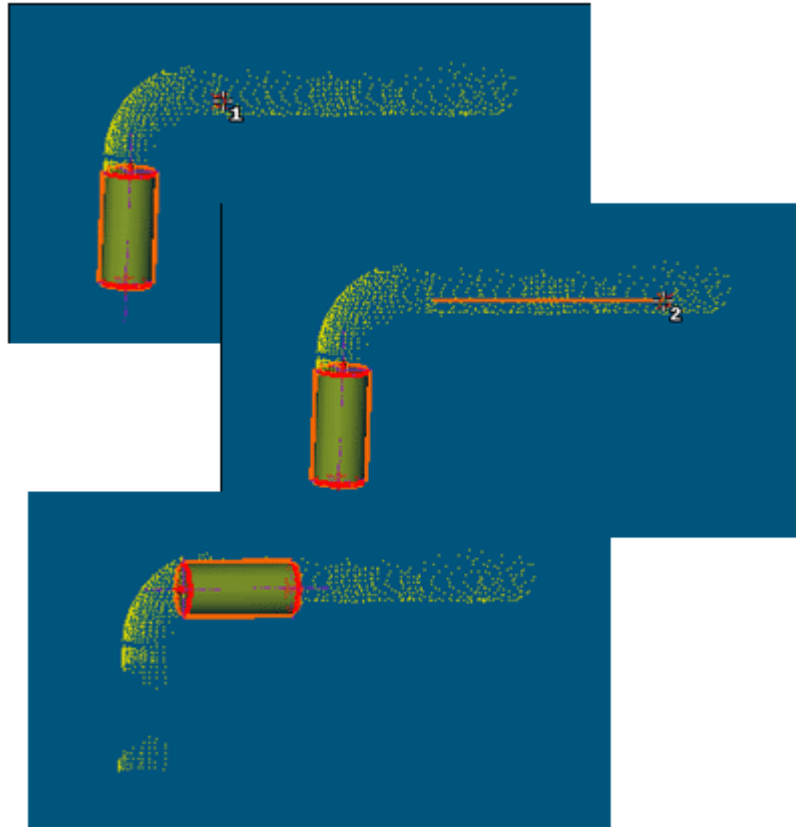
2 - The selected object is translated from the first picked point to the second picked point

Note: You can select **Cancel Picking** from the pop-up menu (or use the **Esc** key) to leave the **Move Geometry using 2-Point Defined Vector** tool.

Aligning a Geometry (Z-Axis) Along a 2-Point-Defined Axis

After choosing the **Align Geometry (Z-Axis) along 2-Point-Defined Axis**  icon, the cursor takes the shape shown in [A] and the **Picking Parameters** toolbar appears in 3D constraint mode. This means that you are in the point picking mode. After picking a point, it becomes as shown in [B]. This means that you need to pick to another point. After picking, it returns to its default shape (Arrow); this means that you are in the selection mode. With this tool, you can align an object's Z-axis along two picked points.

[A]  [B] 

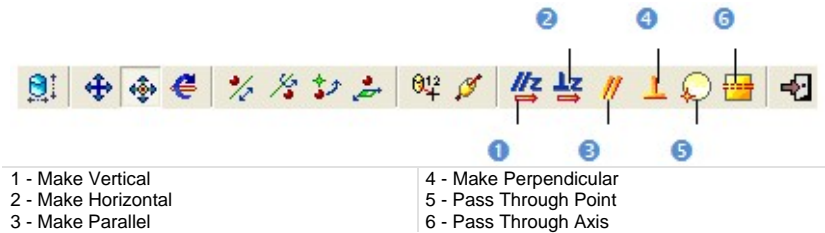


Note: You can select **Cancel Picking** from the pop-up menu (or use the **Esc** key) to leave the **Align Geometry (Z-Axis) along 2-Point-Defined Axis** tool.

Applying Constraints

Plane

After selecting a plane (before or after opening the tool), the **Geometry Modifier Tool** toolbar becomes as shown below. There are seven types of constraints.



To Make a Plane Parallel/Perpendicular to an Entity/Plane/Direction:

- See the **3D Picking Tool/3D Plane Tool/3D Direction Tool**.

To Pass a Plane Through a Point:

- See the **3D Point Tool**.

To Pass a Plane Through an Axis:

- Click **Pass through Axis**. The **3D Axis Tool** toolbar and its information box appear.
- Do one of the following to define an axis:
 - Pick an axial entity,
 - Pick two points (free or constrained),
 - Pick two planes,
 - Enter axis parameters.
- Validate the defined axis.

To Make a Plane Horizontal:

- Select a plane.
- Click **Make Horizontal**. Its **Direction of Normal** becomes parallel to the **Z Axis** (or **Elevation Axis**) but not necessary in the same direction.


To Make a Plane Vertical:

1. Select a plane.
2. Click **Make Vertical**. As the selected plane is defined by two vectors. Applying this constraint will orientate the selected plane so that one of the vectors is parallel to the **Z Axis** (or **Elevation Axis**) of the active coordinate frame.

Tip: All constraints can be selected from the pop-up menu.


Making Vertical

To Make Vertical:

1. Select a geometry.
2. Click the **Make Vertical**  icon. The selected geometry is moved so that its **Direction of Axis** is parallel to the **Z-Axis** of the **Home** frame.


Making Horizontal

To Make Vertical:

1. Select a geometry.
2. Click the **Make Horizontal**  icon. The selected geometry is moved so that its **Direction of Axis** is perpendicular to the **Z-Axis** of the **Home** frame.


Making Parallel

To Make Parallel:

1. Click the **Make Parallel**  icon. The **Make Parallel** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).

Making Perpendicular

To Make Perpendicular:

1. Click the **Make Perpendicular**  icon. The **Make Perpendicular** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Passing Through a Point

To Pass Through a Point:

1. Click the **Pass through Point**  icon. The **3D Point Tool** toolbar and its information box at the top right corner of the **3D View**.
2. Define and validate a 3D point.

Passing Through an Axis

To Pass Through an Axis:

1. Click the **Pass through Point**  icon. The **3D Axis Tool** toolbar as well as its information box appear.
2. Define and validate a 3D axis.

Sphere

After selecting a sphere-shaped geometry (before or after opening the tool), the **Geometry Modifier Tool** toolbar becomes as shown below. There are four constraint types.




1 - Lock Radius
2 - Lock Center

3 - Lock Center on a Line
4 - Pass Through a Point

Tip: All constraints can be selected from the pop-up menu.


Locking a Radius

To Lock a Radius:

1. Click the **Lock Radius**  icon. The **3D Radius Tool** toolbar opens as well as its information box.
2. Do one of the following:
 - Pick a radial entity,
 - Pick two points,
 - Pick an axis and a point,
 - Edit parameters.
3. Validate the radius.

Locking a Center

To Lock a Center:

1. Click the **Lock Center**  icon. The **3D Point Tool** toolbar opens as well as its information box.
2. Define and validate a 3D point.


Locking Center on Line

To Lock a Center on a Line:

1. Click on the **Lock Center on Line**  icon. The **3D Axis Tool** toolbar opens.
2. Define and validate a 3D axis.

Passing Through a Point

To Pass Through a Point:

1. Click the **Pass through Point**  icon. The **3D Point Tool** toolbar and its information box at the top right corner of the **3D View**.
2. Define and validate a 3D point.

Cylinder

After selecting a cylinder-shaped geometry (before or after opening the tool), the **Geometry Modifier Tool** toolbar becomes as shown below. There are six constraint types.




To Apply a Constraint to a Cylinder:

1. Make a cylinder parallel/perpendicular to an entity/plane/direction (see the **3D Picking Tool/3D Plane Tool/3D Direction Tool**).
2. Or lock a cylinder radius (see the **3D Radius Tool**).
3. Or pass a cylinder axis through a point (see the **3D Point Tool**).
4. Or fit a cylinder axis (see the **3D Axis Tool**).
5. Or make a cylinder secant to a cylinder (see the **3D Secant Tool**).
6. Or make a cylinder vertical. Its **Direction of Axis** becomes then parallel to the **Z Axis** (or **Elevation Axis**) but not necessary in the same direction.

Tip: All constraints can be selected from the pop-up menu.


Making Vertical

To Make Vertical:

1. Select a geometry.
2. Click the **Make Vertical**  icon. The selected geometry is moved so that its **Direction of Axis** is parallel to the **Z-Axis** of the **Home** frame.


Making Parallel

To Make Parallel:

1. Click the **Make Parallel**  icon. The **Make Parallel** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Making Perpendicular

To Make Perpendicular:

1. Click the **Make Perpendicular**  icon. The **Make Perpendicular** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Passing Axis Through Point

To Pass an Axis Through a Point:

1. Click the **Pass Axis Through Point**  icon. The **3D Point Tool** toolbar opens with the **Pick Point** mode is set by default.
2. Define and validate a 3D point.


Locking a Radius

To Lock a Radius:

1. Click the **Lock Radius**  icon. The **3D Radius Tool** toolbar opens as well as it information box.
2. Do one of the following:
 - Pick a radial entity,
 - Pick two points,
 - Pick an axis and a point,
 - Edit parameters.
3. Validate the radius.


Fixing to an Axis

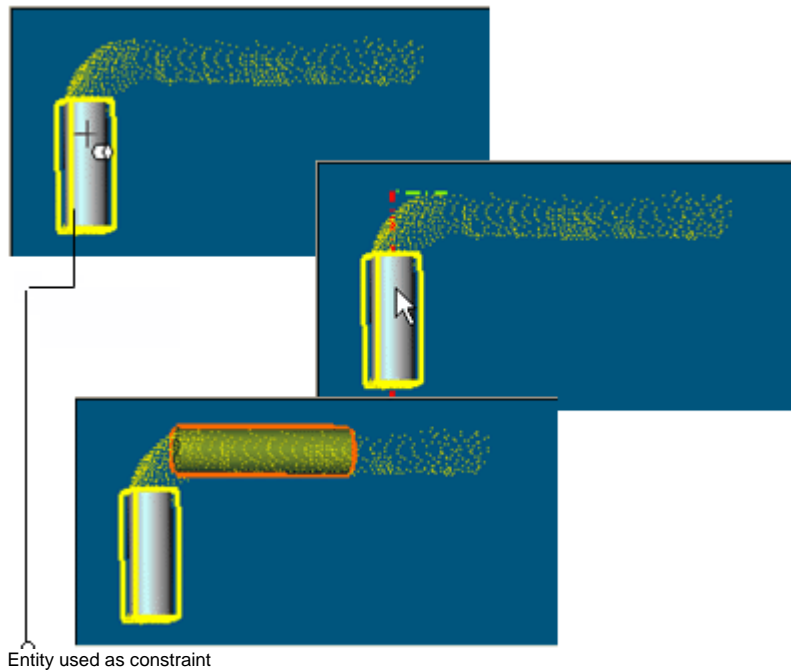
To Fix to an Axis:

1. Click on the **Fix to Axis**  icon. The **3D Axis Tool** toolbar opens.
2. Define and validate a 3D axis.

Making Secant to a Cylinder

To Make Secant to a Cylinder:

1. Click the **Make Secant to Cylinder**  icon. The **3D Secant Tool** dialog as well as its information box appears.
2. Pick a cylinder.
3. If required, check **Use Same Radius** to set the same radius than the picked cylinder.
4. If required, check **Use Given Angle** and give a secant angle.
5. If required, click **Perpendicular** to have a 90° secant angle.
6. Click **OK**.



Note that the **Make Secant to Cylinder** constraint type generates four sub-constraint types according to the option(s) checked.

- If only **Use Same Radius** has been checked, you have the two following constraint types: **Make Axis Secant to Axis** and **Lock Radius**.
- If only **Use Given Angle** has been checked and the given angle value is different to 90° and 270° , you have the two following constraint types: **Make Axis Secant to Axis** and **Lock Angle with Direction**.

- If only **Use Given Angle** has been checked and the given angle value is equal to 90° and 270° , you have the two following constraint types: **Make Axis Secant to Axis** and **Make Perpendicular to Direction**.
- If only **Use Given Angle** has been checked and **Perpendicular** pressed-on, you have the two following constraint types: **Make Axis Secant to Axis** and **Make Perpendicular to Direction**.
- If the two options have been checked with an angle other than 90° or 270° , you have the three following constraint types: **Make Axis Secant to Axis**, **Lock Radius** and **Lock Angle with Direction**.
- If the two options have been checked with an angle equal to 90° or 270° , you have the three following constraint types: **Make Axis Secant to Axis**, **Lock Radius** and **Make Perpendicular to Direction**.
- If the two options are kept unchecked, you have the **Make Axis Secant to Axis** constraint type.

Regular Cone

After selecting a cone-shaped geometry (before or after opening the tool), the **Geometry Modifier Tool** toolbar becomes as shown below. There are three constraint types.




To Apply a Constraint to a Cone:

1. Make a cone parallel/perpendicular to an entity/plane/direction (see the **3D Picking Tool/3D Plane Tool/3D Direction Tool**).
2. Or fit a cone axis (see the **3D Axis Tool**).
3. Or make a regular cone vertical. Its **Direction of Axis** becomes then parallel and in the same direction than the **Z Axis** (or **Elevation Axis**).

Tip: All constraints can be selected from the pop-up menu.


Making Vertical

To Make Vertical:

1. Select a geometry.
2. Click the **Make Vertical**  icon. The selected geometry is moved so that its **Direction of Axis** is parallel to the **Z-Axis** of the **Home** frame.


Making Parallel

To Make Parallel:

1. Click the **Make Parallel**  icon. The **Make Parallel** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).

Making Perpendicular

To Make Perpendicular:

1. Click the **Make Perpendicular**  icon. The **Make Perpendicular** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).

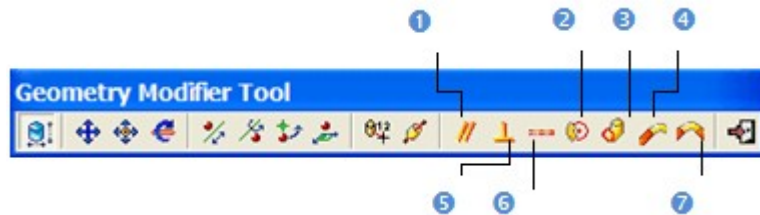
Fixing to an Axis

To Fix to an Axis:

1. Click on the **Fix to Axis** icon. The **3D Axis Tool** toolbar opens.
2. Define and validate a 3D axis.

Circular Torus

After selecting a circular torus-shaped geometry (before or after opening the tool), the **Geometry Modifier Tool** toolbar becomes as shown below. There are seven constraint types.



- | | |
|--|---|
| 1 - Make Parallel | 5 - Make Perpendicular |
| 2 - Lock Center Line Radius | 6 - Fit to Axis |
| 3 - Lock Pipe Radius | 7 - Align to two Existing Secant Cylinders of Same Radius |
| 4 - Align to Join to Existing Cylinder | |

Tip: All constraints can be selected from the pop-up menu.


Making Parallel

To Make Parallel:

1. Click the **Make Parallel** icon. The **Make Parallel** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Making Perpendicular

To Make Perpendicular:

1. Click the **Make Perpendicular**  icon. The **Make Perpendicular** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Fixing to an Axis

To Fix to an Axis:

1. Click on the **Fix to Axis**  icon. The **3D Axis Tool** toolbar opens.
2. Define and validate a 3D axis.


Locking Pipe Radius

To Lock a Pipe Radius:

1. Click the **Lock Pipe Radius**  icon. The **3D Radius Tool** toolbar opens as well as its information box.
2. Do one of the following:
 - Pick a radial entity,
 - Pick two points,
 - Pick an axis and a point,
 - Edit parameters.
3. Validate the radius.


Locking Center Line Radius

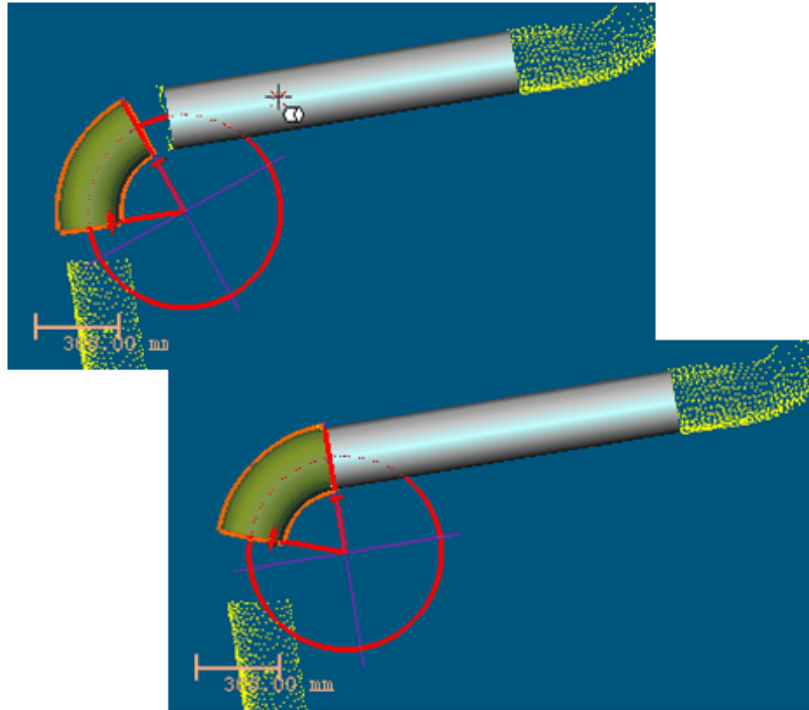
To Lock a Center Line Radius:

1. Click the **Lock Center Line Radius**  icon. The **3D Radius Tool** toolbar opens as well as its information box.
2. Do one of the following:
 - Pick a radial entity,
 - Pick two points,
 - Pick an axis and a point,
 - Edit parameters.
3. Validate the radius.

Aligning to Join to an Existing Cylinder


To Align to Join to an Existing Cylinder:

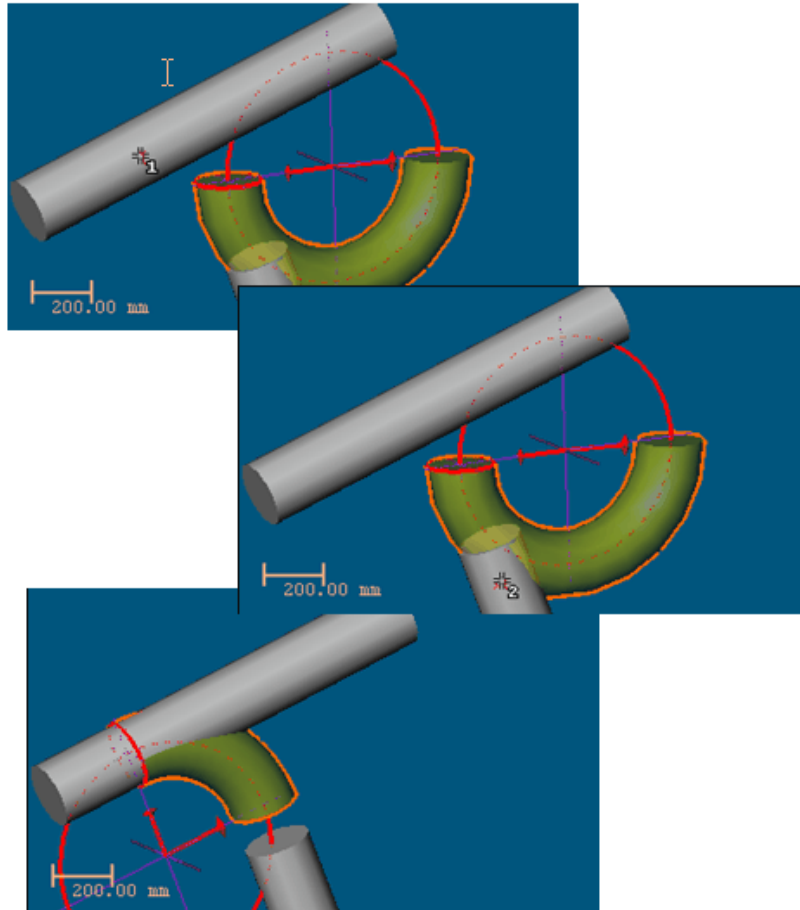
1. Click the **Align to Join to Existing Cylinder**  icon.
2. Pick an existing cylinder.



Aligning to Join to Two Secant cylinders of Same Radius

To Align to Join to Two Secant Cylinders of Same Radius:

1. Click the **Align to Join to Two Secant Cylinders of Same Radius**  icon.
2. Pick two existing cylinders. The result is null if the two picked cylinders are not secant or do not have the same diameter.



3D Point


After selecting a 3D point (before or after opening the tool), the **Geometry Modifier Tool** toolbar becomes as shown below. There are two constraint types.



Tip: All constraints can be selected from the pop-up menu.


Locking on Plane

To Lock on a Plane:

1. Click the **Lock on Plane**  icon. The **3D Plane Tool** toolbar opens with the **Pick Entity with Direction** mode is set by default.
2. Define and validate a 3D plane.

Locking to Line (or Axis)

To Lock on a Plane:

1. Click the **Lock to Line (or Axis)**  icon. The **3D Axis Tool** toolbar opens with the **Pick Axis Entity** mode is set by default.
2. Define and validate a 3D direction.

Segment

After selecting a segment (before or after opening the tool), the **Geometry Modifier Tool** toolbar becomes as shown below. There are four constraint types.




To Apply a Constraint to a Segment:

1. Make a segment parallel/perpendicular to an entity/plane/direction (see the **3D Picking Tool**/**3D Plane Tool**/**3D Direction Tool**).
2. Or lock a segment on a plane (see the **3D Plane Tool**).
3. Or pass a segment through a point (see the **3D Point Tool**).
4. Or make a segment vertical. Its **Direction of Axis** becomes then parallel to the **Z Axis** (or **Elevation Axis**)).

Tip: All constraints can be selected from the pop-up menu.


Making Vertical

To Make Vertical:

1. Select a geometry.
2. Click the **Make Vertical**  icon. The selected geometry is moved so that its **Direction of Axis** is parallel to the **Z-Axis** of the **Home** frame.


Locking on Plane

To Lock on a Plane:

1. Click the **Lock on Plane**  icon. The **3D Plane Tool** toolbar opens with the **Pick Entity with Direction** mode is set by default.
2. Define and validate a 3D plane.


Making Parallel

To Make Parallel:

1. Click the **Make Parallel**  icon. The **Make Parallel** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).


Making Perpendicular

To Make Perpendicular:

1. Click the **Make Perpendicular**  icon. The **Make Perpendicular** toolbar opens with three tabs (**By Picking Entity**, **To Plane** and **To Direction**) inside. Each tab corresponds to a tool (respectively **3D Picking**, **3D Plane** and **3D Direction**). The **By Picking Entity** tab opens first, its information box takes place at the top right corner of the **3D View** and the **Pick Entity with Direction** mode is set by default.
2. Do one of the following.
 - Define a plane,
 - Define a direction,
 - Pick an entity with direction.
3. Validate the defined plane (or direction).

Passing Axis Through Point

To Pass an Axis Through a Point:

1. Click the **Pass Axis Through Point**  icon. The **3D Point Tool** toolbar opens with the **Pick Point** mode is set by default.
2. Define and validate a 3D point.

Extrusion

After selecting an extrusion (before or after opening the tool), the **Geometry Modifier Tool** toolbar becomes as shown below. There are two constraint types.



1 - Make Vertical

2 - Make Parallel to Direction

2 - Make Perpendicular to Plane


To Apply Constraint to an Extruded Entity:

1. Make an extruded entity parallel to a direction (see the **3D Direction Tool**).
2. Or make an extruded entity perpendicular to a plane (see the **3D Plane Tool**).
3. Or make an extruded entity vertical. Its **Direction of Axis** becomes then parallel to the **Z Axis** (or **Elevation Axis**)).

Tip: All constraints can be selected from the pop-up menu.

Making Vertical

To Make Vertical:

1. Select a geometry.
2. Click the **Make Vertical**  icon. The selected geometry is moved so that its **Direction of Axis** is parallel to the **Z-Axis** of the **Home** frame.

Rectangular Torus

To Apply Constraints to a Rectangular Torus:

1. Select a rectangular torus. The **Geometry Modifier Tool** toolbar becomes as shown below. There are three constraint types (detailed below).



Align to Join to Two Existing Secant Boxes of Same Direction

2. Align to join two existing secant boxes of same section.
3. Click **Close**. The **Geometry Modifier Tool** toolbar closes by its own.

Tips:

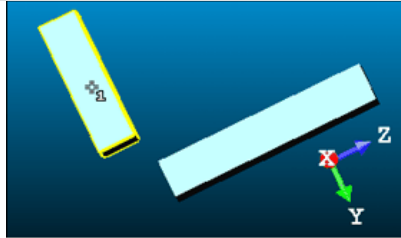
- The **Align to Join to Two Existing Secant Boxes of Same Direction** constraint can also be selected from the pop-up menu.
- Selecting a rectangular-torus-shaped geometry can be done before (or after) opening the **Geometry Modifier Tool**.

Aligning to Join Two Existing Secant Boxes of Same Section

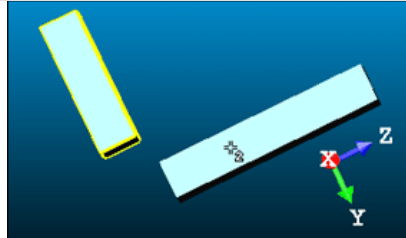
To Align to Join to Two Existing Secant Boxes of Same Section:

1. Click the **Align to Join to Two Existing Secant Boxes of Same Section** icon. The cursor becomes as shown in [A].
2. Pick a first box. The cursor takes the shape shown in [B].

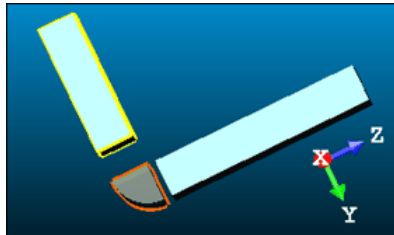
[A]



[B]



3. Pick another box. If the two boxes are secant and have the same section, a rectangular torus appears.



- Its **Direction of Normal** (also called **Direction of Axis**) is parallel to the two boxes' **Direction of Normal** (also called **Direction of Width**).
 - Its **Bend Angle** is equal to the angle drawn by the two boxes' **Direction of Height**.
 - Its **Outer Diameter** is equal to the two boxes' **Depth**.
4. If the two picked boxes are not secant; the "This constraint cannot be activated because the two boxes are not secant" warning message appears. Click **OK**. The warning message closes and the **Align to Join to Two Existing Secant Boxes of Same Section** constraint is left.
 5. If the two picked boxes do not have the same section, the "There is too much indetermination to activate this constraint: two boxes are identical, one of them is a cube or they have no common face" warning message appears. Click **OK**. The warning message closes and the **Align to Join to Two Existing Secant Boxes of Same Section** constraint is left.

Tips:

- If required, make the two boxes secant using the **Make Secant to a Box (With Same Section)** constraint in the **Geometry Modifier Tool**.
- If required, modify manually the two boxes' parameters (like **Center**, **Width**, **Height**, **Depth**, **Directions**, etc.) in the **Property** window to make sure that both are secant and have the same section.

Box

To Apply Constraints to a Box:

1. Select a box. The **Geometry Modifier Tool** toolbar becomes as shown below. There are three constraint types (detailed below).



1 - Make Parallel
2 - Stick to Plane

3 - Make Secant to Box (With Same Section)


2. Make parallel to a direction.
3. Or project a box onto a plane.
4. Or make secant to a box of same section.
5. Click **Close**. The **Geometry Modifier Tool** toolbar closes on its own.

Tips:

- All constraints can be selected from the pop-up menu.
- Selecting a box-shaped geometry can be done before (or after) opening the **Geometry Modifier Tool**.

Making Parallel to a Direction

To Make Parallel to a Direction:

1. Click **Make Parallel** . The **Make Parallel** toolbar opens with the **By Picking Entity** tab selected by-default. In that tab, the **Pick an Entity With Direction** is set by-default.
2. Do one of the following:
 - Pick an entity using the **Pick an Entity With Direction** tool in the **By Picking Entity** tab.
 - Define a plane using available tools in the **To a Plane** tab*.
 - Define a direction using available tools in the **To a Direction** tab*.


The **Normal Z** (also called the **Direction of the Height** in the **Property** window) of the box is parallel** to the **Direction of Axis** of the picked entity (or to the **Direction of Normal** of the defined plane or to the defined direction).

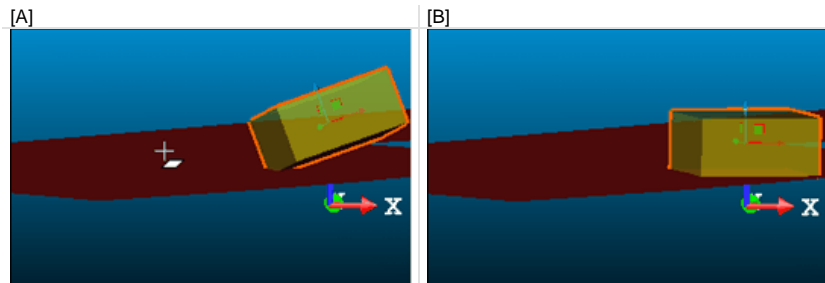
Notes:

- (*) First click on the corresponding tab.
- (**) But not necessary in the same direction.

Projecting a Box onto a Plane

To Project a Box in a Plane:

1. Click **Stick to Plane** . The cursor becomes as shown in [A].
2. Pick a plane. The selected box is projected on the picked plane [B]. The bottom side (of the box) lies on the plane.



The **Direction of Normal** (of the plane) [C] and the **Direction of the Height** (also called the **Normal Z*** of the box) [D] are parallel but not necessarily in the same direction. In the example below, both are opposite.


[C]		[D]	
Properties		Properties	
<div> <div>General</div> <div>TypePlane - Fitted</div> <div>NameOBJECT498</div> </div>		<div> <div>General</div> <div>TypeBox</div> <div>NameOBJECT1979</div> </div>	
<div> <div>Geometry</div> <div>Color of GeometryRGB(255,0,0)</div> <div>Center84.95 m; 206.89 m; -</div> <div>Direction of Normal-0.01; 0.01; -1.00</div> </div>		<div> <div>Geometry</div> <div>Color of GeometryRGB(192,192,192)</div> <div>Center77.40 m; 180.76 m;</div> <div>Width8.94 m</div> <div>Height4.27 m</div> <div>Depth8.01 m</div> <div>Direction of Width0.92; 0.39; -0.00</div> <div>Direction of Height0.01; -0.01; 1.00</div> <div>Direction of Depth-0.39; 0.92; 0.01</div> </div>	
<div> <div>Bounds</div> <div>N° Holes0</div> </div>		<div> <div>Bounds</div> </div>	

Notes:

- Press **Esc** (or select **Cancel Picking** from the pop-up menu) to leave the picking mode.
- (*) In the **X, Y, Z Coordinate System**.

Making Secant to a Box (With Same Section)

To Make Secant to a Box (With Same Section):

1. Click **Make Secant to a Box (With Same Section)** .
2. Pick a box having the same section as the selected one.
3. If the picked box does not have the same section, the "There is too much indetermination to activate this constraint: two boxes are identical, one of them is a cube or they have no common face" warning message appears. Click **OK**. The warning message closes and the **Make Secant to a Box (With Same Section)** constraint is left.

Duplicator Tool

The **Duplicator Tool** enables to duplicate a geometry along (or around) a path defined by the user. A path can be a line, a circle or a combination of both (called polyline). This tool requires a selection as input. If the input is an object from the database (already created), the tool can be used as a main tool. When the object is e.g. being created, the tool is a sub-tool inside a main tool like e.g. the **Geometry Creator**.

Opening the Tool

To Open the Tool:

1. Select a geometry from the **Project Tree**.
2. From the **Modeling** menu, select **Duplicator Tool** . The **Duplicator** dialog opens as the third tab of the **WorkSpace** window.

Tip: You can also click on the **Duplicator Tool** icon in the **Tools** toolbar.

Choosing a Method

There are three duplication methods: **Along a Line**, **Around an Axis** and **Along a Polyline**. Inside each method, you need to define a path along (or around) which the duplication will be carried out. The method which comes first is the one set during the last use of that tool.

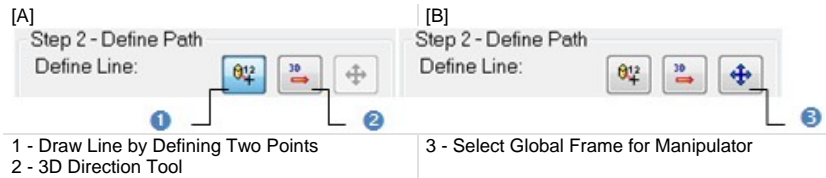
To Choose a Duplication Method:

1. In the **Choose Method** panel, click on the pull-down arrow.
2. Choose among **Along a Line**, **Around an Axis** and **Along a Polyline** from the drop-down list.

Tip: All duplication methods can be selected from the pop-up menu. First select **Choose Type of Path**.

Along a Line

Before defining a path, **Step 2** of the **Duplicator** dialog takes the appearance shown below in [A]. After defining a path, the **Select Global Frame for Manipulator** icon becomes enabled; it enables to switch from a one-handle manipulator to a three-handle manipulator (see [B]).



To Define a Line:


1. Define a line by picking two points.
2. Or define a 3D direction.

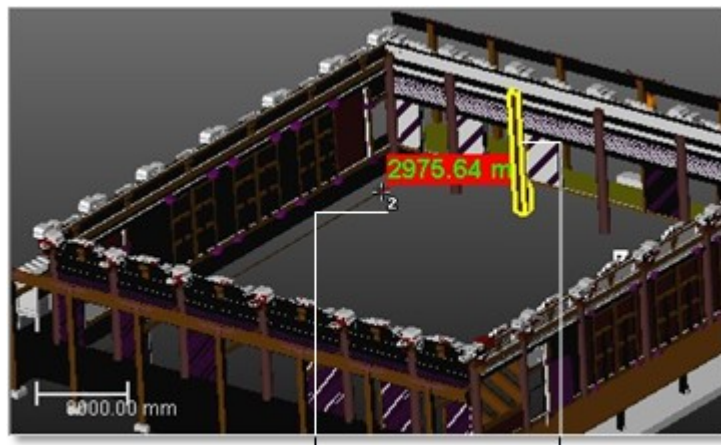
Tip: The **Select Global Frame for Manipulator** can also be selected from the pop-up menu.

Drawing a Line by Defining Two Points

The **Draw Line by Defining Two Points** icon (set by-default) enables to define a path of segment shape by picking two points. The first point is always at the center of the selected item. The second point with the first point defines a path used for duplication.

To Define Two Points:

1. Click on the **Draw Line by Defining Two Points**  icon. The **Picking Parameters** toolbar appears in 3D constraint mode.
2. Pick a point (free or constrained) on displayed items. A **Red Line** linking the first picked point to the cursor appears. This **Red Line** has a label in red showing the distance from the first picked point to the cursor's current position.
3. Pick another point (free or constrained), always on displayed items.

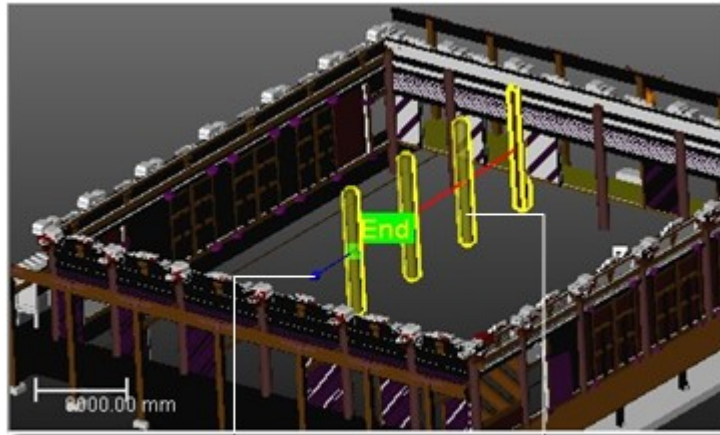


1

2

1 - The selected item

2 - The second picked point



1 - Manipulator

4 - Duplicated items


The distance between the two picked points sets the **Step** value (see **Step 3** of the **Duplicator** dialog). The selected item is duplicated according to the parameters set in **Step 3** along the defined path. The last duplicated item has a **Manipulator** and an **End** label at its center.

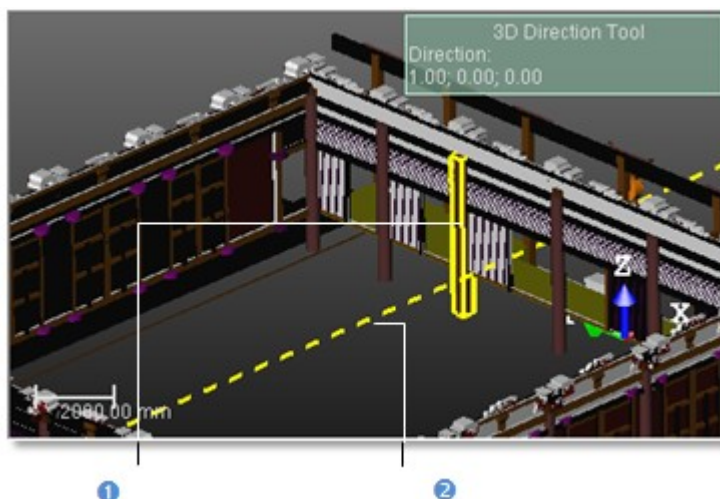
Tip: The **Draw Line by Defining Two Points** icon can also be selected from the pop-up menu.

Defining a 3D Direction

The **3D Direction Tool** icon enables to define a 3D direction.

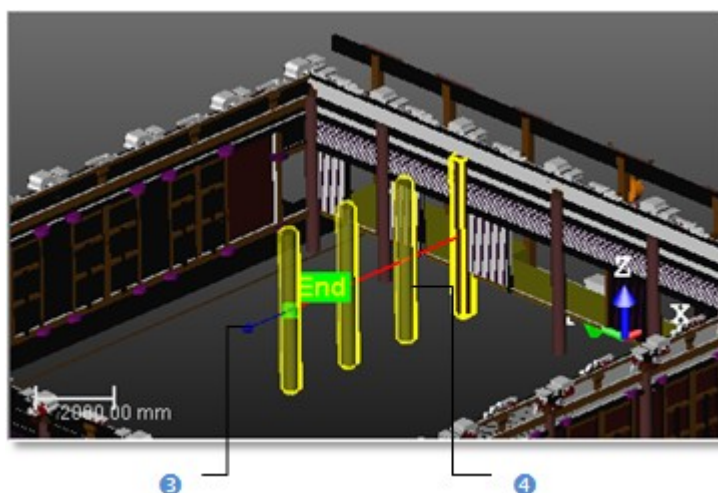
To Define a 3D Direction:

1. Click on the **3D Direction Tool**  icon. The **3D Direction Tool** toolbar appears as well as its information box and a yellow dotted direction.
2. Define a 3D direction and validate it.



1 - The selected item

2 - The defined direction



3 - The one-handle manipulator

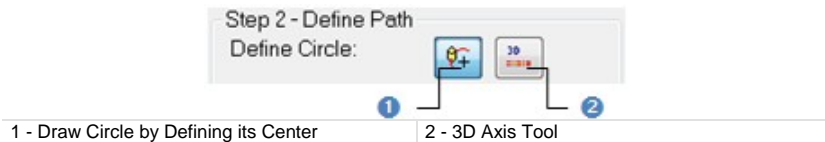
4 - The duplicated items

A **Red Line** starting from the selected item and running parallel to the defined 3D direction appears. The selected item is duplicated according to the parameters set in **Step 3** along the defined path. The last duplicated item has a **Manipulator** and an **End** label at its center.

Tip: The **3D Direction Tool** can also be selected from the pop-up menu.

Around an Axis

After choosing **Around an Axis**, you need to indicate the displacement mode for duplication by checking either the **All Parallel** option or the **All Rotated** option. **Step 2** of the **Duplicator** dialog becomes as shown below.



1 - Draw Circle by Defining its Center

2 - 3D Axis Tool

To Define a Circle:

1. Draw a circle by defining its center.
2. Or define a 3D axis.

This **Red Circle** may have two shapes (dotted and/or continuous arc) with an arrow as duplication direction. The initial item is duplicated around that **Red Circle** according to the parameters in **Step 3**.


If **All Parallel** has been checked, all newly duplicated items have the same direction as the first one.

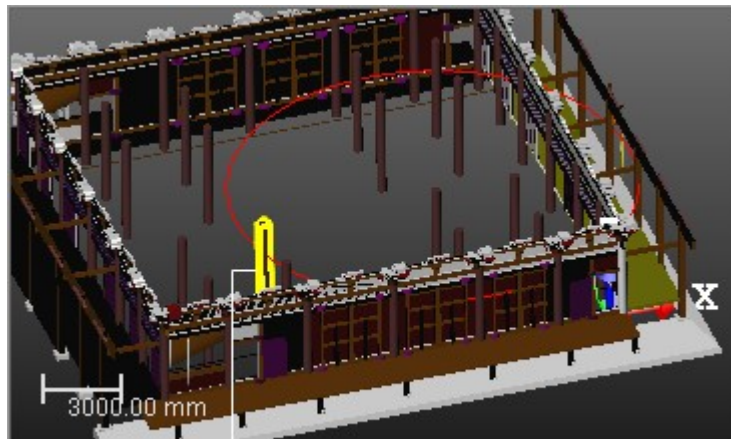
If **All Rotated** has been checked, each newly duplicated item has its own direction. The initial (selected) item still remains selected and the last duplicated item has an **End** label at its center.

Drawing a Circle

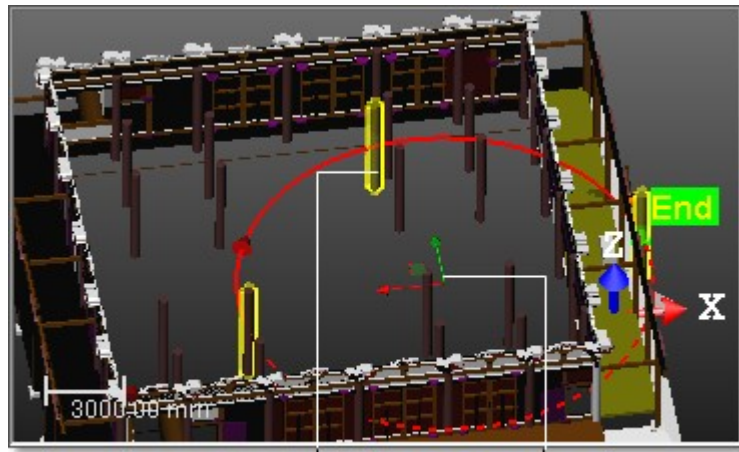
The **Draw Circle by Defining its Center** icon (set by-default) enables to define a circular path by its circle. This circular path has as **Normal** the direction perpendicular to screen view.

To Draw a Circle by Defining its Center:

1. If required, bring the scene to the **Top** view.
2. Click **Draw Circle by Defining its Center** . The **Picking Parameters** toolbar appears in the 3D constraint mode. The cursor takes the shape shown below. A **Red Circle** in the screen plane appears. It has as **Origin** the cursor current position and passes through the initial item's center.
3. Pick a point (free or constrained) anywhere - not necessary on displayed items.



The selected item



1 - The two handle manipulator

2 - Duplicated items

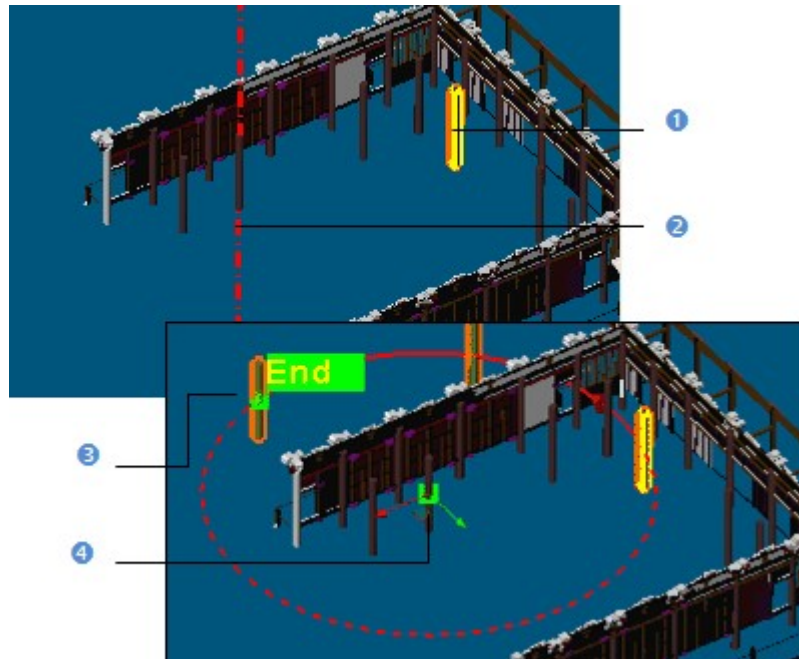
Tip: The **Draw Circle by Defining its Center** can also be selected from the pop-up menu.

Defining a 3D Axis

The **3D Axis Tool** icon enables to define an axis which will be used the **Normal** direction of a circular path.

To Define a 3D Axis:

1. Click on the **3D Axis Tool** icon. The **3D Axis Tool** toolbar appears as well as the **3D Axis Tool** information box.
2. Define a 3D axis. A **Red and Dotted Segment** appears.
3. Validate the 3D axis. The **Red and Dotted Segment** disappears. A **Red Circle** appears with a **Red and Green Manipulator** along and in the middle of the defined axis.



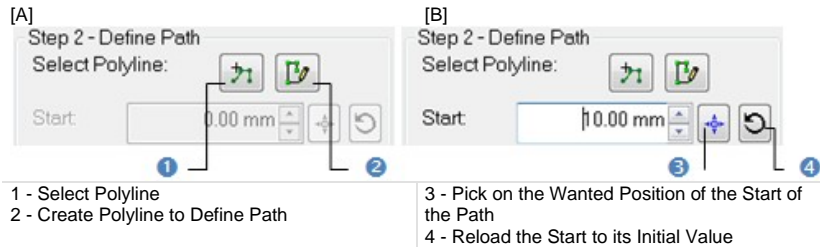
1 - The selected item
2 - The defined direction

3 - Duplicated items
4 - Manipulator


Tip: The **3D Axis Tool** icon can also be selected from the pop-up menu.

Along a Polyline

Before selecting (or defining) a polyline, **Step 2** of the **Duplicator** dialog takes the appearance shown in [A]. After selecting (or defining) a polyline, **Pick on the Wanted Position of the Start of the Path** and **Reload the Start to its Initial Value** become active (see [B]).



To Define a Polyline:


1. Select an existing polyline.
2. Or create a new polyline.
3. Enter a value distance value in the **Start** field.
4. Or click **Pick on the Wanted Position of the Start of the Path** .
5. Go the **3D View** and pick a point on the selected/drawn polyline. The **Start** position changes on the path.
6. If required, get back the **Start** position by clicking **Reload the Start Position**.

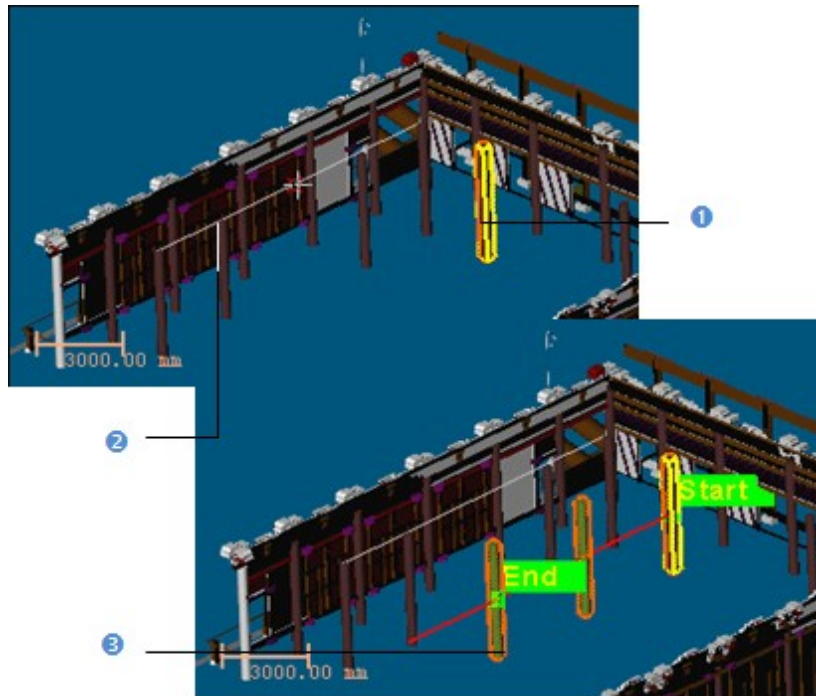
The initial (selected) item is duplicated according to the parameters set in **Step 3** along the defined path (in red). Starting a new polyline selection (or a new polyline drawing) will undo the duplication. Duplicated items except the initial (selected) one are removed from the **3D View**.

Selecting a Polyline

The **Select Polyline** icon enables to select a path of segment (or a combination of segment and circular arc) shape - mainly polyline - if present in your project.

To Select a Polyline:

1. Click **Select Polyline** . The cursor takes the shape as shown below.
2. Pick a polyline to select it.



1 - The selected item

2 - An existing polyline

3 - The duplicated items


A red path of the same shape as the polyline starting from the initial item's center appears. The **Start** and **End** positions on the path are indicated with a label. It's up to you to change the **Start** position.

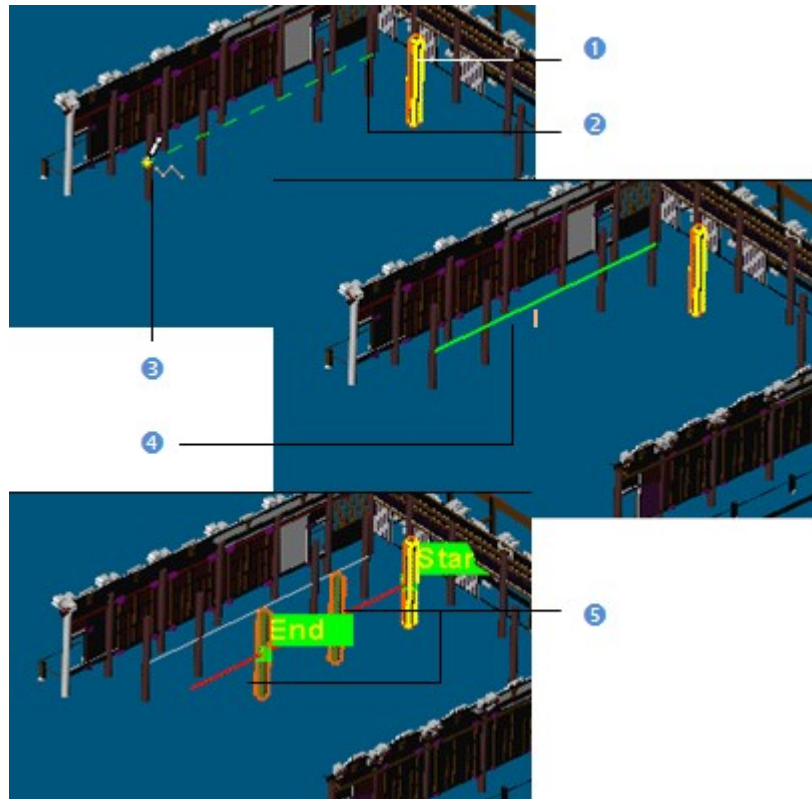
Tip: The **Select Polyline** icon can also be selected from the pop-up menu.

Creating a Polyline

The **Create Polyline to Define Path** icon enables to draw and create a polyline.

To Create a Polyline:

1. Click **Create Polyline to Define Path** . The **Drawing** and **Picking Parameters** (in 3D constraint mode) toolbars appear.
2. Draw a polyline by picking points (free or constrained).
3. Validate the polyline.



- 1 - The selected item
2 - The first picked point
3 - The first second point

- 4 - The drawn polyline
5 - The duplicated items

A red path of the same shape as the polyline starting from the initial item's center appears. The **Start** and **End** positions on the path are indicated with a label. It's up to you to change the **Start** position.

Tip: The **Create Polyline to Define Path** icon can also be selected from the pop-up menu.

Resizing a Path

You can use the **Manipulator** to resize the path previously defined. If the path is a **Red Circle**, the **Manipulator** has two **Axis Handles** (**Red** and **Green**) and on **Plane Handle**. You can enlarge (or reduce) the **Red Circle**'s diameter by displacing its center in a direction using an **Axis Handle** or in an arbitrary position using the **Plane Handle**. For both, the displacement is done on the **Red Circle**'s plane.

If the path is a **Red Line**, the **Manipulator** has only one **Axis Handle** (**Green**). You can extend (or shorten) the **Red Line** along the defined direction by using the **Handle Axis**. But you can also use the **Select Global Frame for Manipulator** which has three **Axis Handles** (**Green**, **Red** and **Blue**) and three **Plane Handles**. In that case, you extend (or shorten) the **Red Line** not along the defined path but along the direction indicates by the **Axis Handle** or on the plane defined by a pair of **Axis Handle**. If the path is a **Red Polyline**, there is no **Manipulator**.

Note: If you start defining a new path; this cancels the current duplication. Duplicated items except the selected item are removed from the **3D View**.

Defining Parameters

There are three sets of parameters available for duplicating items: **Step & Quantity**, **Step & Length** and **Length & Quantity**. The **Step** parameter corresponds to the distance (or angle) between two successive items. The **Length** corresponds to the distance (or angle) from the **Start** position to the **End** position.

To Define Parameters:




1. In the **Define Parameters** panel, click on the pull-down arrow.
2. Choose among **Step & Quantity**, **Step & Length** and **Length & Quantity**.

Step & Quantity

After choosing **Step & Quantity**, **Step 3** of the **Duplicator** dialog changes its appearance according to the duplication method set in **Step 1**.

Along a Line (or Along a Polyline)	Around an Axis
Quantity: <input type="text" value="3"/>	Quantity: <input type="text" value="3"/>
Step: <input type="text" value="13792.51 mm"/>	Step: <input type="text" value="180.00 °"/>
Length: 41377.52 mm	Length: 360.00 °







To Define the Step & Quantity Parameters:

1. Enter a number in the **Quantity** field.
2. Or use  (or ) to set a value in the **Quantity** field.
3. If **Along a Line** (or **Along a Polyline**) has been selected, enter a distance value in the **Step** field.
4. If **Around an Axis** has been selected, enter an angular value in the **Step** field.
5. Or click **Pick on the Wanted Position of the Next Element** .
6. Go to the **3D View** and pick a point on the path.




Note: The **Length** value is automatically updated according to the value set in the **Quantity** (or **Step**) field.

Step & Length

After choosing **Step & Length**, **Step 4** of the **Duplicator** dialog changes its appearance according to the duplication method selected in **Step 1**.

Along a Line (or Along a Polyline)	Around an Axis
Quantity: 3	Quantity: 3
Step: 13792.51 mm 	Step: 180.00° 
Length: 41377.52 mm  	Length: 360.00°  





To Define the Step & Length Parameters:

1. If **Along a Line / Along a Polyline** (or **Around an Axis**) has been selected, enter a distance (or angular) value in the **Step** field.
2. Or click **Pick on the Wanted Position of the Next Element** .
3. Go to the **3D View** and pick a point on the path.
4. If **Along a Line / Along a Polyline** (or **Around an Axis**) has been selected, enter a distance (or angular) value in the **Step** field.
5. Or click **Pick on the Wanted Position of the End of the Path** .
6. Go to the **3D View** and pick a point on the path.
7. If required, click **Reload the Path Length to its Initial Value** .





Note: The **Quantity** value is automatically updated according to the value set in the **Step** (or **Length**) field.

Length & Quantity

After choosing **Length & Quantity**, **Step 3** of the **Duplicator** dialog changes its appearance according to the duplication type selected in **Step 1**.

Along a Line (or Along a Polyline)	Around an Axis
Quantity: <input type="text" value="3"/>	Quantity: <input type="text" value="3"/>
Step: 996.67 mm	Step: 180.00 °
Length: <input type="text" value="2990.00 mm"/>  	Length: <input type="text" value="252.00 °"/>  

To Define the Length & Quantity Parameters:

1. Enter a number in the **Quantity** field.
2. Or use  (or ) to set a value in the **Quantity** field.
3. If **Along a Line** (or **Along a Polyline**) has been selected, enter a distance value in the **Length** field.
4. If **Around an Axis** has been selected, enter an angular value in the **Length** field.
5. Or click **Pick on the Wanted Position of the End of the Path** .
6. Go to the **3D View** and pick a point* on the path.
7. If required, click **Reload the Path Length to Its Initial Value** .

Notes:

- The **Step** value is automatically updated according to the value set in the **Quantity** (or **Length**) field.
- (*) To leave the picking mode, select **Cancel Picking** from the pop-up menu or use **Esc**. Outside the picking mode, use **Esc** to leave the **Duplicator Tool**.

Reversing the Path Direction

You can use **Invert Path Direction** to reverse the duplication direction. If **Along a Line** has been selected, the **End** position will be moved at the opposite end along the path. If **Around an Axis** has been selected, the **End** position remains in the same position but the duplication direction changes. If **Along a Polyline** has been selected, the **Start** and **End** positions are inverted.

Tip: **Invert Path Direction** can also be selected from the pop-up menu.

Duplicating Items

After defining a path and setting parameters, you can create the duplicated items as persistent objects in the database. All are gathered in a folder named **Duplication** which is put under the active group. Each duplicated item has the name set in the **Prefix Name** field with an order.

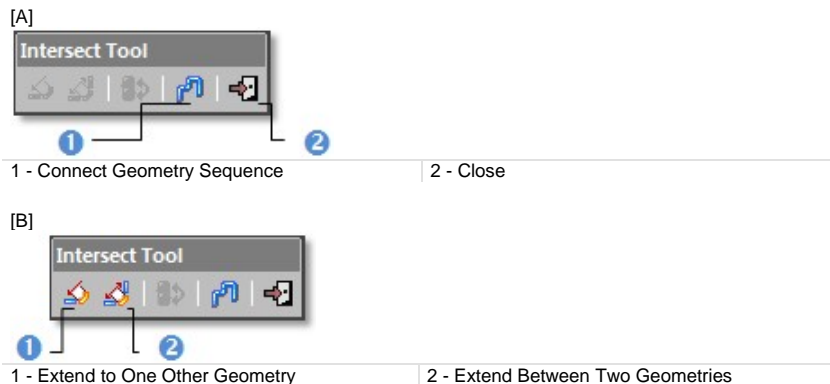
Note: You can duplicate as many items as required without leaving the **Duplicator Tool**.

Intersect Tool


Intersecting an entity with another entity is similar to bound the first with the second. Mainly entities of the following shapes (circular torus, cone, cylinder, extruded geometry, plane and sphere) can be intersected. The first entity will be modified after the intersection and the second entity remains unchanged. Intersecting a series of entities together is similar to bound together. All will be modified after intersecting. The **Intersect Tool** can be used alone or inside main tools like the **Cloud-Based Modeler Tool**.

Opening the Tool

No selection is required to open the tool. Once inside the tool, a selection must be carried out. In [A], the **Intersect Tool** opens as a toolbar when there is no input. In [B], the tool opens with an input.



To Open the Tool:

1. Select an entity from the **Project Tree** if required.
2. From the **Modeling** menu, select **Intersect Tool** . The **Intersect Tool** toolbar appears.

Tips:

- You can also click on the **Intersect Tool** icon in the **Tools** toolbar.
- All commands can be selected from the pop-up menu.

Connecting Two/Three Entities

Use the "Extend to One Other Geometry" feature to intersect an entity with another entity. The table A lists in a column all entities for which the intersection is allowed and in a line with which entity an entity can be intersected. Use the "Extend between Two Other Geometries" feature to intersect an entity with two entities. The table B lists in a column all entities for which the intersection is allowed and with which entities an entity can be intersected. Some constraints may be observed when intersecting entities together. They are detailed below.

Table A:

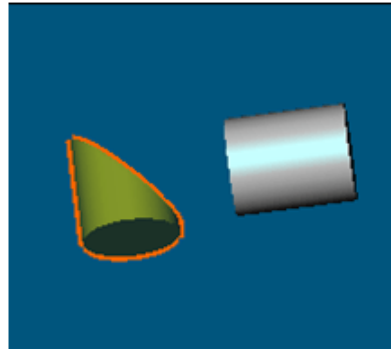
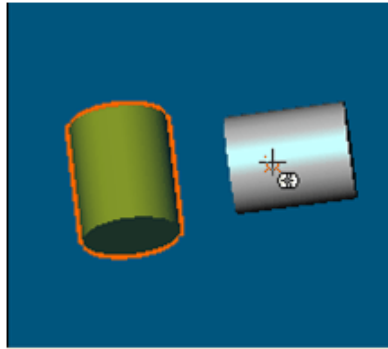
	Circular Torus	Cone	Cylinder	Extrusion	Plane	Sphere
Circular Torus			X		X	
Cone		X	X		X	
Cylinder	X	X	X		X	X
Extrusion					X	
Plane	X	X	X	X	X	X
Sphere			X		X	

Table B:

	Circular Torus	Cone	Cylinder	Extrusion	Plane	Sphere
Circular Torus			X			
Cone			X		X	
Cylinder	X	X	X		X	
Extrusion					X	
Plane					X	
Sphere					X	

Cylinder to Cylinder

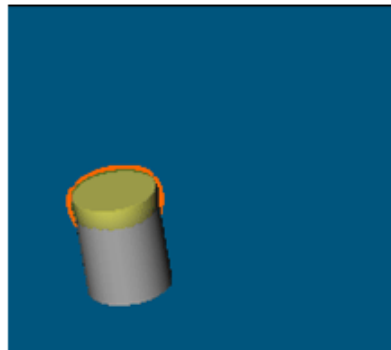
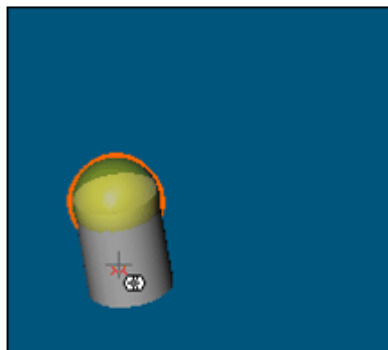
When intersecting two cylinders together; both cylinders need to have secant axes and same radius. If these prerequisites are not observed; open the **Geometry Modifier Tool** and apply the following constraints - **Lock Radius** and **Make Perpendicular** or **Make Secant to Cylinder** - otherwise a warning message appears.



Cylinder to Sphere

When intersecting a cylinder with a sphere or vice versa; the sphere's center needs to be on the cylinder's axis. If this prerequisite is not observed; open the **Geometry Modifier Tool** and apply the constraint below:

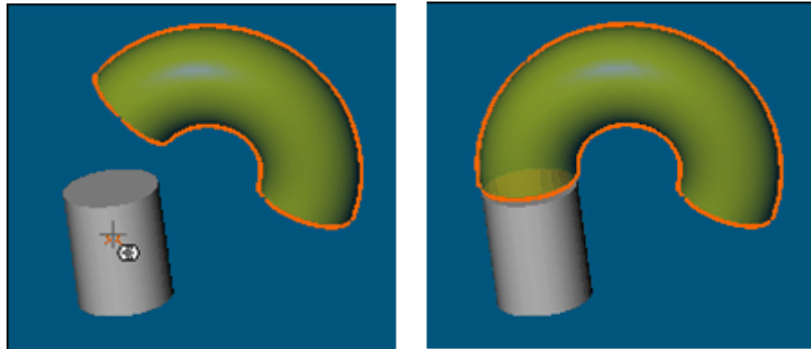
- **Lock Center on Line** to the sphere when intersecting it with the cylinder,
- **Pass Axis through Point** to the cylinder when intersecting it with the sphere,



Otherwise, a warning message appears.

Cylinder to Circular Torus

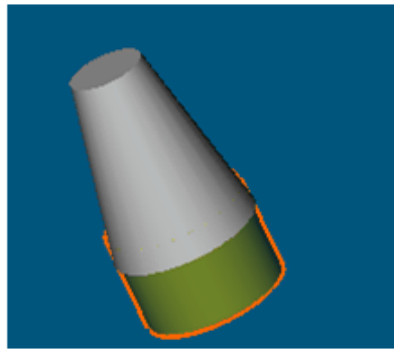
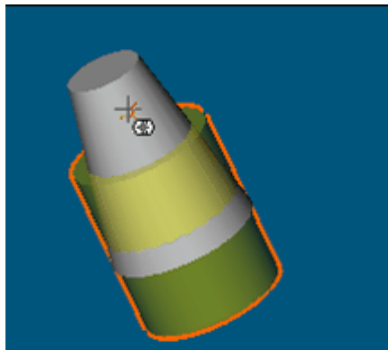
When intersecting a cylinder with a circular torus or vice versa; the cylinder needs to have the same radius as the circular torus. If this prerequisite is not observed; open the **Geometry Modifier Tool** and apply the constraint **Align to Join to Existing Cylinder** to the torus. Otherwise, a warning message appears.



Cylinder to Cone

When intersecting a cylinder with a cone or vice versa; both need to have the same axis. If this prerequisite is not observed; open the **Geometry Modifier Tool** and apply the constraint below:

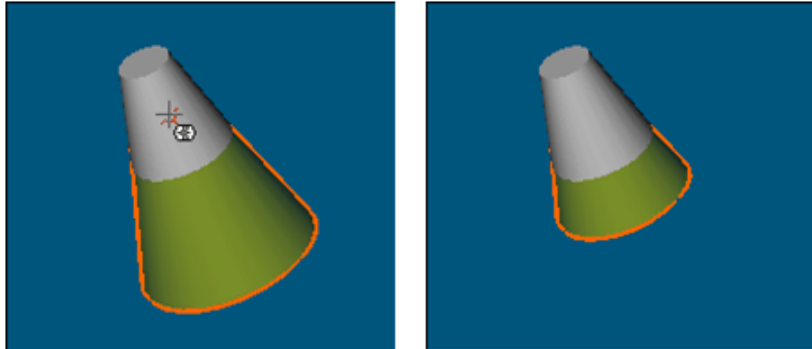
- **Make Parallel** and **Fit to Axis** to the cylinder when intersecting it with the cone,
- **Make Parallel** and **Fit to Axis** to the cone when intersecting it with the cylinder,



Otherwise, a warning message appears.

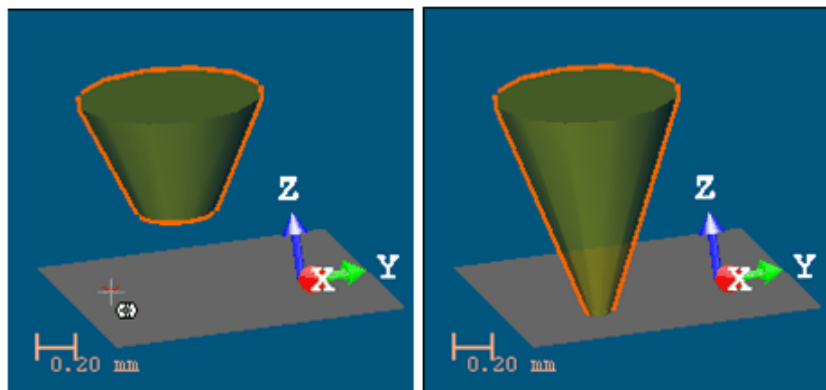
Cone to Cone

When intersecting two cones together; both need to have the same axis. If this prerequisite is not observed; open the **Geometry Modifier Tool** and apply the following constraint - **Fit to Axis** - otherwise a warning message appears.



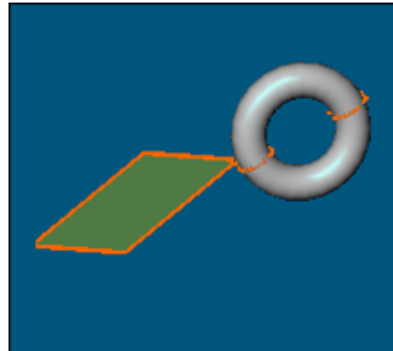
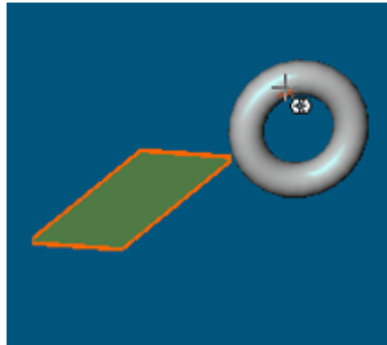
Cone to Plane

When intersecting a cone with a plane or vice versa; both need to have collinear axes. If this prerequisite is not observed; open the **Geometry Modifier Tool** and apply the **Make Perpendicular** constraint to either the cone or the plane - otherwise a warning message appears.



Plane to Circular Torus

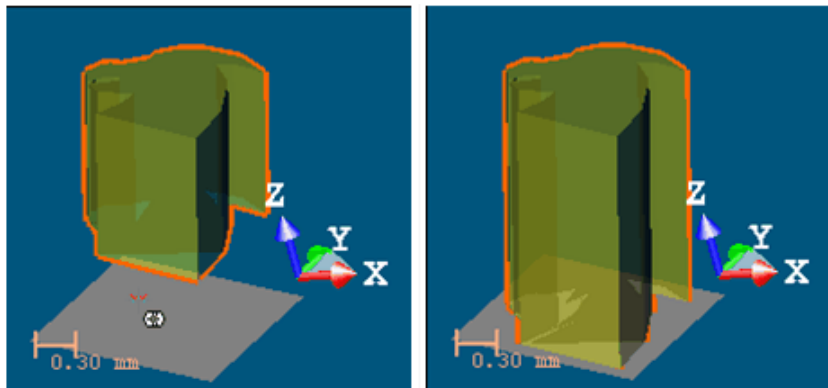
When intersecting a plane with a circular torus or vice versa; the plane needs to contain the circular torus axes. If this prerequisite is not observed; open the **Geometry Modifier Tool** and apply the constraint **Pass Through Axis** to the plane. Otherwise a warning message appears.



Plane to Extruded Entity

When intersecting a plane with an extruded entity or vice versa; both need to have collinear axes. If this prerequisite is not observed; open the **Geometry Modifier Tool** and apply the constraint below:

- **Make Parallel** to Direction to the extruded entity when intersecting it with the plane,
- **Make Perpendicular** to the plane when intersecting it with the extruded entity,

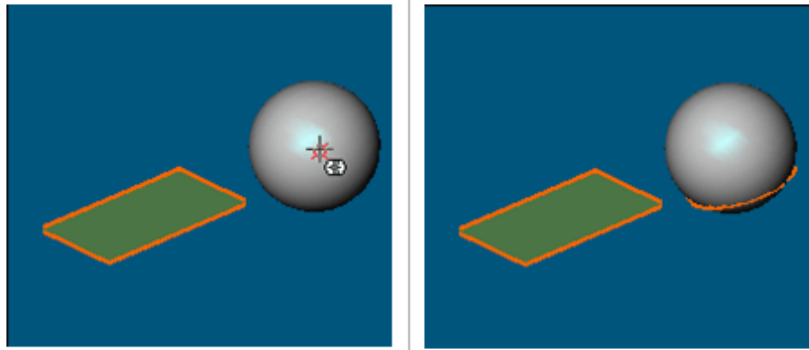


Otherwise a warning message appears.

Plane to Sphere

When intersecting a plane with a sphere or vice versa; the plane needs to be parallel to the sphere existing bound. If this prerequisite is not observed; open the **Geometry Modifier Tool** and apply the constraint below:

- **Pass Through Point** to the plane when intersecting it with the sphere,
- **Lock Center or Pass Through Point** to the sphere when intersecting it with the plane,



Otherwise a warning message appears.

Connecting to a Series of Entities

Use the "**Connect to Geometry Sequence**" feature to intersect a series of entities together. This is very useful for connecting a series of pipes together.

To Connect to a Series of Entities:

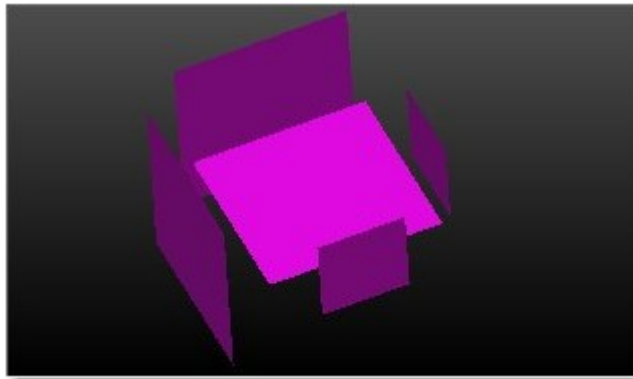
1. Display entities to intersect in the **3D View**.
2. Click the **Connect to Geometry Sequence** icon.
3. Pick an entity. The cursor becomes as shown below.
4. Pick another entity and so on.

Connecting a Plane to a Series of Planes

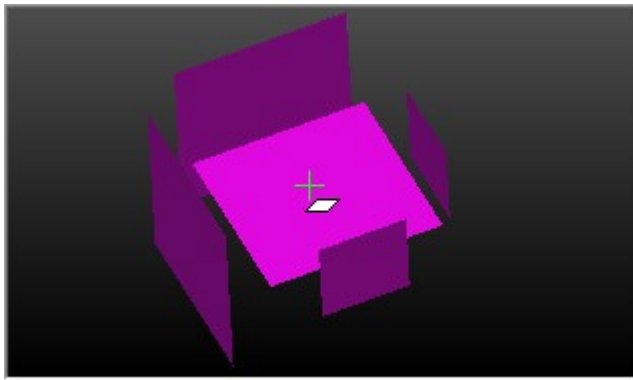
You can also use the "Connect Geometry Sequence" feature to intersect a plane with a series of planes.

To Connect a Plane to a Series of Planes:

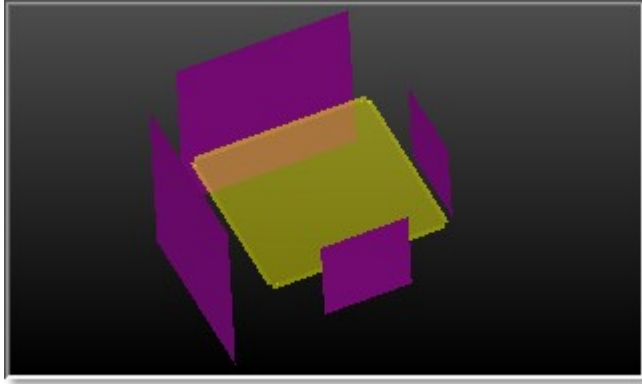
1. Display a series of planes in the **3D View**.



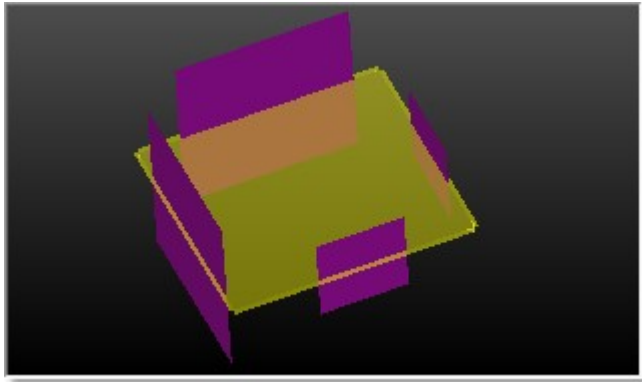
2. Click the **Connect Geometry Sequence** icon.
3. Pick anywhere in the **3D View** except on the displayed planes.
4. Press the **Ctrl** key. The cursor shape becomes as shown below.



5. Pick a plane to bound. It becomes selected.



6. Pick a series of planes which is going to be used as bounds.




Caution: Planes used as bounds need to be picked in order in any direction (clockwise or anti-clockwise).

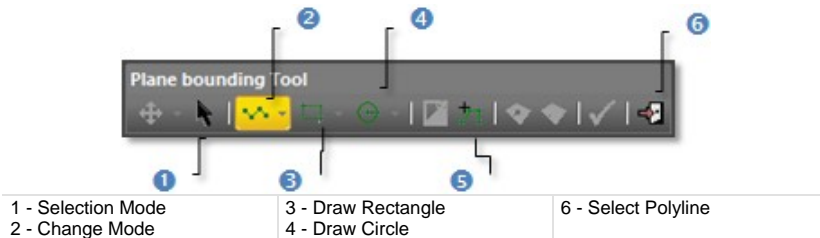
Plane Bounding Tool

The **Plane Bounding Tool** is dedicated to plane modifications. We mean by this you can modify an existing plane bounds, define new ones, create holes, etc. This tool is based on polyline drawing and editing like the **Polyline Drawing Tool**. It requires a selection as input data (mainly a plane from the database (already created)) when used alone as a main tool or is based on local objects (not yet created in the database) when used as a sub-tool in e.g. the **Cloud-Based Modeler Tool**.

Opening the Tool

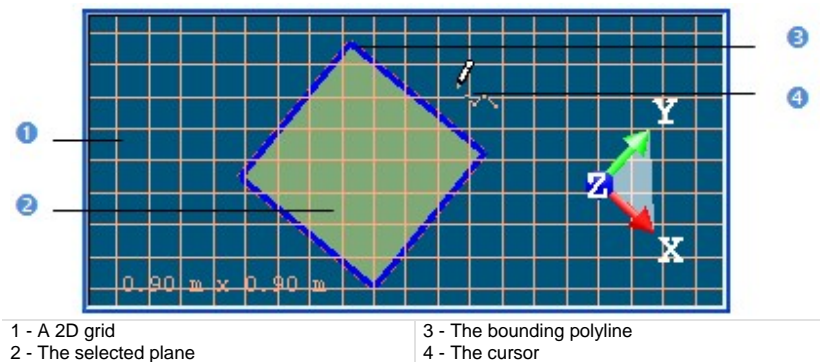
To Open the Tool:

1. Select a plane from the **Project Tree**.
2. From the **Modeling** menu, select **Plane Bounding Tool** . The **Plane Bounding Tool** and **Picking Parameters** (in the 2D constraint picking mode) toolbars appear.



In **Examiner** (or **Walkthrough**) mode, the 3D scene is locked in 2D in the selected plane and brought to the **Top** view with a 2D grid superimposed (if displayed previously). The **Selection Mode** and **Change Mode** are respectively set in the deactivate and line state. The cursor becomes as shown below and the selected plane is displayed with its bounds (a polyline of the selected plane's shape in dotted line and of pink color). The user can modify the selected plane's polyline bounds, create or select a new polyline for bounding.

In **Station-Based** mode, the 3D scene is viewed from the first station viewpoint (the first in the **Project Tree**) with overlapped images in the background (if existed).



Tip: You can also click on the **Plane Bounding Tool** icon in the **Tools** toolbar.


Editing the Selected Plane Bounds

The selected plane is displayed with its bounds: a polyline of the selected plane's shape in dotted line and of pink color. Editing the selected plane bounds is similar to editing a bounding polyline.

Selecting a Polyline for Bounding

If you have a closed polyline in your project; you can select it for bounding purposes. This polyline doesn't need to be selected but displayed in the **3D View**.

To Select a Polyline for Bounding:

1. Click **Select Polyline** . The 3D scene becomes free from the 2D lock. The 2D grid (if displayed) is hidden. The **Picking Parameters** toolbar disappears.
2. Pick a polyline in the **3D View**.
 - If this polyline is unclosed; an error message appears. Click **OK** to close this error message.
 - If this polyline is closed, an editable polyline (in green) superimposes to it. The 3D scene is locked again in 2D. The 2D grid is displayed. The **Picking Parameters** toolbar (in 2D constraint mode) appears.

Notes:

- Press **Esc** to leave the **Select Polyline** mode.
- All nodes of the polyline need to be on the selected plane. Otherwise, you cannot select it for bounding purpose.
- Selecting another polyline will cancel the current one.



Tip: **Select Polyline** can also be selected from the pop-up menu.

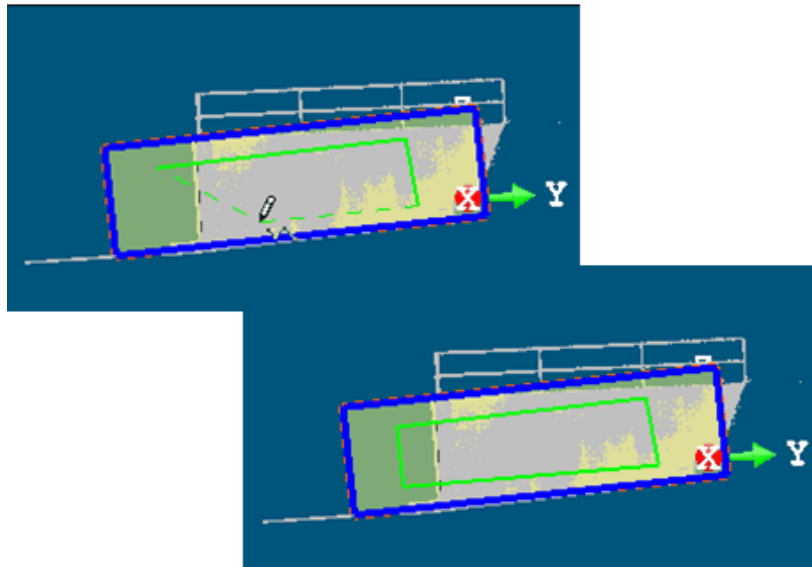
Defining a Bounding Polyline

A bounding polyline can be composed of only segments or a combination of segments and circular arcs. It needs to be of closed shape and all of its nodes have to be on the selected plane. You can only define one bounding polyline; this differs from the **Polyline Drawing Tool** where you can draw a set of polylines.

Drawing a Polygonal Bounding Polyline

To Draw a Polygonal Bounding Polyline:

1. Pick a point to start the first node of a polyline.
2. Pick another point. A segment links these two points.
3. Click on the **Change Mode** pull-down arrow.
4. Choose **Change Mode to Arc** .
5. Pick another point. The newly picked point is linked to the previous picked point by an arc.
6. Click on the **Change Mode** pull-down arrow.
7. Choose **Change Mode to Line** .
8. Pick another point. The newly picked point is linked to the previous picked point by a segment.
9. Continue picking in order to define the other nodes of the polyline.
10. Right-click anywhere in the **3D View** to display the pop-up menu.
11. Select **Close Line** to end and close the polyline. The start node is linked to the last selected node.



Notes:



- Double-click to end drawing. The polyline is closed.
- Press **Esc** while picking points will end and validate (but not create) the polyline in progress. The polyline is open.
- Picking can be free or constrained on displayed objects or not.

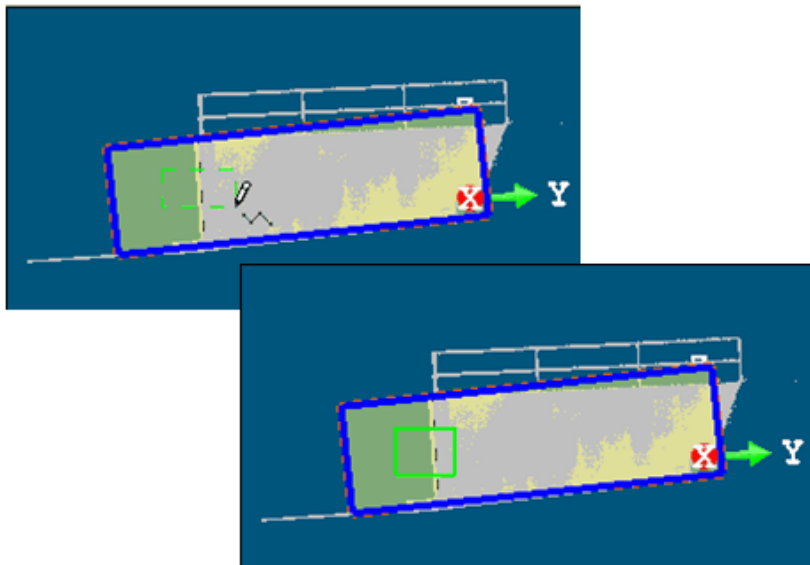
Tip:

- You can switch from the line drawing mode to the arc drawing mode and conversely as often as you wish when you draw a polyline just by pressing respectively the **L** and **C** keys on your keyboard, by clicking on the **Change Mode** button in the **Plane Bounding Tool** toolbar or by selecting **Draw Mode** from the pop-up menu.
- Out of the picking mode, press **Esc** to quit the **Plane Bounding Tool**. You can also select **Close Tool** from the pop- up menu.

Drawing a Rectangular Bounding Polyline

To Draw a Rectangular Bounding Polyline:

1. Click on the **Draw Rectangle** pull-down arrow.
 2. Choose between **Draw Rectangle by Defining 2 Points**  and **Draw Rectangle by Defining 3 Points** .
- If **Draw Rectangle by Defining 2 Points** has been chosen:
 - a) Pick a point to start the first point of a rectangle's side.
 - b) Pick another point. The first and second points will form the first side of a rectangle.
 - c) Pick another point. The second and third points will form the second side (perpendicular to the first side) of a rectangle.
 - If **Draw Rectangle by Defining 3 Points** has been chosen:
 - a) Pick a point to start the first point of a rectangle's diagonal.
 - b) Pick another point. This second point will form the diagonal of a rectangle.





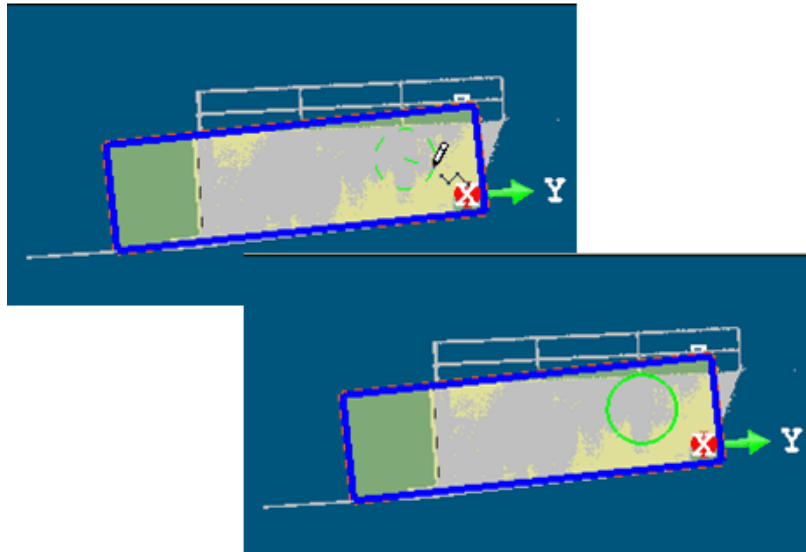
Note: Picking can be free or constrained on displayed objects or not.

Tip: You can switch from 2-point drawing mode to 3-point drawing mode and conversely as often as you wish by clicking on **Draw Rectangle** in the **Plane Bounding Tool** toolbar or by selecting the **Draw Mode** from the pop-up menu.

Drawing a Circular Bounding Polyline

To Draw a Circular Bounding Polyline:

1. Click on the **Draw Circle** pull-down arrow.
 2. Choose between **Draw Circle by Defining the Diameter**  and **Draw Circle by Defining the Middle Point and the Radius** .
- If **Draw Circle by Defining the Diameter** has been chosen:
 - a) Pick a point to start the first point of a circle's diameter.
 - b) Pick another point to set the second point of the diameter.
 - If **Draw Circle by Defining the Middle Point and the Radius** has been chosen:
 - a) Pick a point to set the center of a circle.
 - b) Pick another point. These two points will form the radius of a circle.







Note: Picking can be free or constrained on displayed objects or not.

Tip: You can switch from one mode to the other mode and conversely as often as you wish by clicking on **Draw Circle** in the **Plane Bounding** toolbar or by selecting **Draw Mode** from the pop-up menu.

Editing a Bounding Polyline

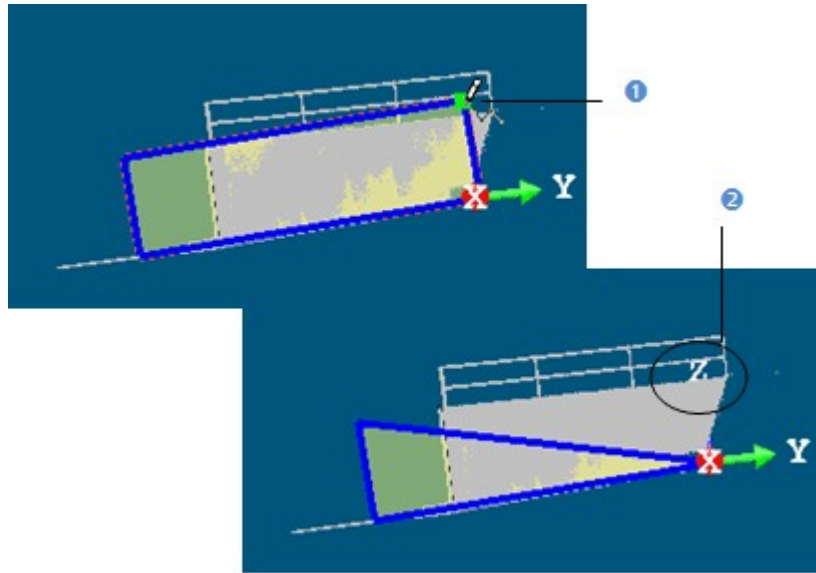
You can edit the bounding polyline of a selected plane after opening the **Plane Bounding Tool**, one selected using the **Select Polyline** command or one defined using the drawing tools (**Draw Rectangle**, **Draw Circle**, etc.).

Editing means to change the bounding polyline shape by moving, deleting, inserting a node, deleting the whole bounding polyline, etc. When you place the cursor over a segment of a polyline, you may see the following symbols:  for nodes,  for middle nodes and  for middle nodes to be inserted. When the cursor is over an arc of a polyline; only  for nodes are available.

Deleting a Node

To Delete a Node:

1. Place the cursor over a node. A solid square appears on the node.
2. Right-click to display the pop-up menu and select **Delete Node**.



1 - The selected node

2 - The node has been deleted

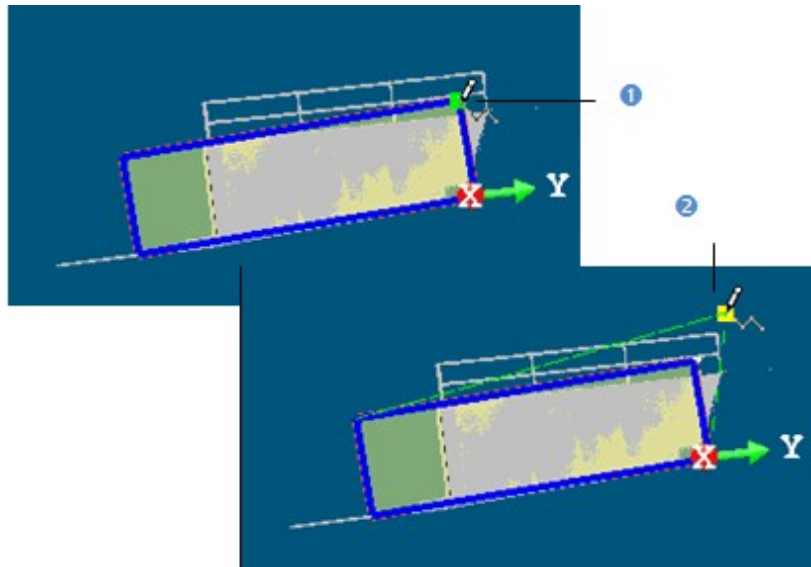
Note: Deleting a node in the middle of a segment will delete this segment. Deleting a node at the end of an arc will delete this arc.

Tip: Press **Del** on your keyboard instead of selecting **Delete Node** from the pop-up menu.

Moving a Node

To Move a Node:

1. Place the cursor over a node. A solid square appears on the node.
2. Or drag and drop the node to a new location. The green square turns to yellow during this operation.



1 - The selected node

2 - The node has been moved

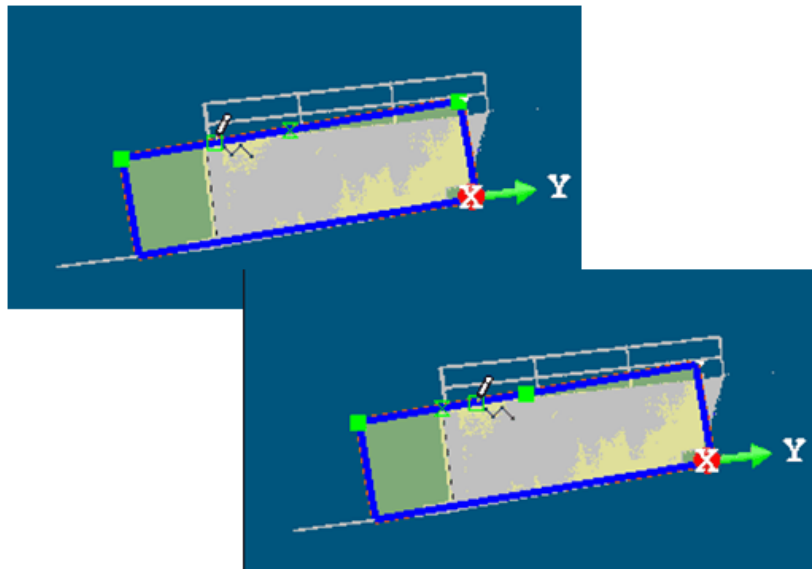
Notes:

- Moving a node at the end of a segment (or arc) will only move that node.
- Moving a node at the middle of a segment will move the whole segment.
- Picking a point anywhere on a segment except on the **End** and **Middle** nodes or on an arc except on the **End** node will transform that point to a node.

Inserting a Middle Node

To Insert a Middle Node:

1. Place the cursor anywhere on a segment (except at the end/middle nodes) or on an arc (except at the end nodes). A hollow square appears on the segment at the cursor position.
2. Right-click to display the pop-up menu and select **Insert Middle Node**. A new middle node is inserted not at the picking position but at the middle of the segment (or arc).



Deleting a Bounding Polyline

You can delete a bounding polyline when drawing is in progress or after selecting one (using **Select Polyline**). You cannot delete the bounding polyline of the selected plane.

To Delete a Bounding Polyline:

- Select **Delete**.

Tip: **Delete** can also be selected from the pop-up menu.

Activating/Deactivating the Selection Mode

You need to activate the **Selection Mode** to be able to move a bounding polyline. The **Selection Mode** is applied to the last drawn (or selected) bounding polyline if one has been drawn (or selected) or to the selected plane's bounding polyline if any has been drawn (or selected). A manipulator appears over the bounding polyline.

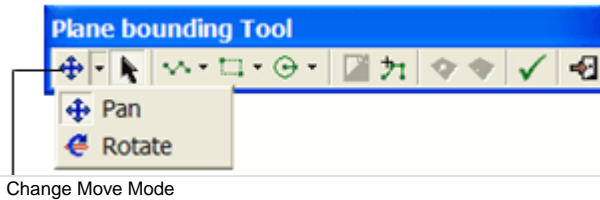
To Activate/ Deactivate the Selection Mode:

1. Click **Selection Mode**  to activate this mode.
2. Click again **Selection Mode** to deactivate this mode.

Tip: **Selection Mode** can also be selected from the pop-up menu.

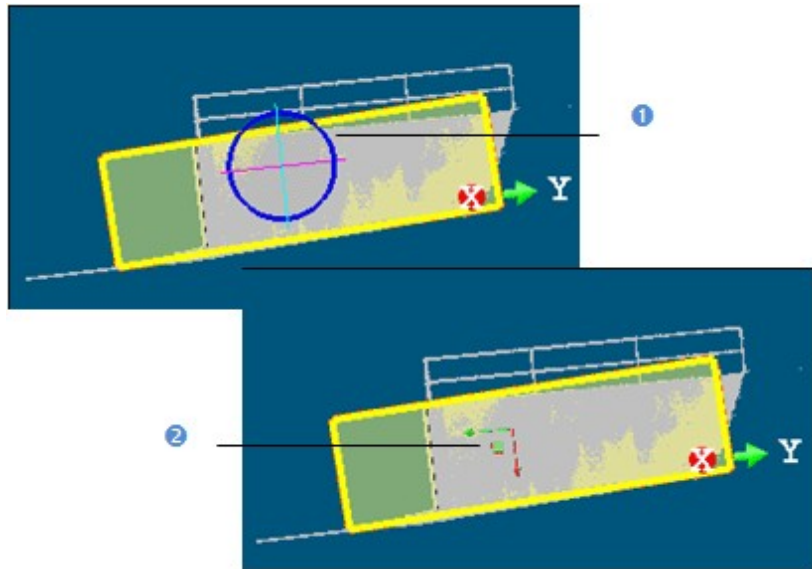
Moving a Bounding Polyline

You can use **Pan** and **Rotate** for moving a bounding polyline within the selected plane.



To Move a Bounding Polyline:

1. After activating the **Selection Mode**, **Change Move Mode** becomes active. The moving mode which comes first is the last used one. If **Pan** has been used last, a distance manipulator having as an origin the current bounding polyline's center appears. If **Rotate** has been used last, a rotation manipulator having as origin the current bounding polyline's center appears.
2. If required, click on the **Change Move Mode** pull-down arrow.
3. And swap **Pan** for **Rotate** and vice versa depending on your need.
4. Move the bounding polyline within the selected plane.



1 - A manipulator in the Rotate mode

2 - A manipulator in the Pan mode

Tip: You can also select **Pan** (or **Rotate**) from the pop-up menu. Just first select **Change Move Mode**.

Applying Bounds

In **Set as External Curve**, the bounding polyline previously is used as contouring bounds for the selected plane. Only one bounding polyline can be set as an external curve at once. In **Create Hole**, the bounding polyline is used as excavating bounds for the selected plane. This command can be applied to several bounding polylines at once.

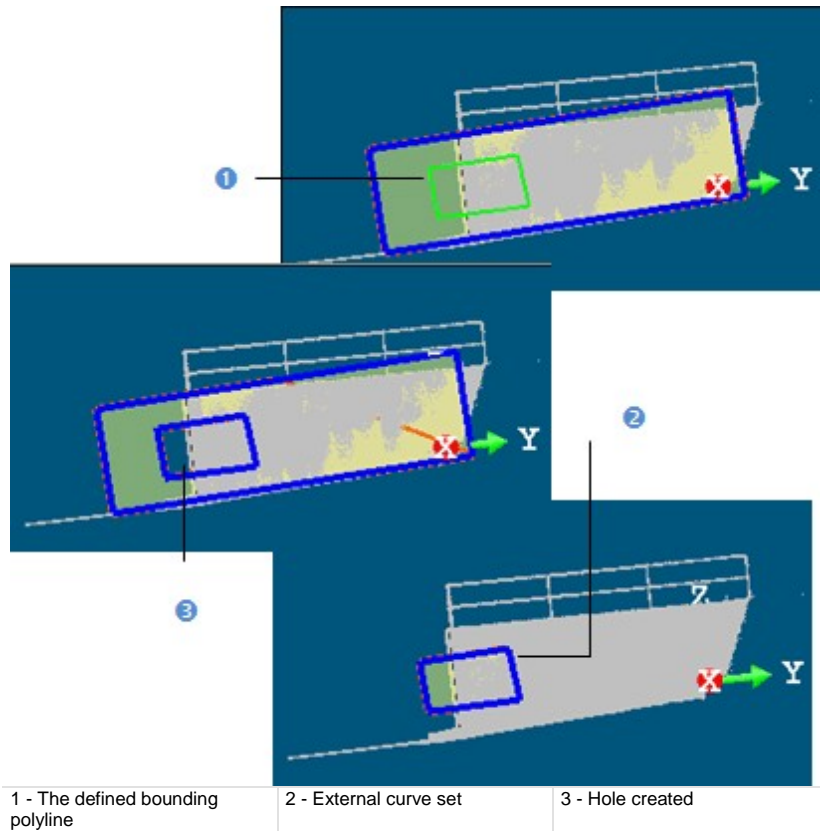


1 - Create Hole

2 - Set as External Curve

To Apply Bounds:

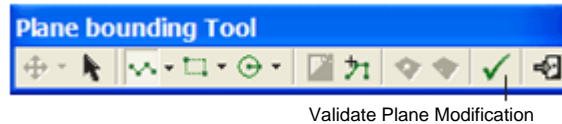
1. Define a bounding polyline as previously described.
2. Choose between **Set as External Curve** and **Create Hole**.



Tip: Set as External Curve and Create Hole can also be selected from the pop-up menu.

Validating Bounds

After applying bounds, **Set as External Curve**, **Create Hole** and **Delete All Temporary** are grayed out and **Validate Plane Modification** becomes active as well as **Change Mode**, **Draw Rectangle** and **Draw Circle**.



You can start drawing a new bounding polyline if required. Validating a modification will not create a new entity in database. You can see the number of created holes in the **Property** window (if opened).

Tips:

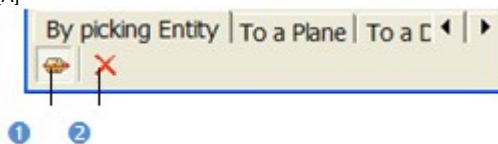
- You can also select **Validate Plane Modification** and **Close Tool** from the pop-up menu.
- For **Close Tool**, you can use its related short-cut key (**Esc**).

3D Picking Tool

The **3D Picking Tool** is a tool for picking entities with direction. It only appears when you apply constraints to object manipulation. When using the **Make Parallel** (or **Make Perpendicular**) constraint, the **3D Picking Tool** opens as a tab of the **Make Perpendicular** (or **Make Parallel**) toolbar with the **Pick Entity with Direction** mode set by-default (see [A]). When the **Make Parallel to Plane** or **Lock on Plane** constraint has been applied to object manipulation; **Pick Entity with Direction** appears in the **3D Plane Tool** opens as shown in [B]*. When the **Make Perpendicular to Direction** constraint has been applied to object manipulation; the **3D Direction Tool** opens as shown in [C]*.

The **3D Picking Tool** information at the top right corner of the **3D View** is empty of information. The **Make Perpendicular** (or **Make Parallel**) toolbar contains three tabs detailed below. When this toolbar appears; the **By Picking Entity** tab comes first. All these constraints can be found in tools like **Geometry Modifier**, **Cloud-Based Modeler**, etc.

[A]



[B]



[C]



1 - Pick Entity With Direction

2 - Cancel

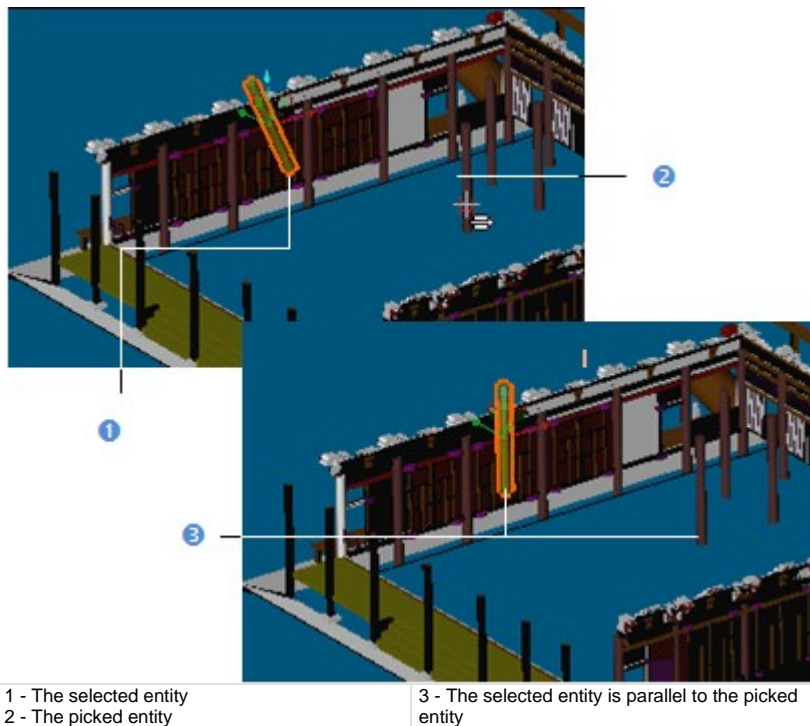
Notes:

- To leave the **Pick Entity with Direction** mode, press **Esc** or select **Cancel Picking** from the pop-up menu.
- (*) In the **Examiner** (or **Walkthrough**) navigation mode.

Picking an Entity with Direction

To Pick an Entity with Direction:

1. Select an entity from the **Project Tree** (or in the **3D View**). The **Make Perpendicular** (or **Make Parallel**) becomes active.
2. Click **Pick Entity with Direction** if required.
3. Click a point on a displayed entity.
 - If **Make Parallel** has been chosen, the selected entity will be parallel to the picked entity.
 - If **Make Perpendicular** has been chosen, the selected entity will be perpendicular to the picked entity.
4. Click **Cancel**. The **Make Perpendicular** (or **Make Parallel**) toolbar closes.



Note: When picking a plane as constraint, the entity in selection will be parallel (or perpendicular) not to the plane's normal direction but to the plane itself.

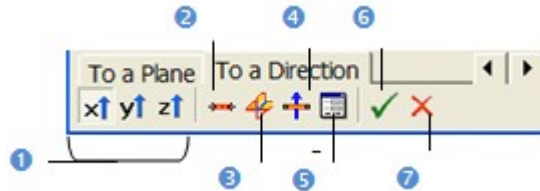
Tip: You can also select **Pick Entity with Direction** and **Cancel** from the pop-up menu. For **Cancel**, you can press **Esc**.

3D Direction Tool

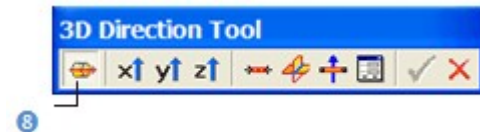
The **3D Direction Tool** allows you to define 3D primitives of linear shape (only direction). This tool appears when applying constraints to object manipulation. When using the **Make Perpendicular** or **Make Parallel** constraint, the **3D Direction Tool** opens as a tab of the **Make Perpendicular** (or **Make Parallel**) toolbar with the X-axis* mode set by-default (see [A]). The **Make Perpendicular** (or **Make Parallel**) toolbar contains three tabs detailed below. When this toolbar appears; the **By Picking Entity** tab comes first.

To use the **3D Direction Tool**, you need to click on the **To Direction** tab. A 3D direction in yellow and dotted appears. To use the tool fully, you need to have objects selected and displayed. When the **Make Perpendicular to Direction** constraint has been applied to object manipulation; the **3D Direction Tool** opens as shown in [B]. The **3D Direction Tool** information box at the top right corner of the **3D View** displays the current 3D direction (default or drawn) parameters. These constraints can be found in tools like **Geometry Modifier**, **Cloud-Based Modeler**, etc.

[A]



[B]



1 - X*, Y and Z* Axis	4 - Pick Two Screen Points	7 - Cancel
2 - Pick Two Points	5 - Edit Parameters	8 - Pick Entity With Direction
3 - Pick Two Planes	6 - Validate	







Tip: All direction definition modes can be selected from the pop-up menu or using available short-cut key (**Space Bar** for **Validate** and **Esc** for **Cancel**).

Note: (*) In the **X, Y and Z Coordinate System**.


Defining a 3D Direction Using Precise Methods

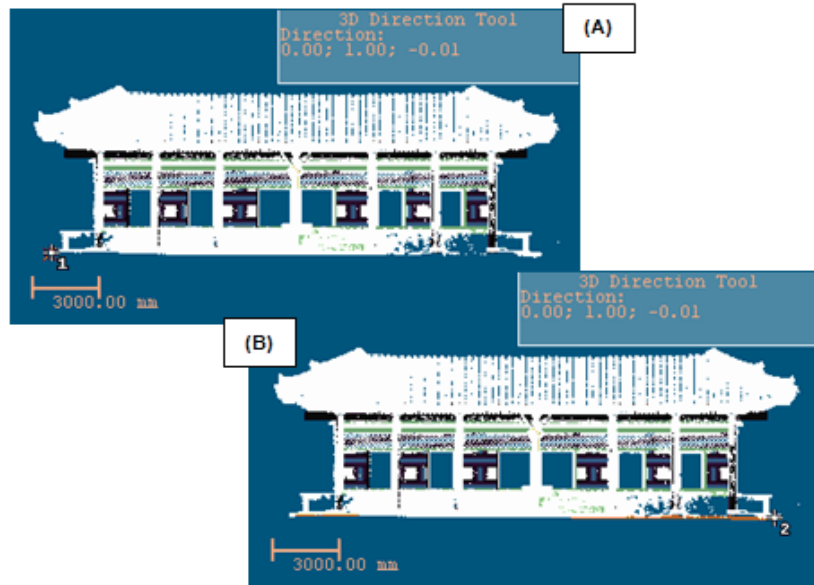
There are four methods for precisely defining the orientation of a 3D segment. The first method is to select an axis (from the active coordinate frame) so that the initial 3D direction becomes parallel to it. The second method is by picking points on displayed objects. In general, the 3D direction's orientation will be aligned to these two picked points. The third method is to edit parameters. The fourth method is to pick an entity with direction.

To Define a 3D Direction Perpendicular to an Axis:

1. Choose among **X axis** , **Y axis**  and **Z axis**  (in the **X, Y and Z Coordinate System**).
2. Or choose among **North Axis** , **East Axis**  and **Elevation Axis**  (in the **North, East and Elevation Coordinate System**).

To Define a 3D Direction by Picking Two Points:


1. Click **Pick Two Points** . The **Picking Parameters** toolbar appears in 3D constraint mode and the cursor becomes as shown in [A] and the initial 3D direction disappears from the **3D View**.
2. Pick a point (free or constrained) on the displayed entity. The cursor becomes as shown in [B].
3. Pick another point (free or constrained). A direction in the form of yellow dotted line appears.



To Define a 3D Direction by Editing Parameters:

1. Click **Edit Parameters** . The **Direction Editing** dialog opens.
2. Enter a direction in the **Direction** field.
3. Click **OK**. The **Direction Editing** dialog closes.

To Define a 3D Direction by Picking an Entity with Direction:

1. Click **Pick Entity with Direction** .
2. Pick an entity with a direction in the **3D View**.


Defining a 3D Direction Using Visual Methods

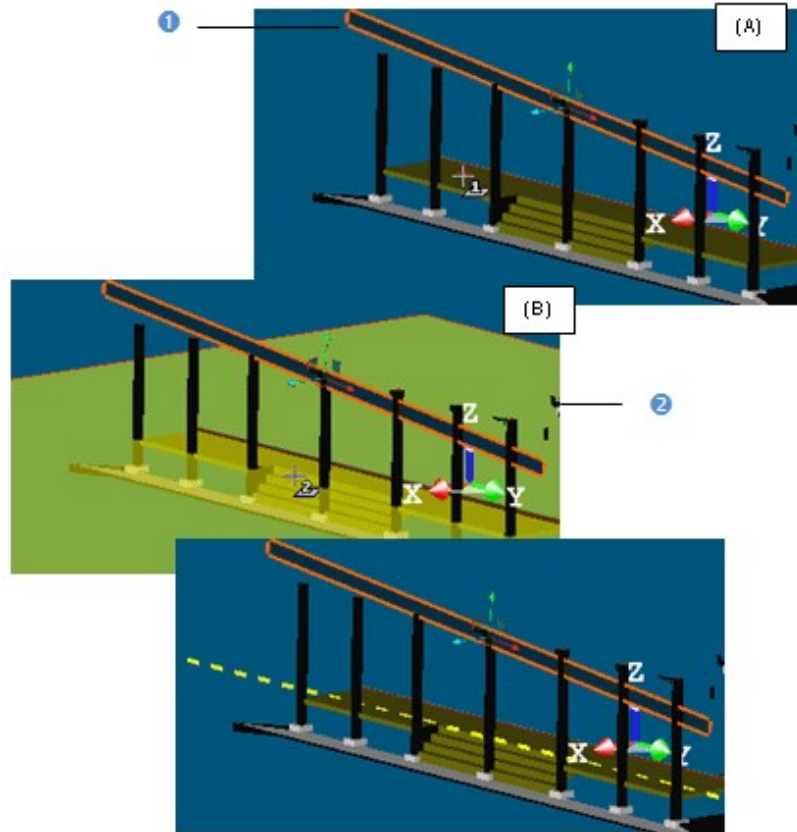
There are two methods for visually defining a 3D segment's orientation. The first method is to pick two points. A 3D segment will pass through the line defined by these two points. The second method is to pick two secant planes.

To Define a 3D Direction by Picking Two Points:

1. Click **Pick Two Screen Points** .
2. Pick two points. No need to pick on the displayed object.

To Define a 3D Direction by Picking two Planes:

1. Click **Pick Two Planes** . The cursor becomes as shown in (A).
2. Pick a plane. A red frame with a yellow background appears upon the picked plane and the cursor becomes as shown in (B).
3. Pick another plane. An axis (yellow dotted line) appears.



1 - The selected geometry

2 - The first picked plane

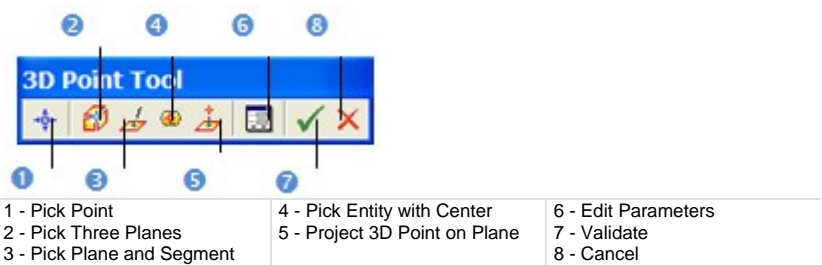
Validating a 3D Direction

Once you are satisfied with the defined 3D direction, you can validate it by clicking **Validate**. Note that any persistent object will be created in the database. Be sure to first validate the result before leaving the **3D Direction Tool**; there is no warning message prompting you to save the result or not.

Note: To leave the **3D Direction Tool**, you can click **Cancel** in the **Make Perpendicular**, **Make Parallel** or **3D Direction** toolbar, select **Cancel** from the pop-up menu or press **Esc**.

3D Point Tool



The **3D Point Tool** allows you to define a 3D primitive of **Point** shape. This tool can only be used as a sub tool inside main tools like **Geometry Modifier**, **Cloud-Based Modeler**, etc. The **3D Point Tool** opens with the **Pick Point** mode set by-default and the cursor in the picking mode. The **3D Point** information box at the top right corner of the **3D View** displays the current 3D Point's parameters - Position. Press on the **Esc** key (or select **Cancel Picking**) from the pop-up menu to leave the **Pick Point** mode. A 3D Point appears and the other modes become enabled. To use the tool fully, you need to have objects selected and displayed.



Tip: All point definition modes can be selected from the pop-up menu or using available short-cut key (**Space Bar** for **Validate** and **Esc** for **Cancel**).

Picking a Point


To Pick a Point:

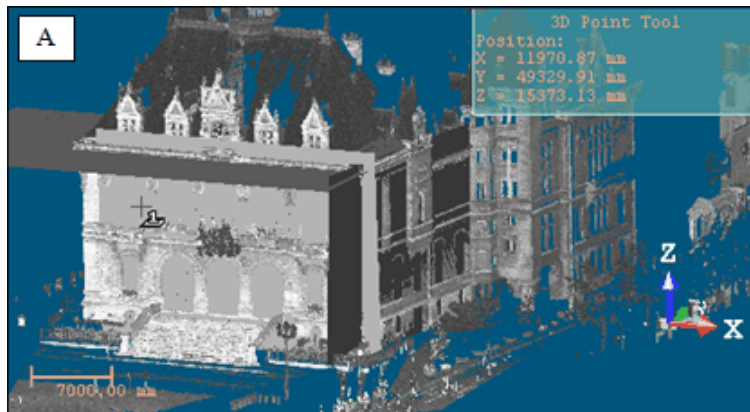
1. Click the **Pick Point**  icon. The cursor takes the following shape , the initial **3D Point** is hidden and the **Picking Parameters** toolbar opens in the 3D constraint mode.
2. Pick a point (free or constrained) on displayed objects (or not).

Picking Three Planes

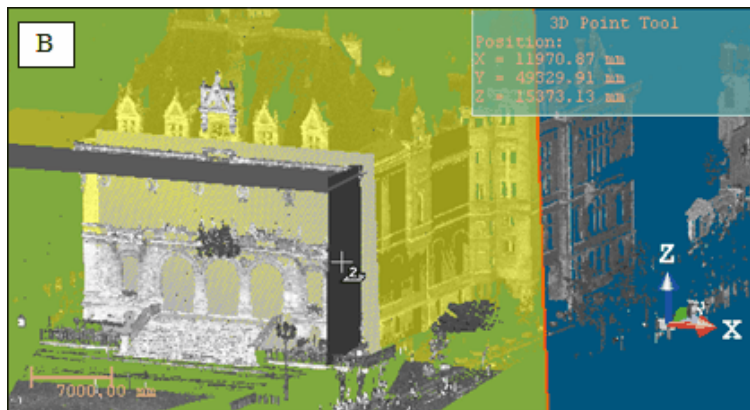
If three planes are not parallel two-by-two (with no coplanar normals), then they will intersect (cross over) somewhere at a point.

To Pick Three Planes:

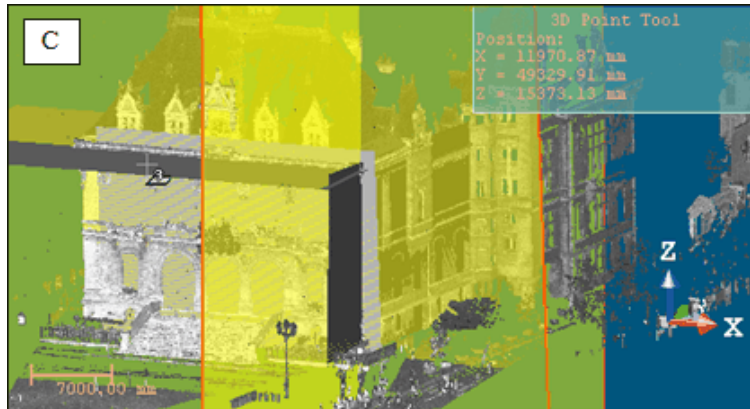
1. Click the **Pick Three Planes**  icon. The initial **3D Point** is hidden in the **3D View** and the cursor becomes as shown in [A].



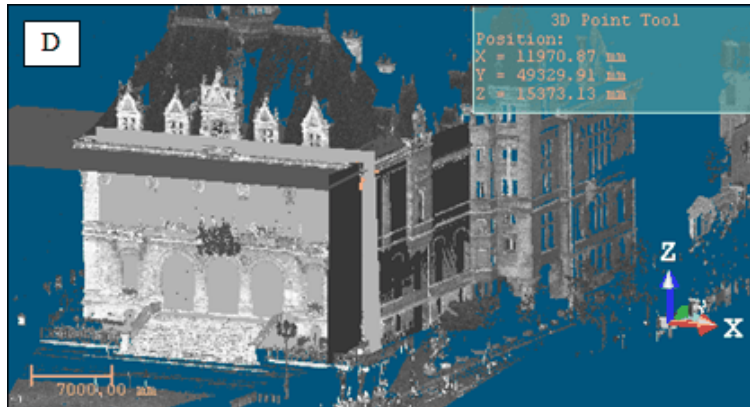
2. Pick a plane. A red frame with a yellow background upon the picked plane appears. The cursor becomes as shown in [B].



3. Pick another plane. Another red frame with a yellow background appears upon the picked plane. The cursor becomes as shown [C].



4. Pick again another plane. A 3D Point (the intersection of the three picked planes) appears. [D]

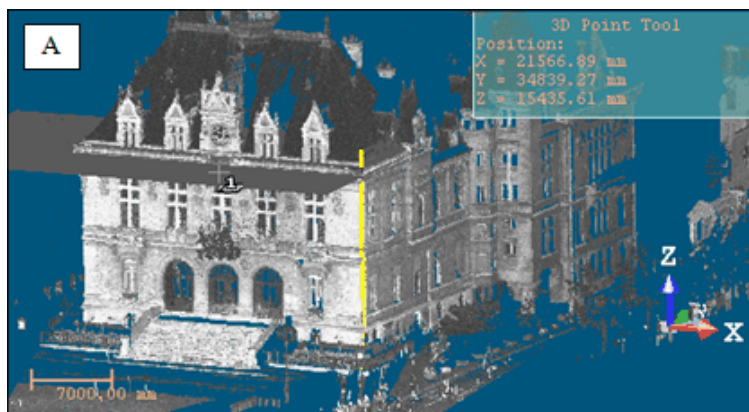


Picking a Plane and a Segment

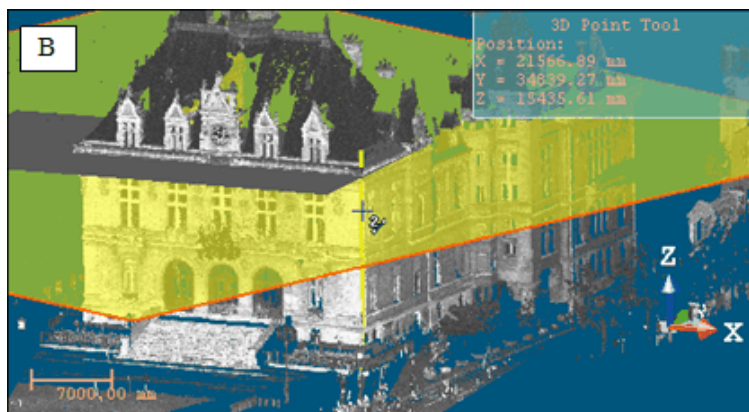
In this creation mode, you need to have a plane and a segment - both intersected at a point.

To Pick a Plane and a Segment:

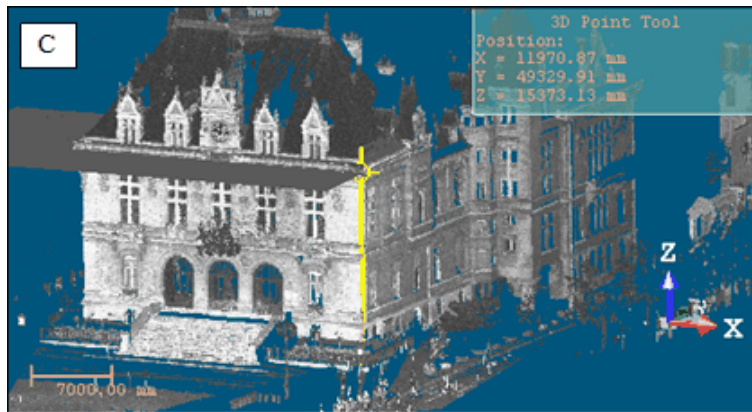
1. Click the **Pick Plane and Segment**  icon. The initial **3D Point** disappears from the **3D View** and the cursor takes the shape shown in [A].



2. Pick a plane. A red frame with a yellow background upon the picked plane appears. The cursor takes the shape shown in [B].





3. Pick a segment. The plane and the segment intersect at a 3D Point [C].




Picking an Entity with Center

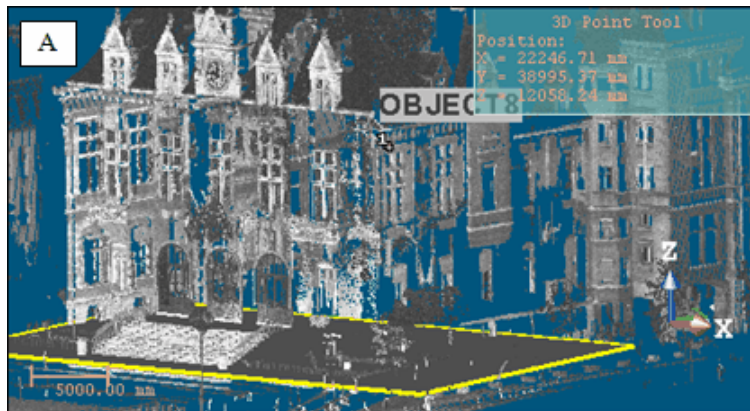
To Pick an Entity With Center:

1. Click the **Pick Entity with Center**  icon. The cursor takes the following shape . The initial **3D Point** disappears from the **3D View**.
2. Pick an object of any type having a center (except point cloud, mesh, etc.).

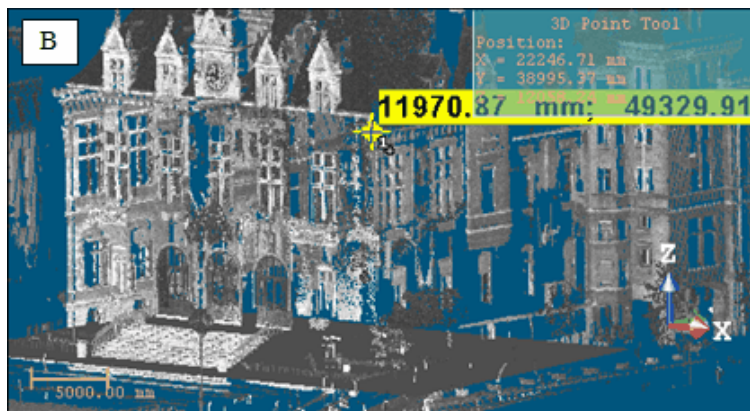
Projecting a 3D Point on a Plane

To Project a 3D Point on a Plane:

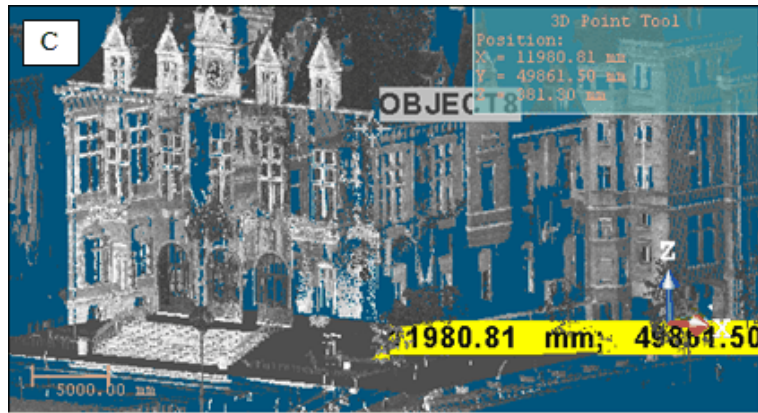
1. Click the **Project 3D Point on Plane**  icon. The initial 3D point disappears from the **3D View** and the cursor takes the shape shown in [A].



2. Pick a **3D Point**. A yellow label with the following text "**Point to Project on Plane**" appears over the picked point [B].




3. Pick a plane. A 3D point appears on that plane [C].



Editing Parameters

To Edit Parameters:

1. Click **Edit Parameters** . The initial 3D point is hidden and the **Position Editing** dialog opens.
2. Enter a point's position in that dialog.
3. Press **OK**.

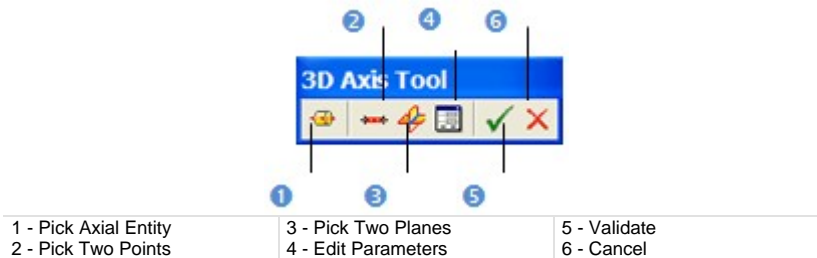
Validating a 3D Point

Once you are satisfied with the defined 3D Point, you can validate it by clicking **Validate**. No persistent object will be created in the database. Be sure to first validate the result before leaving tool because there is no warning message prompting you to validate the result (or not).

3D Axis Tool

The **3D Axis Tool** allows you to define primitives of **Segment** shape (with **Direction** and **Position**). This tool appears when you apply constraints such as **Lock Center on Line** or **Fit to Axis** in object manipulation. These constraints can be found in tools like e.g. **Geometry Modifier** and **Cloud-Based Modeler**. This tool also appears when evoking the **3D Axis Tool** inside the **Duplicator Tool**.

In the first case, the **3D Axis Tool** opens with the **Pick Axial Entity** mode set by default - the other modes are dimmed - and the cursor in the picking mode. In the second case, the **Pick Axial Entity** mode is not set by default. The **3D Axis** information box at the top right corner of the **3D View** displays the current 3D axis parameters - **Direction** and **Position**. If the **Pick Axial Entity** has been set by default; press on the **Esc** key (or select **Cancel Picking**) from the pop-up menu to leave this mode. A 3D axis (red dotted line) appears and the other modes become available. To use the tool fully, you need to have objects selected and displayed.



Tip: All axis definition modes can be selected from the pop-up menu or by using available short-cut keys (**Space Bar** for **Validate** and **Esc** for **Cancel**).


Picking an Axial Entity



To Pick an Axial Entity:

1. Click **Pick Axial Entity** . The cursor becomes as follows and the initial 3D axis disappears from the **3D View**.
2. Pick an object with a direction.

Picking Two Points


To Pick Two Points:

1. Click **Pick Two Points** . The initial 3D axis is hidden and the cursor takes the shape shown in (A).
2. Pick a point. After picking, it takes the shape shown in (B).
3. Pick another point.

[A]  [B] 

Note: Picking can be free or constrained and the **Picking Parameters** toolbar opens in 3D constraint mode. In this creation mode, you can have objects of any type.

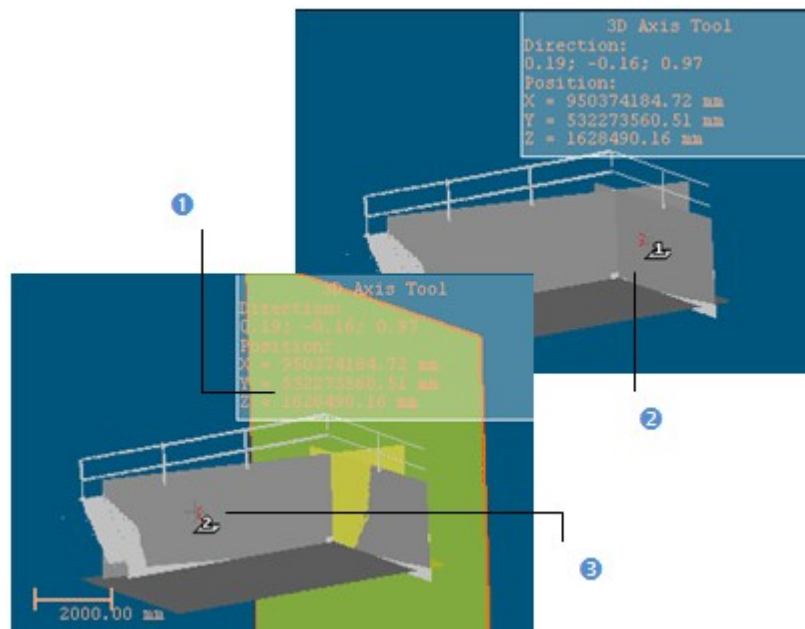
Picking Two Planes

After clicking **Pick Two Planes** , the initial 3D axis is hidden and the cursor takes the shape shown in [A]. This means that you are in the picking mode. After picking a plane, it takes the shape shown in [B]; this means that you need to pick a second plane. In this creation mode, you need to have secant planes.



To Pick Two Planes:

1. Pick a plane. A red frame with a yellow background upon the picked plane means that it is selected.
2. Pick another plane. An axis (the intersection of the two picked planes) in red and dotted appears.




1 - The first picked plane
2 - The cursor before picking the first plane

3 - The cursor before picking the second plane

Editing Parameters

To Edit Parameters:

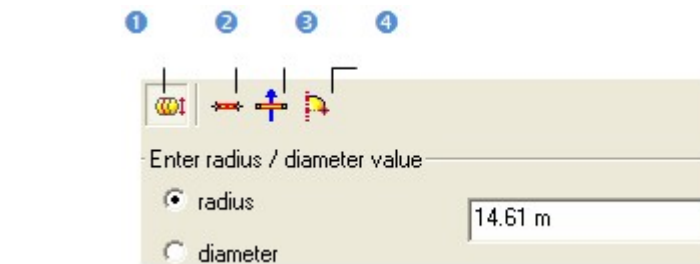
1. Click **Edit Parameters** . The **Axis Editing** dialog opens and the initial 3D axis disappears from the **3D View**.
2. Click on the pull-down arrow in the **Axis Editing** dialog.
3. Do one of the following:
 - Choose **Axis : Direction + Position**.
 - a) Give an orientation in the **Direction** field.
 - b) Enter a point position in the **Position** field.
 - Choose **Axis : 2 Points**.
 - a) Enter a point position in the **Position 1** field.
 - b) Enter another point position in the **Position 2** field.
4. Click **OK**. The **Axis Editing** dialog closes.

Validating an 3D Axis

Once you are satisfied with the defined 3D axis, you can validate it by clicking **Validate**. No persistent object will be created in the database. Be sure to first validate the result before leaving the **3D Axis Tool** because there is no warning message prompting you to validate the result (or not).

3D Radius Tool

The **3D Radius Tool** only appears when you apply constraints like **Lock Radius** and **Lock Center Line Radius** or **Lock Pipe Radius** respectively to spheres or cylinders and to regular torus. These constraints can be found in tools like e.g. **Geometry Modifier**. The **3D Radius** dialog opens with the **Pick Radial Entity** mode set by-default. The **3D Radius Tool** information box appears at the top right corner of the **3D View**. This information box displays the current entity radius value. To leave the **Pick Radial Entity** mode, press **Esc** (or select **Cancel Picking**) from the pop-up menu.



1 - Pick Radial Entity

2 - Pick Two Points on Diameter

3 - Pick Two Points on Radius

4 - Pick Axis and Point

Tip: All commands in the **3D Radius Tool** dialog can be selected from the pop-up menu. The user should first leave the picking mode.


Editing Parameters

To Edit Parameters:

1. Choose between **Radius** and **Diameter**.
2. Enter a distance value in the corresponding field.
3. Click **OK**. The **3D Radius Tool** dialog closes.


Picking a Radial Entity

To Pick a Radial Entity:

1. Click the **Pick Radial Entity**  icon.
2. Pick an entity with a regular radius.
3. If **Radius** has been checked, the entity radius is displayed.
4. If **Diameter** has been checked, the entity diameter is displayed.
5. Click **OK**. The **3D Radius Tool** dialog closes.


Picking Two Points of a Diameter

Pick Two Points of a Diameter:

1. Click the **Pick Two Points on Diameter**  icon. The **Picking Parameters** toolbar appears.
2. Pick two points free or constrained on the displayed point cloud. If **Diameter** is unchecked; it will be checked and the distance between these two picked is displayed in the corresponding field.
3. Click **OK**. The **3D Radius Tool** dialog closes.


Picking Two Points of a Radius

To Pick Two points of a Radius:

1. Click the **Pick Two Points on Radius**  icon.
2. Pick two points on displayed objects or not. If the **Radius** option is unchecked; it will be automatically checked and the distance between these two picked is displayed in the field.
3. Click **OK**. The **3D Radius Tool** dialog closes.

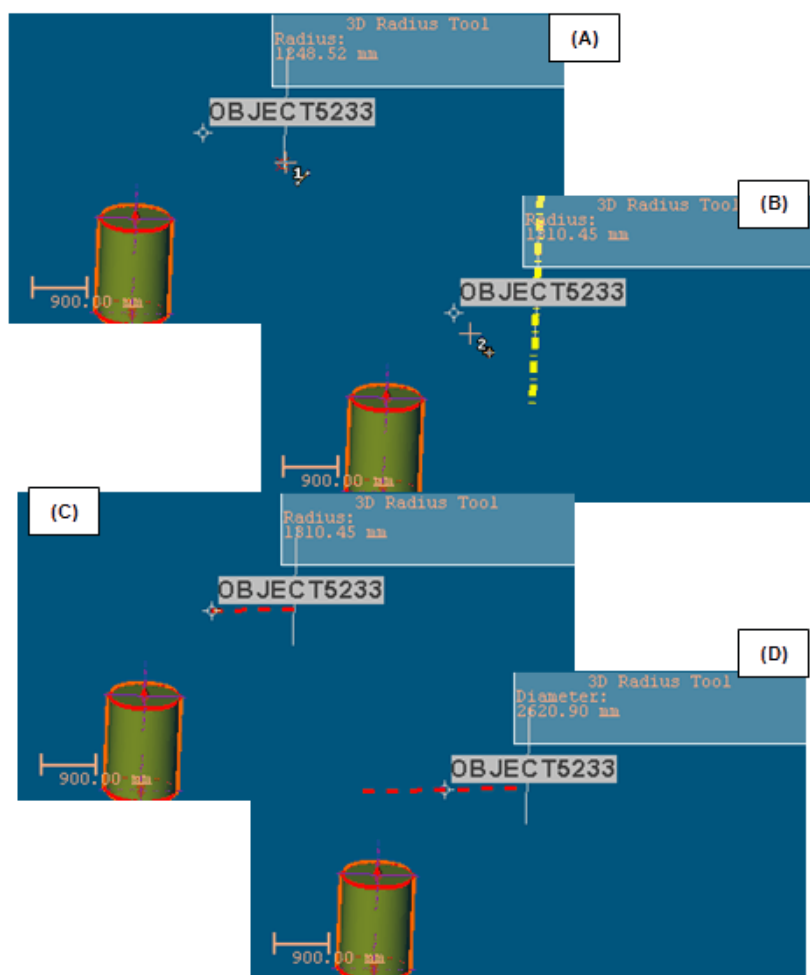
Picking an Axis and a Point

To Pick an Axis and a Point:

1. Click the **Pick Axis and Point**  icon. The cursor is as shown in [A].
2. Pick a segment. A line (in dot and yellow) appears upon the picked segment and the cursor becomes as shown in [B].
3. Pick a point.

If the **Radius** option has been checked; a line (in dot and red) from the picked point and perpendicular to the picked segment appears [C]. The distance of that line is displayed in the field next the **Radius** and **Diameter** options. This distance will be used as constrained radius.

If the **Diameter** option has been checked; the picked point will be the center of a line (in dot and red and perpendicular to the picked segment) appears [D]. The distance of that line is displayed in the field next to the **Radius** and **Diameter** options. This distance will be used as constrained diameter.



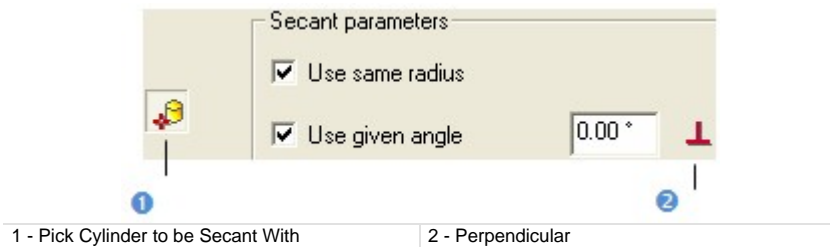
4. Click **OK**. The 3D Radius Tool dialog closes.

3D Secant Tool

The **3D Secant** dialog appears when you use the **Make Secant to Cylinder** constraint in tools like **Geometry Modifier** or **Cloud-Based Modeler**. The **Pick Cylinder to be Secant With** mode is set by default. To leave this mode, press **Esc** or select **Cancel Picking** from the pop-up menu.

The **3D Secant** information box at the top right corner of the **3D View** contains information related to the secant cylinder and the secant angle if **Use Same Radius** and **Use Given Angle** are unchecked, the secant cylinder, the radius value and the secant angle if **Use Same Radius** is checked and the secant cylinder and the secant value if **Use Given Angle** is checked.


All are undefined before applying the constraint except the secant angle which has the value in the **Use Given Angle** field or is equal to 90 degrees if **Perpendicular** is pressed-on.



Note: When you use the **Make Secant to Cylinder** constraint inside the **Geometry Modifier Tool**; you need to have a cylinder first selected. This condition is unnecessary in the **Cloud-Based Modeler Tool**.

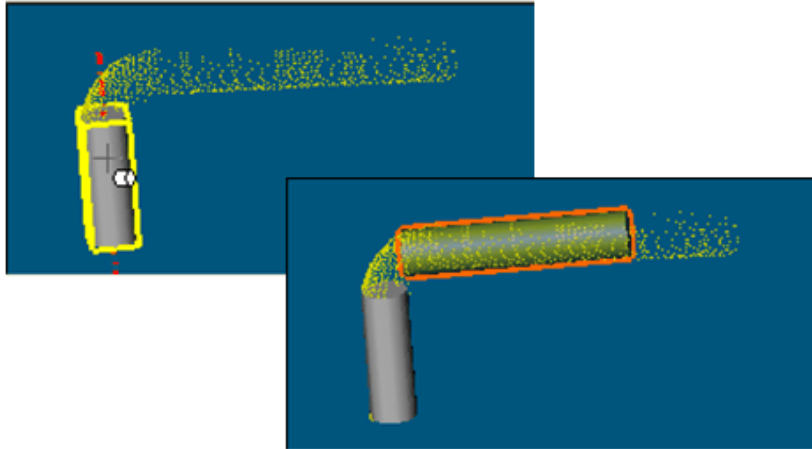
Picking a Cylinder to be Secant With

To Pick a Cylinder to be Secant With:

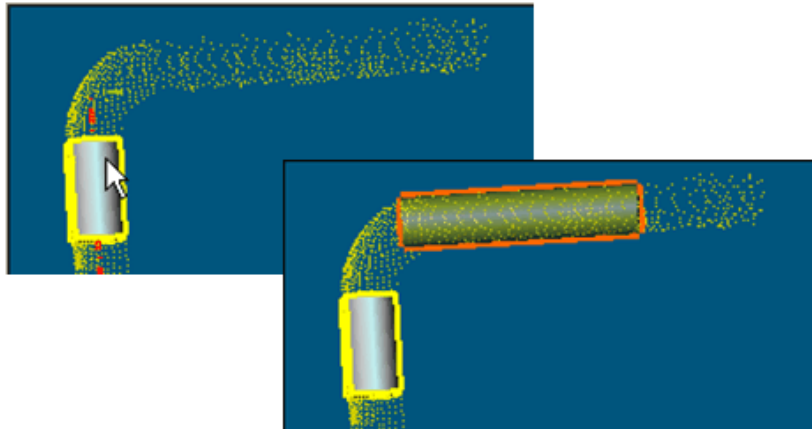
1. Click **Pick a Cylinder to be Secant With**  (if required).
2. Do one of the following:
 - Keep unchecked **Use Same Radius** and **Use Given Angle** and pick a cylinder. The selected cylinder axis is secant to the picked cylinder axis [A1].
 - Check **Use Same Radius** and pick a cylinder. The selected cylinder and the picked cylinder have secant axes and same radius [A2].
 - Check **Use Given Angle** and enter a value different from 90° or 270° . The selected cylinder axis is secant to the picked cylinder axis with the given angle [B1].
 - Check **Use Given Angle** and enter a value equal to 90° or 270° . The selected cylinder axis is secant and perpendicular to the picked cylinder axis [2].
 - Check **Use Given Angle** and press on **Perpendicular**. The selected cylinder axis is secant and perpendicular to the picked cylinder axis [B3].
 - Check **Use Same Radius** and **Use Given Angle** with an angle other than 90° or 270° . The selected cylinder axis is secant to the picked cylinder axis with the given angle and both entities have the same radius [B4].
 - Check **Use Same Radius** and **Use Given Angle** with an angle equal to 90° or 270° . The selected cylinder axis is secant and perpendicular to the picked cylinder axis both entities have the same radius [B5].
3. Click **OK**. The **Make Secant Tool** dialog closes.

Notes:

- For [A1] and [A2], you may see an axis in red and in dotted running through the cylinder used as a constraint.



- For from [B1] and [B5], you may see two axes - one in red and in dotted running through the cylinder used as a constraint and one in green and in dotted which indicates the direction of the cylinder to constrain.



3D Plane Tool

The **3D Plane Tool** allows you to define a 3D primitive of planar shape. In the **Modeling** processing mode and whatever the navigation mode you use (**Examiner**, **Walkthrough** or **Station-Based**), this tool appears when you apply constraints to object manipulation. When using the **Make Perpendicular** (or **Make Parallel**) constraint, the **3D Plane Tool** opens as a tab of the **Make Perpendicular** (or **Make Parallel**) toolbar with the **X-Axis*** mode set by-default (see [A1] and [A2] respectively in the **Examiner/Walkthrough** and **Station-Base** mode).

The **Make Perpendicular** (or **Make Parallel**) toolbar contains three tabs. When it appears; the **By Picking Entity** tab comes first. To use the **3D Plane Tool**, you need to click on the **To a Plane** tab (or click on the ◀ (or ▶) button). A 3D plane perpendicular to the screen appears and the **3D Plane Tool** information box at the top right corner of the **3D View** displays the current (default or drawn) 3D plane parameters - **Normal** and **Position**.

To use the tool fully, you need to have an object selected and displayed. When the **Make Parallel to Plane** (or **Lock on Plane**) constraint has been applied to object manipulation; the **3D Plane Tool** opens as shown in [B1] and [B2] (respectively in the **Examiner/Walkthrough** and **Station-Base** mode). All these constraints can be found in tools like e.g. **Geometry Modifier**, **Cloud-Based Modeler**, etc.

[A1]

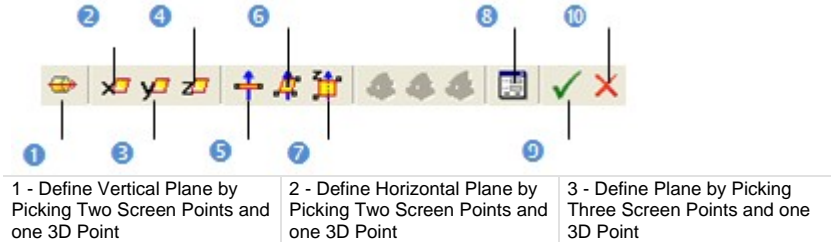


[A2]

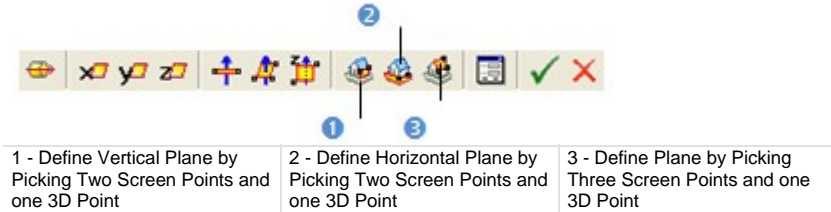


In the OfficeSurvey™ processing mode, the 3D Plane Tool opens as shown in [B1] and [B2] when navigating respectively in the Examiner/Walkthrough and Station-Based mode.

[B1]



[B2]









Note: (*) X Axis, Y Axis and Z Axis in the X, Y and Z Coordinate System (or North Axis, East Axis and Elevation Axis in the North, East and Elevation Coordinate System).


Defining a 3D Plane in all Navigation Modes

There are three methods for precisely defining the orientation of a 3D plane. The first method is to select an axis (from the active coordinate frame) so that the initial 3D plane becomes perpendicular to it. The second method is to edit parameters. The third method is to pick an entity with a direction so that the initial 3D plane normal becomes parallel to the picked entity direction.


1st Method:

1. Choose among **X Axis** , **Y Axis**  and **Z Axis**  (in the **X, Y and Z Coordinate System**).
2. Or choose among **North Axis** , **East Axis**  and **Elevation Axis**  (in the **North, East and Elevation Coordinate System**).

2nd Method:

1. Click **Edit Parameters** . The **Plane Editing** dialog opens.
2. Enter a direction in the **Normal** field.
3. Enter a point position in the **Point** field.
4. Click **OK**. The **Plane Editing** dialog closes.

3rd Method:


1. Click **Pick Entity with Direction** .
2. Pick an entity with a direction in the **3D View**.

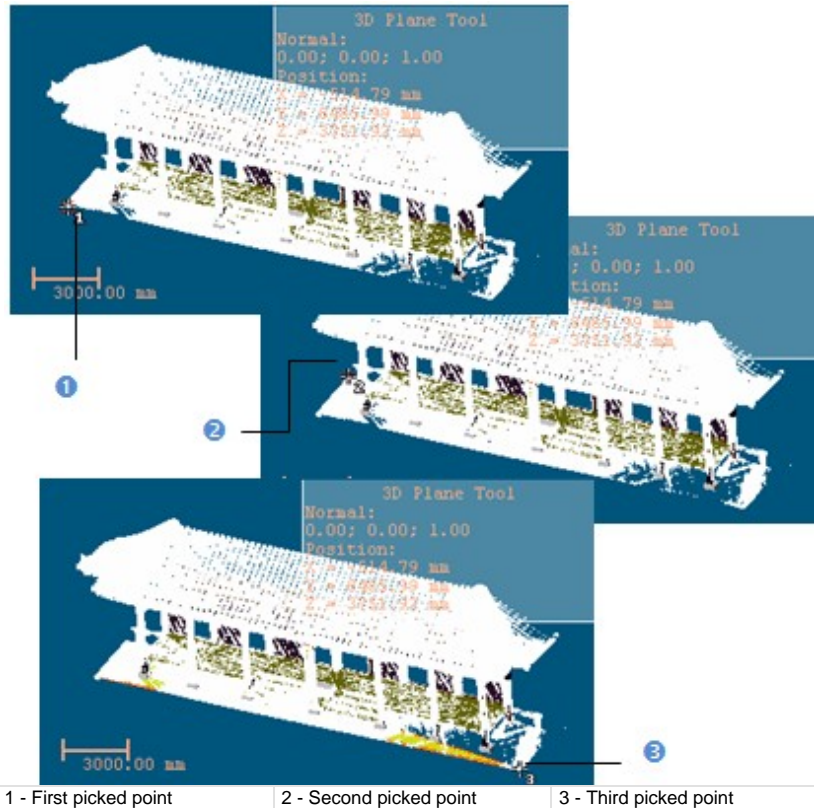
There are three methods for visually defining the orientation of a 3D plane. The first method is to pick two points. The initial 3D plane will pass through the line defined by these two points and perpendicular to the screen plane. The second method is to pick three points. The initial 3D plane will pass through the plane drawn by these three points. The third method is to pick two points which define a vector. As a plane is defined by two vectors. Applying this constraint will orientate the selected plane so that the second vectors is parallel to the **Z Axis** (or **Elevation Axis**) of the active coordinate frame.

1st Method:



1. Click **Pick Two Screen Points** .
2. Pick two points. No need to pick on displayed objects.

2nd Method:

1. Click **Pick 3 Points** . The **Picking Parameters** toolbar appears.
2. Pick three points (free or constrained). Picking is always on displayed objects.



3rd Method:


1. Click **Pick Two Points**  (in the **X, Y and Z Coordinate System**).
2. Or click  in the **North, East and Elevation Coordinate System**.
3. Pick two points. No need to pick on displayed objects.

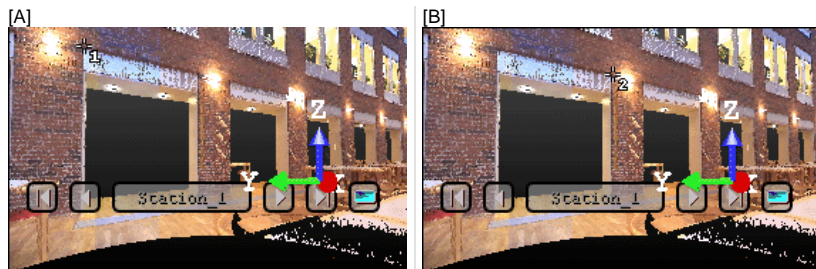
Tip: All plane definition modes can be selected from the pop-up menu or using available short-cut key (**Space Bar** for **Validate** and **Esc** for **Cancel**).

Defining a 3D Plane in the Station-Based Navigation Mode

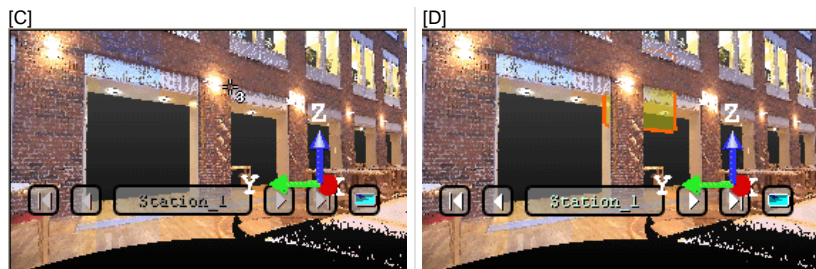
There are ten methods available in the **Station-Based** mode for defining a 3D plane. Three are specific to that mode: two are based on two screen points and a 3D point and one on three screen points and a 3D point. The other methods are already described in the **Examiner** (or **Walkthrough**) mode.

To Define a Vertical Plane by Picking two Screen Points and one 3D Point:


1. Click on the  button. The cursor becomes as shown in [A].
2. Pick a point anywhere on the screen. The cursor becomes as shown in [B].



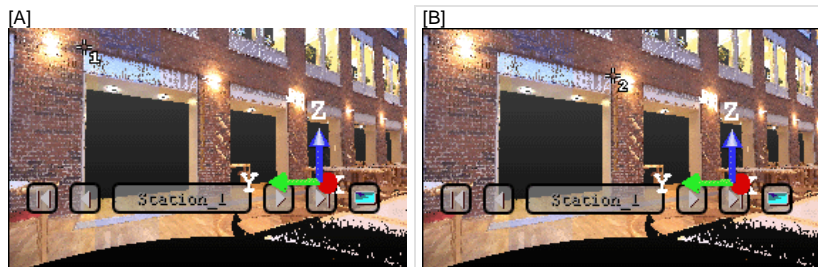
3. Pick another point anywhere in the **3D View** (on the displayed point cloud or not). The cursor becomes as shown in [C] and the **Picking Parameters** toolbar appears in 3D constraint mode.
4. Pick the last point anywhere in the **3D View** (only on the displayed point cloud). A vertical plane appears with the third picked point as center.



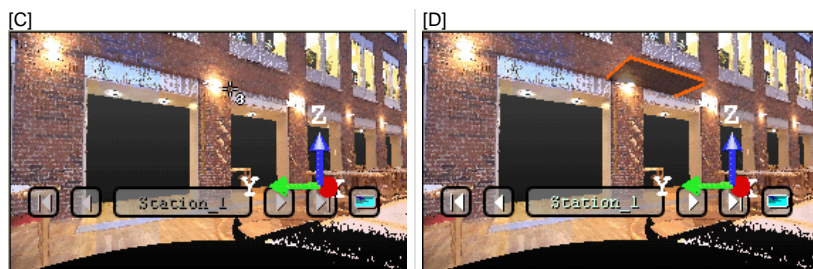
To Define a Horizontal Plane by Picking Two Screen Points and one 3D Point:

1. Click on the  button. The cursor becomes as shown in [A].


- Pick a point anywhere on the screen. The cursor becomes as shown in [B].



- Pick another point on the screen. These two points will define the orientation of the first axis of the horizontal plane's frame. The cursor becomes as shown in [C] and the **Picking Parameters** toolbar appears in 3D constraint mode.
- Pick the last point in 3D (on a cloud point, a measured point or a geometry). This point defines the height of the vertical plane. A horizontal plane appears with the third picked point as center.

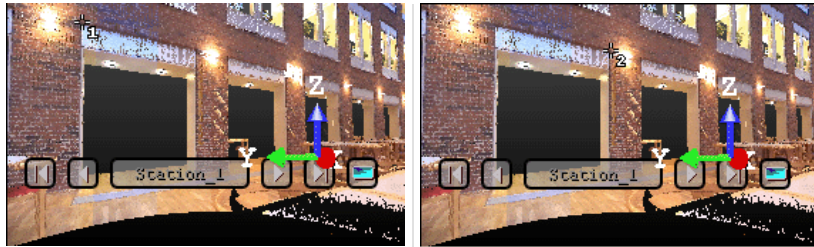


To Define a Plane by Picking Three Screen Points and one 3D Point (this Enables to Define a Plane with any Orientation):

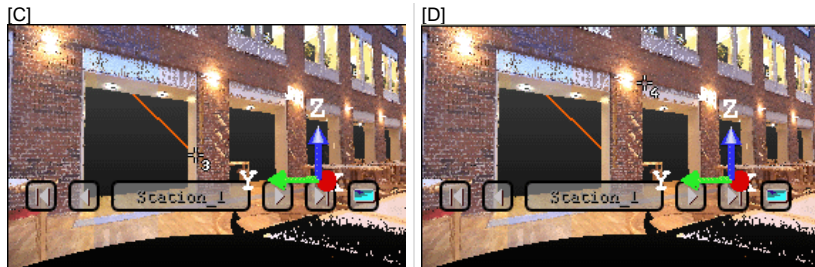
- Click on the  button. The cursor becomes as shown in [A].
- Pick a point anywhere on the screen. The cursor becomes as shown in [B].

[A]

[B]






3. Pick another point on the screen so that the two points represent a horizontal segment in the 3D space. These two points define the orientation of a horizontal segment drawn on the final plane. The cursor becomes as shown in [C].
4. Pick another on the screen so that the previous point and this new one represent the steepest slope direction of the final plane. The cursor becomes as shown in [D] and the **Picking Parameters** toolbar appears in 3D constraint mode.



5. Pick the last point in the **3D View** (only on the displayed point cloud). The three first picked points - which are not collinear (not lying on the same line) - draw a 3D plane; the fourth picked point is its center.



Tip: ,  and  can be selected from the pop-up menu.

Modifying a 3D Plane's Size

You can resize the previous 3D plane. The resized 3D plane keeps the same parameters as before except the dimensions. You can do this by dragging & dropping a corner in the **3D View**.

To Modify a 3D Plane's Size:

1. Place the mouse cursor upon any handle of a 3D plane. A green square appears.
2. If a corner handle is selected, drag it to increase (or reduce) the 3D plane size. During this operation, the green square becomes yellow.
3. If a middle handle is selected, drag it to increase (or reduce) the 3D plane width (or length). During this operation, the green square becomes yellow.

Validating a 3D Plane

Once you are satisfied with the defined 3D plane, you can validate it by clicking **Validate**. Note that any persistent object will be created in the database. Be sure to first validate the result before leaving the **3D Plane Tool**; there is no warning message prompting you to save the result or not.

Note: To leave the **3D Plane Tool**, you can click **Cancel** in the **Make Perpendicular**, **Make Parallel** or **3D Plane Tool** toolbar, select **Cancel** from the pop-up menu or press **Esc**.

Managing SteelWorks Catalogs

A **SteelWorks Catalog** provides parametric definition of all components in the required size ranges, ratings and types. You can use it to create beams with constraints.

Importing SteelWorks Catalogs

You can import a lone (or a set of) steelworks catalog(s) before starting the **SteelWorks Creation Tool** (or within that tool).

To Import a SteelWorks Catalog:

1. From the **Plant** menu, select **Import SteelWorks Catalog(s)**. The **Import Catalog** dialog opens.
2. Navigate to the drive/folder where the SteelWorks catalog file is located in the **Look In** field.
3. Click on the SteelWorks catalog file to select it. Its name appears in the **File Name** field.
4. Click **Open**. The **Import Catalog** dialog closes.

Note: Importing a catalog file that is already imported will open a warning message.

Some catalog files may have been installed in X:\Program Files\Trimble\RealWorks 8.0\Tables\SteelWorks after installing **RealWorks**. These catalog files are samples. If the user decides to not install these tables, he needs to first choose the "**Custom**" option when installing **RealWorks** and then uncheck the "**RealWorks Plant Tables**" option in the **Select Features** window.

Removing SteelWorks Catalog List

You can delete all steelworks catalogs inside (or out of) the **SteelWorks Creation Tool**. The **Remove SteelWorks Catalog List** command will remain dimmed until a catalog file is first imported.

To Remove the SteelWorks Catalog List:


- From the **Plant** menu, select **Remove SteelWorks Catalog List**.

SteelWorks Creator Tool

This feature allows you to model structural steelworks. Standard and specific catalogs can be used within it.

Starting the Tool

To Start the Tool:

1. Select one point cloud* (or more**) from the **Project Tree**.
2. From the **Plant** menu, select **SteelWorks Creator Tool** . The **SteelWorks Creator** and the **Drawing Tool** both open, as the third tab of the **WorkSpace** window for the first and as a toolbar the second. We will call the input point cloud a **Cloud Data**.

Notes:



- The **Picking Parameters** toolbar appears in 3D constraint mode.
- (*) If the selected point cloud is **On** before starting the tool, it automatically swaps to **Off**. We advise you to maintain it **Off**.

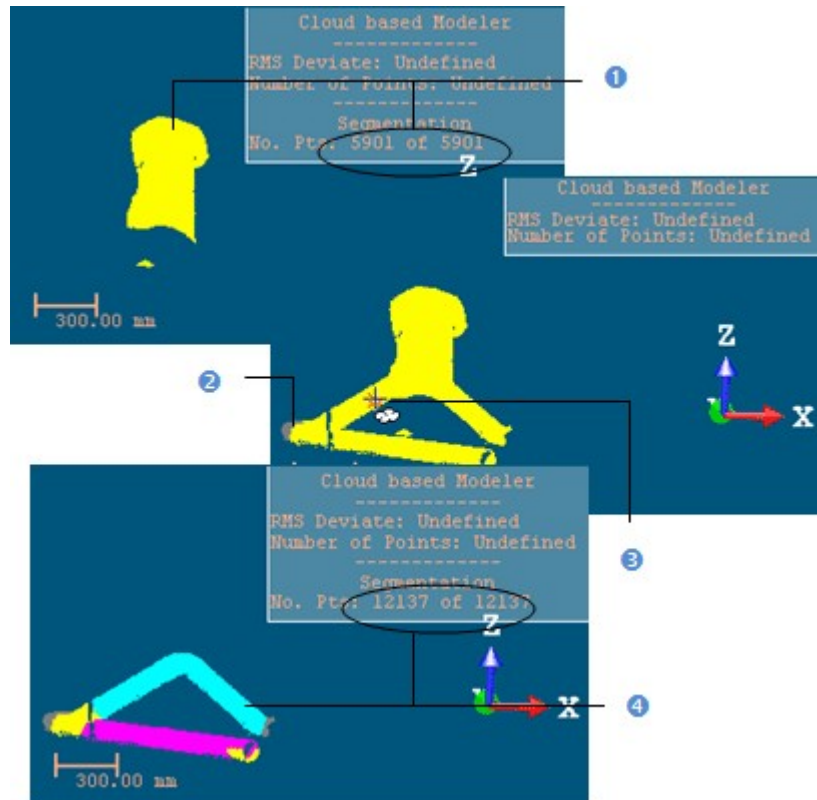
Caution: (**) You can select several point clouds as input of the tool but one of them should not be the **Project Cloud**.

Selecting a New Cloud Data

The **Set New Cloud Data** is for swapping the default **Cloud Data** (not necessary the one selected before starting the **Cloud-Based Modeler Tool**) for another one. You cannot choose and set a subset of the default **Cloud Data** as the new **Cloud Data**; you need to choose a different point cloud.

To Select a New Cloud Data:

1. Select another point cloud from the **Project Tree**.
2. Display the selected point cloud in the **3D View**.
3. If required, hide the default **Cloud Data** (by clicking  in the **Cloud-Based Modeler** dialog).
4. Click the **Set New Cloud Data**  icon. The cursor becomes as shown below and the information box related to the **Segmentation Tool** disappears from the **3D View**.
5. Pick a point on the selected point cloud. It becomes the new **Cloud Data**. The **Segmentation** information box appears again with the new cloud data number of points.



1 - The initial Cloud Data
2 - The newly selected Point Cloud

3 - The cursor in the Set New Cloud Data mode
4 - The new Cloud Data

Note: (*)The **Hide Cloud** icon becomes **Display Cloud** after clicking on it.

Defining a Set of Points on the Cloud Data

Frequently, the **Cloud Data** contains many points; you need to decimate them before doing the fitting. You may also decide to fit a geometry just on a part of it. To do these, you can use the **Segmentation** and the **Sampling** sub-tools.



1 - The Segmentation Tool

2 - The Sampling Tool

After segmenting/sampling the **Cloud Data**, **Keep Only Displayed Cloud in Cloud Data** and **Delete Displayed Cloud from Cloud Data** (respectively for keeping/deleting points in/from the current **Cloud Data** (after decimating/fencing) and **Reload Points** become active.






1 - Reload Points

2 - Keep Only Displayed Cloud
in Cloud Data

3 - Delete Displayed cloud from
Cloud Data

Keeping Only Displayed Cloud

To Keep Only Displayed Data:

1. Draw a fence on the **Cloud Data**.
2. Use the **In**  (or **Out** )* icon to keep points inside (or outside) the fence.
3. Or sample the **Cloud Data**.
4. Click the **Keep Only Displayed Cloud in Cloud Data**  icon. Points displayed in the **3D View** inside are kept.




Notes:

- **Reload Points** is only for reloading points of the current **Cloud Data** after sampling or segmenting.
- (*) You can skip the step of keeping **In** (or **Out**). In that case, points inside the fence are kept.

Tip: Outside the segmentation mode, you can select the **Segmentation Tool** from the pop-up menu or use its related short-cut key (**S**).

Deleting Displayed Cloud

To Delete the Displayed Cloud:






1. Draw a fence on the **Cloud Data**.
2. Use the **In**  (or **Out** )* icon to keep points inside (or outside) the fence.
3. Or sample the **Cloud Data**.
4. Click the **Delete Displayed Cloud from Cloud Data**  icon. Points displayed in the **3D View** are unkept (removed from the **Cloud Data**).

Notes:

- **Reload Points** is only for reloading points of the current **Cloud Data** after sampling or segmenting.
- (*) You can skip the step of keeping **In** (or **Out**). In that case, points inside the fence are unkept.

Tip: Outside the segmentation mode, you can select the **Segmentation Tool** from the pop-up menu or use its related short-cut key (**S**).

Choosing a Section Type

There are five types of Sections you can create:  H Section,  I Section,  L Section,  T Section and  U Section.

To Choose a Section Type:

- In **Step 1**, choose a **Section Type** by clicking on its related icon.

Selecting a Catalog File

If there is a catalog file in your project; it is by default selected and its name* appears in the **Catalog Files** field. Note that a catalog file has **SPEC** as extension. "None" in the **Catalog Files** field means one of two things. First, there is no catalog file has been imported into your project. Second, it is because you don't want to create a beam based on a catalog.

To Select a Catalog File:

1. In **Step 2**, click on the **Catalog Files** pull-down arrow.
2. Choose a catalog file from the drop-down list.

Tips:

- You can import a catalog file within the **SteelWorks Creator Tool**. You may not see it once imported. In that case, select it by dropping down the **Catalog Files** list.
- You can delete the whole catalog files within the **SteelWorks Creator Tool**.

Note: (*) The name you see in the **Catalog Files** field is not necessarily the name of the imported catalog file. This name is the one you find in #SPEC NAME# in the catalog file.

Selecting a Table

The available tables* are: HEA, HEB (for **H Section**), IPE, INP (for **I Section**), UNP (for **U Section**), LEA (for **L Section**), TEA (for **T Section**). If there is no catalog file that has been imported into your project; you may see "No Table" in the **Table** field.

To Select a Table:

1. In **Step 2**, click on the **Table** pull-down arrow.
2. Choose a table from the drop-down list.

Note: (*) The number of **Tables** as well as their name or the kind of **Tables** may differ from one catalog to another. The **Tables** given above are based on the standard **DIN** catalog. In a catalog like the **AFNOR** catalog, new tables can be added (UAP, UPF, HEM tables only).

Selecting a Reference

If there is no catalog file that has been imported into **RealWorks**; you may see "None" in the **Selected Reference** field.

To Select a Reference:

1. In **Step 3**, first check the **Catalog Constraint** option.
2. Then click on the **Selected Reference** pull-down arrow.
3. Choose a **Reference** from the drop-down list.

Tip: You can select a **Reference** from the drop-down list without checking the **Catalogue Constraint** option. It will be automatically checked.




2D Sections

The **Change Mode**, **Draw Rectangle** and **Draw Circle** icons are not present in the **Drawing Tool** toolbar. Drawings are constrained to the shape chosen in **Step 1**.

Note: The **Walkthrough** navigation mode is forbidden. If you are in the **Walkthrough** mode, the navigation mode will swap of its own from that mode to **Examiner** after starting drawing.

Define a 3D Plane

To Define a 3D Plane:

1. Click the **Start 3D Plane Tool**  icon. The **3D Plane Tool** toolbar and a 3D plane both appear.
2. Define and validate a 3D plane*. The scene is locked on the defined 3D plane (which is hidden) with a 2D grid superimposed (if not hidden previously). The **Picking Parameters** toolbar appears in the 2D constraint mode. The **Show/Hide Plane**  icon becomes active and **Lock in 2D**  is default-set.
3. Click the **Show/Hide Plane** icon to display the defined 3D plane (if required).

Note: (*) Please, refer to the **3D Plane Tool** on how to define a 3D plane.

Tips:

- You can also select **Start 3D Plane Tool** (or **Show/Hide Plane**) from the pop-up menu.
- The **Find Best Extrusion View**  is only present when using the **3D Plane Tool** in the **SteelWorks Creator Tool**. It enables to find the best cutting direction.


Using the Current View as 3D Plane


To Use the Current View as 3D Plane:

1. Navigate through the 3D scene to find the best view on which you want to draw a 2D section.
2. Pick a first point. The scene is locked on the current view plane with a 2D grid superimposed (if not hidden previously) and the **Picking Parameters** toolbar in the 2D constraint mode.

Drawing a 2D Section

To Draw a 2D Section:


1. Pick a first point to select the anchor point of the 2D section.
2. Move the mouse to change the size of the 2D section. The **Selected Reference** field is automatically updated. If the **Catalog Constraint** option has been checked; the 2D section size is constrained to the values stored in the selected table. If the option hasn't been checked, there is no constraint to the 2D section size drawing.
3. Click the second point to complete the 2D section.
4. Validate the 2D section by clicking .
5. If required, click **New SteelWorks** to cancel the current 2D section.

Tip: The **New SteelWorks**  icon can also be selected from the pop-up menu.

Changing the 2D Section Position and Orientation

You can change the 2D section position and orientation so that it fits exactly the working cloud.



To Change the 2D Section Position and Position:

1. Click on the **Selection Mode**  icon. The **Change Move Mode** button becomes enabled with **Pan** as default mode.
2. A manipulator (with two secant handles and a plane) appears.
3. Click on a handle; it turns to yellow. The direction along which you can displace the 2D section is highlighted in yellow and the one along which you cannot displace it is in mauve.
4. Move the 2D section along that direction.
5. Click on the plane. It turns to yellow.
6. Move the 2D Section in any direction in that plane.
7. Click on the **Change Move Mode** pull-down arrow.
8. Choose **Rotate** from the drop-down list. The **Manipulator** changes of shape.
9. Click on the **Manipulator**. It turns to yellow.
10. Turn left or right the **Manipulator**.

Defining a Length

You need to define a **Length** only if you don't want to create a beam based on a catalog. Otherwise, defining a **Length** has no effect.

To Define a Length:

1. Enter a distance value in the **Length** field.
2. Or use the **Up**  (or **Down** ) button.

Tip: The extrusion's **Length** can also be modified by using the **Geometry Modifier Tool**.

Creating Beams

An extruded entity will be created in the **RealWorks** database. Each extruded entity has as name "**ObjectX - Reference - Catalog**" where **X** is its order. **X** always starts at **One**. **Reference** will be replaced by the **Selected Reference**'s value and **Catalog** by the **Catalog**'s name. If the **Catalog Constraint** option has not been checked; the extruded entity has as name "**ObjectX - SteelWorks**".

To Create Beams:

1. Click **Create** in the **SteelWorks Creator** dialog.
2. Or click **Create** in the **Drawing Tool** toolbar.
3. Or press **Enter**.

Tip: **Create** can also be selected from the pop-up menu.


EasyPipe Tool

The **EasyPipe™ Tool** is very easy to use because you only need a few clicks to execute the following tasks: extract a pipe path from more than one million points and model the extraction with geometric entitie(s) if needed. The procedure given below guides you step-by-step through the use of this tool. For each command, you can use its short-cut key (if available); this allows you to accelerate your work.

Opening the Tool

Only an object having the point cloud representation can be selected as entry for the **EasyPipe** tool.

To Open the Tool:

1. Select a point Cloud* (or more**) from the **Project Tree**.
2. From the **Plant** menu, select **EasyPipe Tool** . The **EasyPipe** dialog opens as the third tab of the **WorkSpace** window. We will call the input **Point Cloud** a **Cloud Data**.



Note: (*) If the selected **Point Cloud** is **On** before starting the tool, it automatically tilts to **Off**. We advise you to maintain it **Off**.

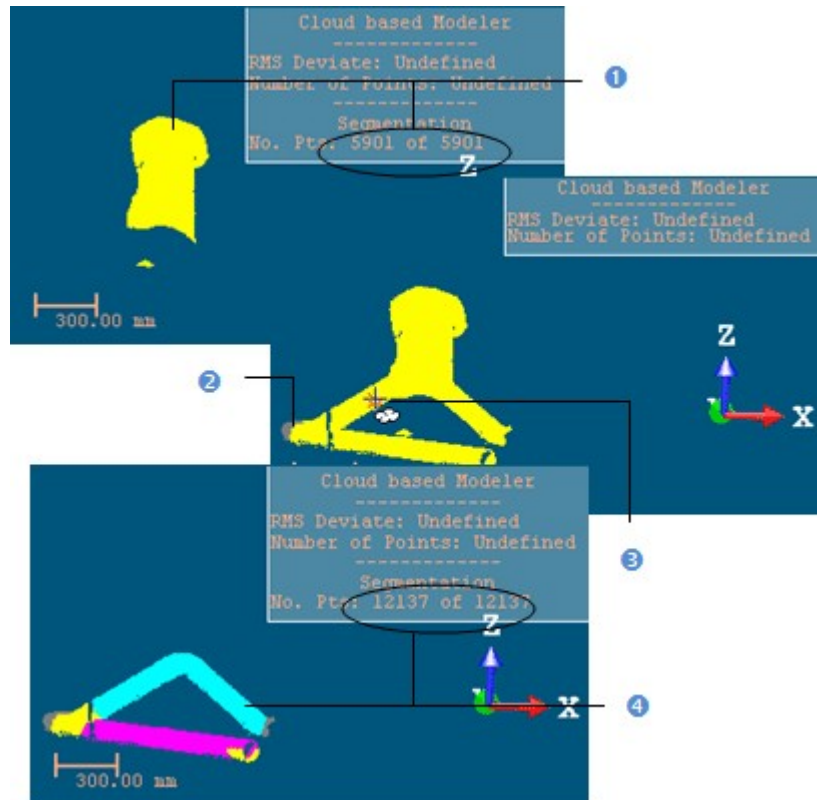
Caution: (**) You can select several point clouds as input of the tool but one of them should not be the **Project Cloud**.

Selecting a New Cloud Data

The **Set New Cloud Data** is for swapping the default **Cloud Data** (not necessary the one selected before starting the **Cloud-Based Modeler Tool**) for another one. You cannot choose and set a subset of the default **Cloud Data** as the new **Cloud Data**; you need to choose a different point cloud.

To Select a New Cloud Data:

1. Select another point cloud from the **Project Tree**.
2. Display the selected point cloud in the **3D View**.
3. If required, hide the default **Cloud Data** (by clicking  in the **Cloud-Based Modeler** dialog).
4. Click the **Set New Cloud Data**  icon. The cursor becomes as shown below and the information box related to the **Segmentation Tool** disappears from the **3D View**.
5. Pick a point on the selected point cloud. It becomes the new **Cloud Data**. The **Segmentation** information box appears again with the new cloud data number of points.



1 - The initial Cloud Data
2 - The newly selected Point Cloud

3 - The cursor in the Set New Cloud Data mode
4 - The new Cloud Data

Note: (*)The **Hide Cloud** icon becomes **Display Cloud** after clicking on it.

Defining a Set of Points on the Cloud Data

Frequently, the **Cloud Data** contains many points; you need to decimate them before doing the fitting. You may also decide to fit a geometry just on a part of it. To do these, you can use the **Segmentation** and the **Sampling** sub-tools.



1 - The Segmentation Tool

2 - The Sampling Tool

After segmenting/sampling the **Cloud Data**, **Keep Only Displayed Cloud in Cloud Data** and **Delete Displayed Cloud from Cloud Data** (respectively for keeping/deleting points in/from the current **Cloud Data** (after decimating/fencing) and **Reload Points** become active.






1 - Reload Points

2 - Keep Only Displayed Cloud in Cloud Data

3 - Delete Displayed cloud from Cloud Data

Keeping Only Displayed Cloud

To Keep Only Displayed Data:

1. Draw a fence on the **Cloud Data**.
2. Use the **In**  (or **Out** )* icon to keep points inside (or outside) the fence.
3. Or sample the **Cloud Data**.
4. Click the **Keep Only Displayed Cloud in Cloud Data**  icon. Points displayed in the **3D View** inside are kept.




Notes:

- **Reload Points** is only for reloading points of the current **Cloud Data** after sampling or segmenting.
- (*) You can skip the step of keeping **In** (or **Out**). In that case, points inside the fence are kept.

Tip: Outside the segmentation mode, you can select the **Segmentation Tool** from the pop-up menu or use its related short-cut key (**S**).

Deleting Displayed Cloud

To Delete the Displayed Cloud:

1. Draw a fence on the **Cloud Data**.
2. Use the **In**  (or **Out** )* icon to keep points inside (or outside) the fence.
3. Or sample the **Cloud Data**.
4. Click the **Delete Displayed Cloud from Cloud Data**  icon. Points displayed in the **3D View** are unkept (removed from the **Cloud Data**).

Notes:

- **Reload Points** is only for reloading points of the current **Cloud Data** after sampling or segmenting.
- (*) You can skip the step of keeping **In** (or **Out**). In that case, points inside the fence are unkept.

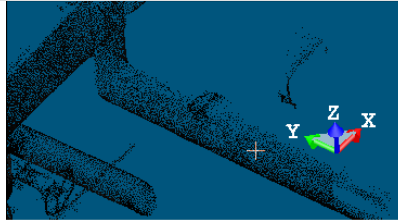
Tip: Outside the segmentation mode, you can select the **Segmentation Tool** from the pop-up menu or use its related short-cut key (**S**).

Extracting an Initial Cylinder by Picking

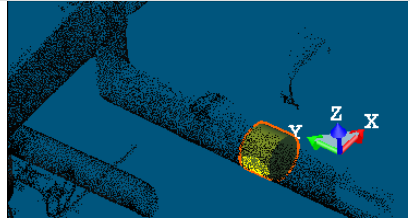
To Extract an Initial Cylinder by Picking:

1. In **Step 1**, click on the **Extract Cylinder** button. The cursor becomes as shown below [A].
2. Pick one point on the set of points. A first cylinder is extracted, the **Start** button in **Step 2** becomes enabled and the **Number of Elements** is equal to **One** [B].

[A]



[B]




Note: An information box at the top right corner of the **3D View** displays the first extracted cylinder parameters like its **Diameter**, the **Number of Points** (used for fitting) and the **Standard Deviation**.

Tip: You can cancel the first extracted cylinder by choosing **Undo** from the pop-up menu or first clicking **Start** again and then picking a point.

Selecting an Initial Cylinder for Tracking

If you already have a cylinder inline with a set of points for which you want to extract a set of cylinders; you can set it as the first cylinder in the tracking process.

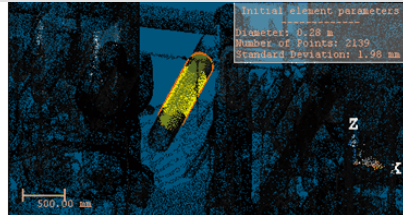
To Select an Initial Cylinder for Tracking:

1. First select a cylinder from the **Project Tree**.
2. Display the cylinder in the **3D View**.
3. In **Step 1**, click on the **Pick a Cylinder**  button. The cursor becomes as shown below [A].
4. Pick a cylinder in the **3D View**. The picked cylinder becomes the first cylinder, the **Start** button in **Step 2** becomes enabled and the **Number of Elements** is equal to **One** [B].

[A]



[B]



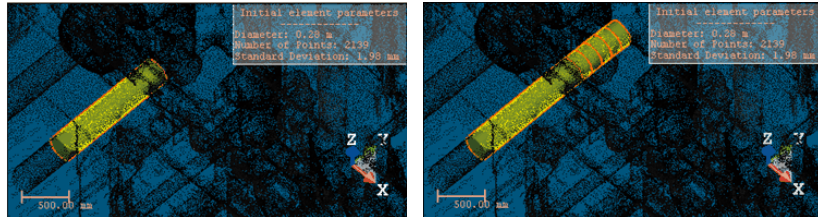
Note: An information box at the top right corner of the **3D View** window displays the information related to the picked cylinder like its **Diameter**, **Number of Points** and **Standard Deviation**.

Starting Tracking Cylinders

The cylinder tracking will consist of building and propagating in both directions a series of consecutive constrained cylinders (all based on the first cylinder and all ball-jointed at a pivot point). The tracking will stop on its own when the fitting error between the current (last) cylinder and its points is too large or the number of points in the immediate neighborhood is insufficient to continue tracking.

To Start Tracking Cylinders:

- In **Step 2**, click on the **Start** button.



Notes:

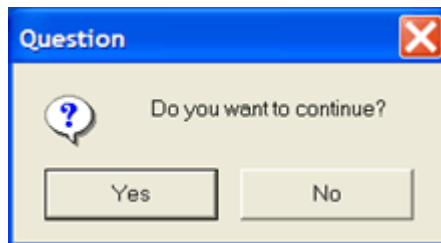
- The **Start** button takes the name of **Pick to Continue**.
- The **Delete Elements**, **Smooth** and **Model** buttons become enabled. The **Number of Elements** will be updated according to the fitted cylinders.

Continuing in Tracking Cylinders

You can continue in extracting cylinders along a line of points for which the extraction has been performed. You have to do this from the first (or the last) extracted cylinder.

To Continue in Tracking Cylinders:

1. Click the **Pick to Continue** button. If the fitting is too important; the extraction stops on its own and the dialog below appears.




2. Click **Yes** to continue.
3. Click **No** to abort.

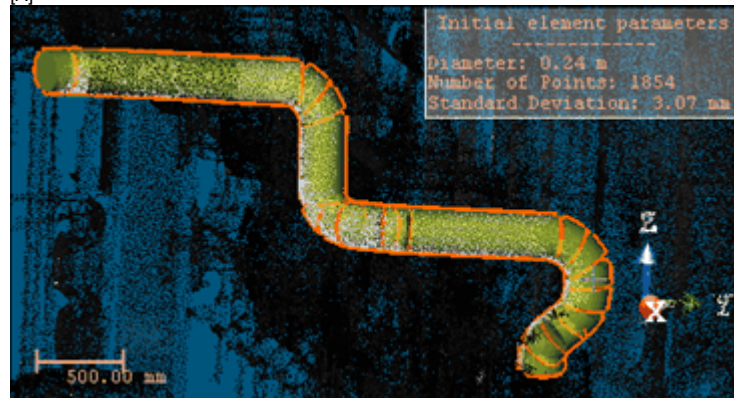
Deleting Extracted Cylinders

You can delete an alone (or a set of) cylinder(s) which is not correctly fitted. You can do this by picking a cylinder in the **3D View**.

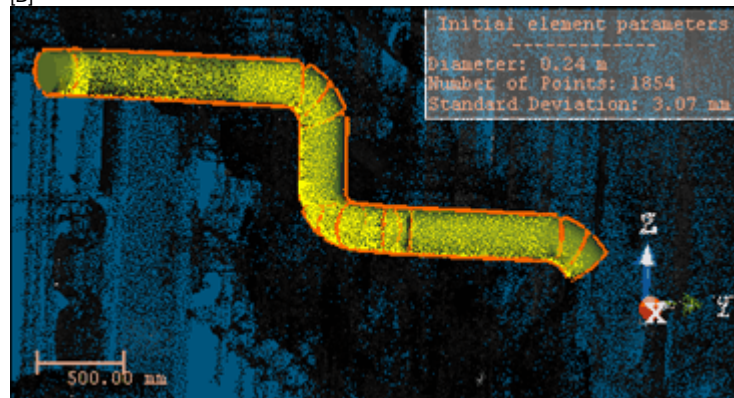
To Delete Extracted Cylinders:

1. Click **Delete Elements** . The mouse cursor becomes as shown below [A].
2. Pick a cylinder in the **3D View**. The picked cylinder and those that follow after are deleted [B].

[A]



[B]



Notes:

- You can undo the deletion by selecting **Undo Delete Elements** from the pop-up menu (or by using the following short-cut **Ctrl + Z**).
- You cannot delete the first cylinder; the one used for tracking. The cursor will stay in the picking mode until a valid cylinder will be selected.

Smoothing Extracted Cylinders

The stack of extracted cylinders may be not aligned; you can then use the **Smooth** command. It allows you to align all cylinder axes together. This is an interactive procedure. You can try as often as you want until you reach the result you need; but applying too many the **Smooth** command consecutively may result in removing valid elbows or deviating cylinders from the initial fitting.

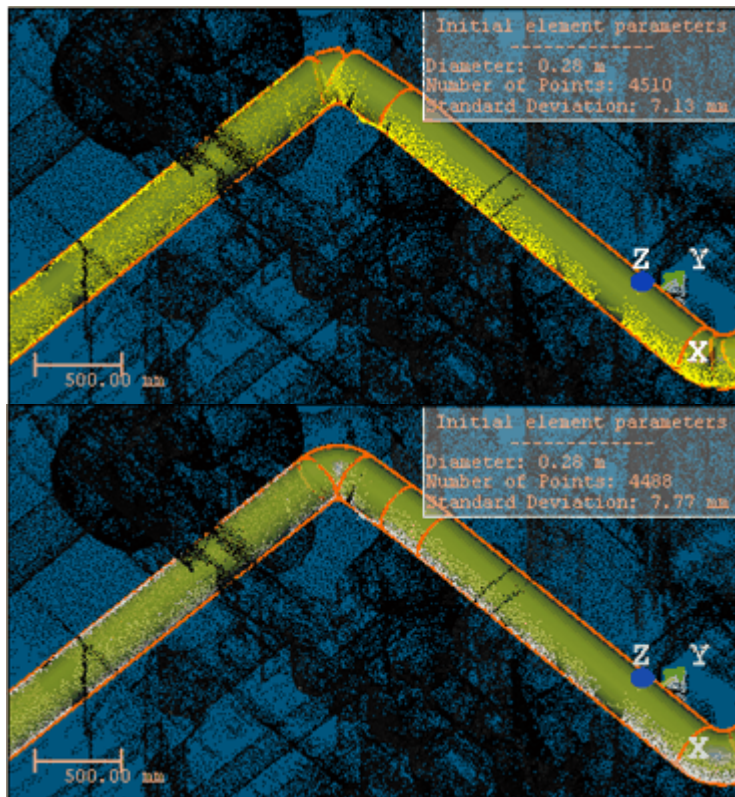
Note: The **Number of Elements** will be updated once the extracted cylinders have been smoothed.

Modeling Extracted Cylinders

The **Modeling** feature is the last step and consists of merging small cylinders into long pipes and replacing small cylinders for which the axes cannot be aligned with an elbow.

To Model the Extracted Cylinders:

- In **Step 3**, click on the **Model** button.



Creating Extracted Cylinders

Once you are satisfied with the tracking result, you can save it in the **RealWorks** database. A new folder named "**Branch (X)**" is created and rooted under the current project. This folder contains all computed cylinders named "**ObjectY**". X and Y are respectively the folder and cylinder order.

To Create Cylinders:

1. Click **Create**. You can still extract another pipe from remaining points. The **EasyPipe** dialog remains open.
2. Click **Close**. The **EasyPipe** dialog closes

Note: If the extracted cylinders have not been modeled; a list of cylinders will be created in **RealWorks**. If the modeling has been applied, a mix of cylinders and circular torus will be created.

Exporting Pipe Center Lines

A **Center Line** is an imaginary line running through the center of a **Pipe**.

To Export Pipe Center Lines:

1. Select a lone (or a set of) fitted **Pipe(s)** from the **Project Tree**.
2. From the **Plant** menu, select **Export Pipe Center Lines**. The **Save As** dialog opens.
3. Navigate to the drive / folder where you want to store the file.
4. Keep the default name which is the project name.
5. Or input a new name in the **File Name** field.
6. Click on the **File of Type** pull down arrow.
7. Choose among **Solids AutoCAD Files (*.dwg)**, **Solids AutoCAD (*.dxf)** and **MicroStation Files (*.dgn)**.
8. Click **Save**.

Note: A unique format file will be exported regardless of the number of **Pipes** selected as input.

As a DWG Format File

AutoCAD's native file format, DWG, and to a lesser extent, its interchange file format, DXF, have become de facto standards for CAD data interoperability. From 1982 to 2007, Autodesk created versions of AutoCAD which wrote no less than 18 major variants of the DXF and DWG file formats. Here below are the numerous versions of AutoCAD.

Product	Version
AutoCAD® 2010	v.u.24
AutoCAD® 2009	v.u.23.1.01
AutoCAD® 2008	v.u.22.1.01
AutoCAD® 2007	v.u.21.1.01
AutoCAD® 2006	v.u.20.1.01
AutoCAD® 2005	v.u.19.1.01
AutoCAD® 2004	v.u.18.1.01
AutoCAD® 2002	v.u.16.1.01
AutoCAD® 2000	v.u.15.0.02
AutoCAD® Release 14	v.u.14.1.04
AutoCAD® Release 13	v.u.13.1.01

To Export a DWG Format File:

1. In the **Export as DWG File** dialog, customize the option(s) below.
 - **Version:** This option allows you to choose from the various versions of AutoCAD.
 - **Export Frame:** A project may contain several coordinate frames. This option allows you to select which coordinate frame you want to apply to the exported data.
 - **Unit:** This option allows you to select the unit system you want to apply to the exported data.
2. Click **Export**. The **Export as DWG File** dialog closes.

As a DXF Format File

AutoCAD's native file format, DWG, and to a lesser extent, its interchange file format, DXF, have become de facto standards for CAD data interoperability. From 1982 to 2007, Autodesk created versions of AutoCAD which wrote no less than 18 major variants of the DXF and DWG file formats. Here below are the numerous versions of AutoCAD.

Product	Version
AutoCAD® 2010	v.u.24
AutoCAD® 2009	v.u.23.1.01
AutoCAD® 2008	v.u.22.1.01
AutoCAD® 2007	v.u.21.1.01
AutoCAD® 2006	v.u.20.1.01
AutoCAD® 2005	v.u.19.1.01
AutoCAD® 2004	v.u.18.1.01
AutoCAD® 2002	v.u.16.1.01
AutoCAD® 2000	v.u.15.0.02
AutoCAD® Release 14	v.u.14.1.04
AutoCAD® Release 13	v.u.13.1.01

To Export a DXF Format File:

1. In the **Export as DXF File** dialog, customize the option(s) below.
 - **Version:** This option allows you to choose from the various versions of AutoCAD.
 - **Export Frame:** A project may contain several coordinate frames. This option allows you to select which coordinate frame you want to apply to the exported data.
 - **Unit:** This option allows you to select the unit system you want to apply to the exported data.
2. Click **Export**. The **Export as DXF File** dialog closes.

As a DGN Format File

DGN for DesiGN is a file format of Bentley MicroStation®. Exporting to this format means exporting a selection from **RealWorks** to the MicroStation® format. You can only export one project at a time. MicroStation® includes the notion of layers which can be used as a tool for organizing and gathering information about a drawing. These layers can be considered as an electronic version of traditional layers. In addition to the layers, this format includes the notion of working units which are the real-world units that you work with in drawing or creating your models in a DGN file. The working units are set as **Master Units** (the largest units in common use in a design, such as meters) and fractional **Sub Units** (the smallest convenient unit to use, such as centimeters or millimeters). The **Sub Units** cannot be larger than **Master Units**.

To Export as DGN Format File:

1. In the **Export as DGN File** dialog, customize the option(s) below.
 - **Layer**: This option allows you to define a number of layers. One is set by default and the option is dimmed.
 - **Export Of**: This option allows you to choose which the type of object(s) you want to export: **Selected Clouds and Geometries**, **Selected Geometries** and **Selected Clouds**. **Selected Geometries** is set by default and the option is dimmed.
 - **Export Frame**: A project may have several coordinate frames. This option allows you to select which coordinate frame from the drop-down list you want to apply to the exported data.
 - **Master Unit**: This option allows you to select a unit system to the **Master** unit.
 - **Sub Unit**: This option allows you to select a unit system to the **Sub** unit.
 - **Positional Unit**: This option allows you to enter a value for the **Positional** unit.
2. Click **Export**. The **Export as DGN File** dialog closes.

CHAPTER 13

Advanced Features

All the **RealWorks** family of products contain the advanced features described in the hereafter topics except for **RealWorks Viewer** which only contains the **Set as Home Frame** feature.

Setting as Home Frame

Each project has a **Home Frame** under which all data reside. If needed, you can select one of the frames of the project and set it as the new **Home Frame**. It is important to note that this operation will transform the coordinates of the whole database to this new frame. You can use this operation to orient, for example, a building scene so that its Z-axis is perpendicular to its ground plane, and its origin is on a specific corner of a building. The difference between this operation and that of setting Active frame is that in the latter case, there is no coordinate transformation.

To Set as Home Frame:

1. Select a coordinate frame from the **List** window.
2. From the **Edit** menu, select **Advanced**. A sub-menu drops down.
3. Select **Set As Home Frame**.

Note: There is no **Undo** for this operation. So you should use it with care.

Equalizing Point Cloud Luminance

You can equalize the intensity of all points of a project. The equalization augments the intensity dynamics to the whole range (0,255). You can see the results immediately if you are in the intensity displaying mode (**Gray Scale Intensity** or **Color Coded Intensity**).

To Equalize Point Cloud Luminance:

1. Select a project from the **Project Tree**.
2. In the **Edit** menu, select **Advanced**. A submenu drops down.
3. Select **Equalize Point Cloud Luminance** from the submenu. A message will prompt you to confirm (or cancel) the operation.
4. Click **Yes** to continue.

Notes:

- There is no **Undo** for this operation. So you should use it with care.
- This operation can take a significant amount of time in the case of large datasets.

Equalizing Point Cloud Color

Generally speaking, data acquired by a 3D laser scanning system (scanner and embedded camera) contain a 3D point cloud and a collection of 2D images. Each point of the point cloud can contain not only its 3D coordinates, but also other attributes such as intensity or color. The intensity information is given by the scanner and the color information by the camera. Point color equalization merges both the intensity information and the color information inside a single project. You can see the result immediately if you are in the intensity displaying mode (**Gray Scale Intensity** or **True Color**).

To Equalize Point Cloud Color:

1. Select a project from the **Project Tree**.
2. In the **Edit** menu, select **Advanced**. A submenu drops down.
3. Select **Equalize Point Cloud Color**. A message prompts you to confirm or cancel the operation.
4. Click **Yes** to continue.

Notes:

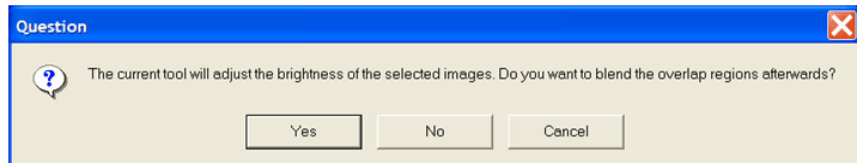
- There is no **Undo** for this operation. So you should use it with care.
- This operation can take a significant amount of time in the case of large datasets.

Equalizing Image Color

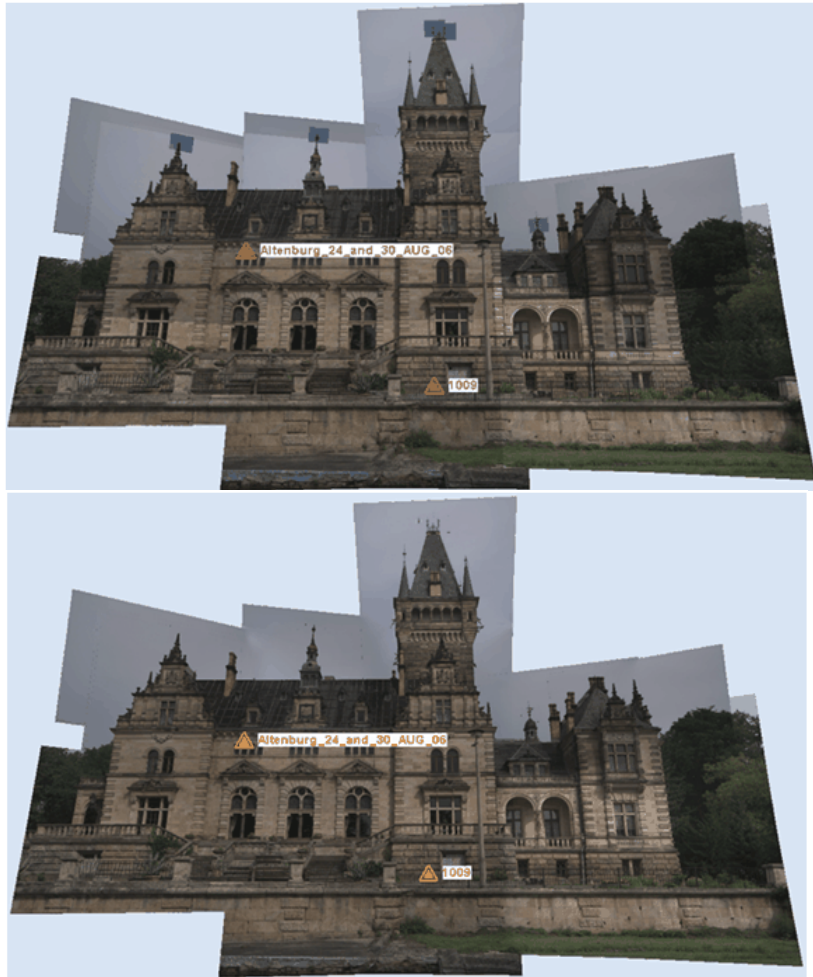
In the **Station-Based** mode, a scene is viewed from a station's point of view with overlapped images in the background (if not hidden). Each image has brightness characteristics different from its neighbor. This tool allows adjustment of the brightness of the images and blending overlapping regions if required.

To Equalize Image Color:

1. Select a set of images (or a project with images inside).
2. Set the **Station-Based** mode (if required).
3. In the **Edit** menu, select **Advanced**. A submenu drops down.
4. Select **Equalize Image Color** from the submenu. The dialog below appears.



- Clicking **Yes** will adjust the brightness of the images and blend the overlapped regions.
- Clicking **No** will only adjust the brightness of the images.



Notes:

- You can undo the operation if required.
- You do not need to display (or open) the selected images to perform this operation.
- The selected images need to belong to a station.

Coloring Points Using Station Images

This feature enables to color the points of a station using the associated images. The only prerequisite is that the images should be matched images. In any processing mode, the user can select a project and the point coloring will be applied to all stations of the project. In **Registration**, the user can select a station (or a set of stations), the point coloring will be applied to the selected station(s). Note that there is no undo. Point coloring is permanent.

To Color Points using Station Images:

1. Select a project from the **Project Tree** (in any processing mode).
2. Or select a station (or a set of stations) from the **Project Tree**.
3. In the **Edit** menu, select **Advanced**. A sub-menu drops down.
4. Select **Color Points Using Station Images** from the sub-menu. A dialog appears and prompts you to continue or to abort the action.

Deleting Geometries

If an object in the **Models Tree** has both representations (geometric and cloud), you can delete its geometric representation.

To Delete a Geometry:

1. Select an object with both properties (cloud and geometric) from the **Project Tree**.
2. In the **Edit** menu, select **Advanced**. A sub-menu drops down.
3. Select **Delete Geometry** from the sub-menu bar.

Notes:

- You can also right-click on an object with both properties (cloud and geometry) in the **Models Tree** (or in the **3D View**) to display the pop-up menu and select **Delete Geometry**.
- You can also use the **Del** key.

Exporting Data

All the **RealWorks** family of products contain the export features described in the hereafter topics except for **RealWorks Viewer**, **Base**, **Advanced** and **Advanced Modeler** which do not contain the **PDMS** export. The table below listed the type of export the user can have according to the license (of **RealWorks**) he has.

Item(s) (to be Exported From)	Viewer	Base	Advanced Advanced - Modeler	Advanced - Plant
Image(s)	✓	✓	✓	✓
Ortho-Image(s)	✓	✓	✓	✓
Measurement(s)	✓	✓	✓	✓
Feature Set(s)	✓	✓	✓	✓
Export TZF/TZS Luminance Images	✓	✓	✓	✓
Convert to BSF		✓	✓	✓
Convert to PTX/PTS		✓	✓	✓
Inspection Map(s)		✓	✓	✓
Inspection Map Horizontal Slice(s)		✓	✓	✓
Inspection Map Vertical Slice(s)		✓	✓	✓

Caution: All data export features are removed from **RealWorks Viewer**.

Caution: For all export features, a dialog opens and prompts you to input a **File Name**. Please, note that you cannot leave the **File Name** field empty. You have to enter a name. Otherwise, you cannot export.

Exporting Selection

The feature "**Export Selection**" is dedicated to the export of a selection from **RealWorks** towards a file for which the format can come from a third party software, a competitor, etc.

To Export a Selection as a File:

1. Select data to be exported from the **Project Tree**.
2. From the **File** menu, select **Export Selection**. The **Export Selection 'Name of Data to be exported'** dialog opens.
3. Click on the **File of Type** pull down arrow.
4. Select a type of file from the drop-down list.
5. Locate a drive/folder to store the file in the **Look In** field.
6. Enter a name in the **File Name** field.
7. Click **Save**. The **Export as File** dialog opens.

Google Earth (KMZ) Format

A **KML** (Keyhole Marked Language) file is a XML-based-language file from **Google Earth** (originally called **Earth Viewer** and created by Keyhole Inc.). **Google Earth** is a virtual globe program which maps the earth by superimposing images obtained from satellite imagery and aerial photography, etc. A KML file contains geo-referenced information (about points, lines and text) to display in **Google Earth**. A **KMZ** file is simply a zip compressed KML file with images. **KMZ** is the default **Google Earth** format. In **RealWorks**, meshes (textured with images or not) and geometries (plane, cylinder, sphere, etc.) can be exported to **Google Earth**.

To Export as a KMZ Format File:

- In the **Export as KMZ File** dialog, do one of the following:

In a basic Geodetic System, a location (or a point) on the Earth has as coordinates its longitude and latitude, both expressed in angles. A latitude is measured from the equator and a longitude from a meridian (the Greenwich meridian is used as reference). There are around a hundred Geodetic Systems in use around the world differing from country to country. A unified Geodetic System (called WGS84, dating from 1984) is in use in Google Earth. In the WGS84 coordinate system, the distance of one degree in longitude changes according to the latitude. This drawback disappears in the Universal Transverse Mercator (UTM) system which is a grid-based method of specifying locations on the surface of the Earth. The surface of the Earth is divided between 80° S latitude and 84° N latitude into 60 zones, each 6° of longitude in width and centered over a meridian of longitude. Zones are numbered from 1 to 60.

- Define your own conversion Parameters.
- Convert UTM coordinates to WGS84 latitude and longitude.

Notes:

- You can also select a project with meshes (or geometries) inside.
- A geometry is converted in a mesh when exporting.

User Defined

If the selected mesh (or geometry) has been geo-referenced in a coordinate system different from the UTM coordinate system (for example the Lambert or the Trimble GX™ scanner), the **Export as KMZ File** dialog opens with the "User Defined" option set as default. Exporting to the Google Earth format involves defining a "**Reference Point**" on the selected mesh/geometry (mainly a point on the ground) and giving its related latitude and longitude coordinates in the WGS84 coordinate system.


To Define your Own Conversion Parameters:

1. Check the **User Defined** option.

2. Enter reference point coordinates in the **Define Reference Point**.
3. Or click on the **Pick Reference Point** icon. The **Picking Parameters** toolbar appears in 3D constraint mode and the cursor shape changes to a pointer.
4. Pick a point on displayed items in the **3D View** (for example the ground). Its coordinates in the current unit of measurement appear in the **Define Reference Point** field.
5. Enter an angle value in **Latitude** field.

6. Click on the **Latitude** pull-down arrow.
7. Choose between **North** and **South** from the drop-down list.
8. Enter an angle value in the **Longitude** field.
9. Click on the **Longitude** pull-down arrow.
10. Choose between **East** and **West** from the drop list.
11. If required, add a description.
12. Click **Export**. The **Export as KML File** dialog closes.

Note: If the selected mesh (or geometry) hasn't been geo-referenced in any coordinate system, the user will have to orientate the scene by himself in **Google Earth**.

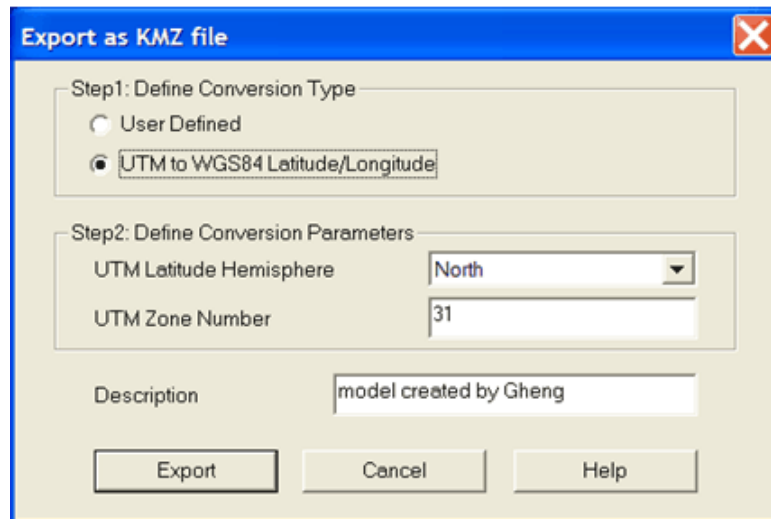
Tip: You can use the **Add Placemark tool**  in **Google Earth** to get the longitude and latitude coordinates of the **Reference Point**.

UTM to WGS84 Latitude and Longitude

If the selected mesh (or geometry) is geo-referenced in the UTM coordinate system, the **Export as KMZ File** dialog opens with the "UTM to WGS84 Latitude/Longitude" option set by default. Exporting to the **Google Earth** format means converting the mesh (or geometry) coordinates expressed in the UTM coordinates to the WGS84 coordinates (latitude, longitude and height).

To Convert UTM Coordinates to WGS84 Latitude and Longitude:

1. Check the **UTM to WGS84 Latitude/Longitude** option.



2. Click on the **UTM Latitude Hemisphere** pull-down arrow.
3. Choose between **North** and **South** from the drop-down list.
4. Enter a number in the **UTM Zone Number** field.
5. If required, add a description.
6. Click **Export**. The **Export as KML File** dialog closes.

BSF Format

You can export a project, a station (or a scan) or an object with point cloud inside from **RealWorks** to a **BSF** format file at once*. This station can be one created from a Trimble FX Controller file or not.

To Export as a BSF Format File:

1. In the **Export to BSF** dialog, click on the **Export Frame** pull-down arrow.
2. Select a frame to apply to the data from the drop-down list.
3. Click on the **Unit** pull-down arrow.
4. Select a unit among **Meter**, **International Foot** or **U.S. Survey Foot**.
5. Click **Export**. The **Export to BSF** dialog closes.

Notes:

- A file of **BSF** format has ***.bsf** as extension.
- A warning message appears if no cloud (or an empty cloud) has been selected.
- (*) You can select several objects as input but only the first (of the selection) is exported.

PDMS Macro Format

You can export any entities created within **RealWorks** into a PDMS macro file like Box, Circular Torus, Cone, Cylinder, Eccentric Cone, Ellipsoid (with or without one or two bounds), Point, Pyramid, Rectangular Torus, Sphere (with or without one or two bounds), Plane and Extrusion..

To Export as a PDMS Macro Format File:

1. In the **PDMS Export** dialog, click on the **Export Frame** pull-down arrow.
2. Select a frame to apply to the exported data from the drop-down list.
3. Click on the **Unit** pull-down arrow.
4. Select a unit* from the drop-down list.
5. Click **Export**. The **PDMS Export** dialog closes.

Notes:

- A file of **PDMS Macro** format carries the **pdmsmac** file extension.
- (*) Only the **Millimeter** unit is available.

E57 Format

The **E57** format is a file format specified by the ASTM (American Society for Testing and Materials), an international standards organization. It is compact and vendor-neutral. It was developed for storing data (Point Clouds, images and metadata) produced by 3D imaging systems such as laser scanners. Such format enables data interoperability among 3D imaging hardware and software systems and is not dependent on proprietary formats for storing and exchanging data.

To Export as an E57 Format File:

1. In the **Export as E57 File** dialog, click on the **Export Frame** pull-down arrow.
2. Choose a frame to apply from the drop-down list.
3. Check an option among **Export Intensity** and **Export RGB Color**.
4. Click **Export**. The **Export as E57 File** dialog closes.

Notes:

- Only an object with a point cloud inside can be exported. If an object has sub-objects with no points inside, a warning message appears and warns the user that this (or these) sub-object(s) are not exported.
- A file of **E57** format carries the **e57** file extension.

Caution: Data are exported to the **E57** format as an irregular grid point sets.

LAS Format

The **LAS** file format is a public file format for the interchange of 3-dimensional point cloud data between data users. It is binary-based.

To Export as a LAS Format File:

1. In the **Export as LAS File** dialog, click on the **Export Frame** pull-down arrow.
2. Choose a frame to apply from the drop-down list.
3. Check an option among **Export Intensity** and **Export RGB Color**.
4. Click **Export**. The **Export as LAS File** dialog closes.

Notes:

- Only an object with a point cloud inside can be exported. If an object has sub-objects with no points inside, a warning message appears and warns the user that this (or these) sub-object(s) are not exported.
- A file of **LAS** format has *.las as extension.
- Data and bounding box limits are exported in meter.

AutoCAD PCG format (AutoDesk Revit MEP)

Before being able to be used in an **AutoCAD** drawing, points are first exported into a **LAS** format file which is an **ASCII** based format file. This file, which is a temporary file, is then converted as a **PCG** format file. This conversion step is called **Indexing**. Once completed, the PCG format file is attached to the drawing. This is similar to attaching an external reference (e.g. an image file) to the drawing.

To Export as a PCG Format File:

1. In the **Export as PCG File** dialog, click on the **Export Frame** pull-down arrow.
2. Select a frame to apply to the data to export.
3. Check an option among the options below:
 - **Export Intensity.**
 - **Export RGB Color.**
4. Click **Export**. The **File Indexing** dialog opens.
5. Click **Close**.

Notes:

- You should have **AutoDesk Revit MEP** (Vers. 2013) installed on your computer.
- Data and bounding box limits are exported in meter.

Alias/WaveFront (OBJ) Format

Alias/Wavefront is a provider of 2D/3D graphics technology for the film, video, games, interactive media, industrial design, automotive industry and visualization markets. Their .OBJ ASCII file format is widely accepted for exchanging graphical data between drafting applications. OBJ files contain solids which are made up of 3 or 4 sided faces. Only meshes can be exported to this format.

To Export as an Alias/WaveFront (OBJ) Format File:

1. In the **Export as OBJ File** dialog, click on the **Export Frame** pull-down arrow.
2. Select a coordinates frame to apply from drop-down list.
3. Click on the **Unit** pull-down arrow.
4. Select a unit system to apply in the **Unit** field.
5. Click **Export**.

Notes:

- A file of **Alias/WaveFront (OBJ)** format carries the **obj** file extension.
- Only a mesh can be exported. Trying to export anything else makes an error dialog appeared.

MicroStation (DGN) Format

DGN for DesiGN is a file format of Bentley MicroStation®. Exporting to this format means exporting a selection from **RealWorks** to the MicroStation® format. You can only export one project at a time. MicroStation® includes the notion of layers which can be used as a tool for organizing and gathering information about a drawing. These layers can be considered as an electronic version of traditional layers. In addition to the layers, this format includes the notion of working units which are the real-world units that you work with in drawing or creating your models in a DGN file. The working units are set as **Master Units** (the largest units in common use in a design, such as meters) and fractional **Sub Units** (the smallest convenient unit to use, such as centimeters or millimeters). The **Sub Units** cannot be larger than **Master Units**.

To Export as a DGN Format File:

1. Select data to be exported from the **Models Tree**.
2. In the **File** menu, select **Export Selection**. The **Export** dialog opens.
3. Click on the **File of Type** pull down arrow.
4. Select **MicroStation Files (*.DGN)** as file type.
5. Locate a drive/folder to store the file in the **Look In** field.
6. Enter a name in the **File Name** field.
7. Click **Save**. The **Export as DGN File** dialog opens.
 - **Layer**: This option allows you to define a number of layers.
 - **Export Of**: This option allows you to choose which kind of objects you want to export: **Selected Clouds and Geometries**, **Selected Geometries** and **Selected Clouds**.
 - **Export Frame**: A project may have several coordinate frames. This option allows you to select which coordinate frame from the drop-down list you want to apply to the exported data.
 - **Master Unit**: This option allows you to select a unit system to the **Master** unit.
 - **Sub Unit**: This option allows you to select a unit system to the **Sub** unit.
 - **Positional Unit**: This option allows you to enter a value for the **Positional** unit.
8. Select the kind of objects to be exported in the **Export of** field.
9. Select the coordinates frame to be applied in the **Export Frame** field.
10. Select the unit system to be applied for the master unit in the **Master Unit** field.
11. Select the unit system to be applied for the sub unit in the **Sub Unit** field.
12. Enter a value for the positional unit in the **Positional Unit** field.
13. Click **Export**.

Note: You can also select and export a scan/station of a project in the **Models** (or **Scans**) **Tree**.

AutoCAD (DXF) Format

DXF for Drawing eXchange Format is an ASCII file format of an AutoCAD® drawing file. Exporting to the DXF format means to export a selection from **RealWorks** to the AutoCAD® application. AutoCAD® includes the notion of layers which can be used as a tool for organizing and gathering information about a drawing. These layers can be considered as an electronic version of traditional layers. The selection hierarchy is preserved during the export; each group or lone object has its own layer. You can only export one project at a time; in this case every type of object (geometries and clouds) in the project can be exported, except frames, measurements, feature code sets and registration entities.

To Export as a DXF Format File:

1. In the **Export as DXF File** dialog, choose an option among those listed below.
 - **Export Of:** This option allows you to choose which kind of objects you want to export: **Selected Clouds and Geometries**, **Selected Geometries** or **Selected Clouds**.
 - **Export Frame:** A project may contain several coordinate frames. This option allows you to select which coordinate frame you want to apply to the exported data.
 - **Unit:** This option allows you to select the unit system you want to apply to the exported data.
 - **Cloud Rendering:** This option allows you to select a rendering that will be applied to the exported point cloud.
 - **Export Coplanar Polylines as 3D DXF Polylines:** Coplanar polylines will be exported as 3D DXF polylines. That is to say that all circle arcs will be discretized in segments.
 - **Export Coplanar Polylines in XY Plane:** All nodes of the polyline(s) are exported in the XY plane.
2. Click **Export**. The **Export as DXF File** dialog closes.

Notes:

- You can also select and export a scan/station of a project in the **Models** (or **Scans**) **Tree**.
- Selecting **Selected Geometries** in the **Export Of** field will gray out the **Cloud Rendering** field.
- Selecting **Selected Geometries** in the **Export Of** field will gray out both the **Export Coplanar Polylines as 3D DXF Polylines** and **Export Coplanar Polylines in XY Plane** options.

ASCII Format

ASCII is the acronym for American Standard Code for Information Interchange. Exporting to this file format involves exporting a selection from **RealWorks** to the ASCII format. You can only export one project at a time; in this case only point clouds are exported. In such conditions, an ASCII file format is composed of a header (mainly comments) and a set of lines. Each line is composed of one point with coordinates (X, Y and Z), and where present, attributes like intensity, normal or color.

To Export as an ASCII Format File:

1. In the **Export as ASCII File** dialog, choose an option among those listed below.
 - **Export Frame** allows you to select which frame will be applied to the exported data.
 - **Unit** allows you to select the unit system that will be applied to the exported data.
 - **Separator** allows you to specify a separator to set between each value (**Semicolon**, **Comma**, **Tabulation** and **Space**).
 - **Decimal Char** allows you to specify the decimal char (**Point** or **Comma**).
 - **Coordinate System** allows you to choose between **Cartesian system** (X, Y and Z) and **Global system** (also called **Geodetic Northing**, **Easting**, and **Elevation system**).
 - **Decimal Places** allows you to define the decimal places.
 - **Export Intensity** allows you to export data with intensity attributes.
 - **Export Normal** allows you to export data with normal attributes.
 - **Export RGB** color allows you to export data with RGB color attributes.
2. Click **Export**. The **Export as ASCII File** dialog closes.

Note: You can also select and export a scan/station of a project in the **Models** (or **Scans**) **Tree**.

LandXML Format

Several companies, including Autodesk, teamed up to create a method for exchanging project information across different software packages and LandXML is the result. LandXML is a generic, text-based file format used to save project data. It is similar to a DXF™ file, which is a generic file format for vector-based drawing information.

To Export as a LandXML Format File:

1. Select a **Mesh** (or a **Polyline**)* from the **Models Tree**.
2. From the **File** menu, select **Export Selection**. The **Export Selection** dialog opens.
3. Click on the **File of Type** pull-down arrow.
4. Select **LandXML Files (*.xml)** as file type.
5. Locate a drive/folder to store the file in the **Look In** field.
6. Enter a name in the **File Name** field.
7. Click **Save**. The **Export Selection** dialog closes.
 - Each **Mesh** is exported as a surface,
 - Each break-line (a 3D **Polyline** or a 2D **Polyline** with a **Normal** direction different from the **Z**-axis) is exported as a surface,
 - All **Contours** (**Polylines** with a **Normal** direction parallel to the **Z**-axis) are exported as a surface.

Note: (*) Selecting anything else will generate the following warning message "**Name** of the selected item has not been exported".

Solids for AutoCAD

AutoCAD's native file format, DWG, and to a lesser extent, its interchange file format, DXF, have become de facto standards for CAD data interoperability. From 1982 to 2007, Autodesk created versions of AutoCAD which wrote no less than 18 major variants of the DXF and DWG file formats. Here below are the numerous versions of AutoCAD.

Product	Version
AutoCAD® 2010	v.u.24
AutoCAD® 2009	v.u.23.1.01
AutoCAD® 2008	v.u.22.1.01
AutoCAD® 2007	v.u.21.1.01
AutoCAD® 2006	v.u.20.1.01
AutoCAD® 2005	v.u.19.1.01
AutoCAD® 2004	v.u.18.1.01
AutoCAD® 2002	v.u.16.1.01
AutoCAD® 2000	v.u.15.0.02
AutoCAD® Release 14	v.u.14.1.04
AutoCAD® Release 13	v.u.13.1.01

In **RealWorks**, the user can export the primitives listed in the table below.

Primitive	Special Case	Export DXF1	Export DWG
Box		Polygon Mesh	3D Solid
Composite Curve3		Polyline	(LW) Polyline4
Circle Arc	Full Arc	Arc Arc	Circle Arc
Circular Torus		Polygon Mesh	3D Solid
Cloud		N Points	N Points
Cloud 2D		N Points	N Points
Cone		Polygon Mesh	3D Solid
Cylinder	No Bounds With Bounds	Circle with Thickness Polygon Mesh	3D Solid Polyface Mesh
Eccentric Cone		Polygon Mesh	Polyface Mesh
Ellipse Arc		Polyline	Ellipse
Ellipsoid		Polygon Mesh	Polyface Mesh
Extrusion6	Closed Open & Bound (DWG)	Polyface Mesh Polyface Mesh	3D Solid Polyface Mesh
Inspection 1D		N Lines (LW)Polyline4	N Lines (LW)Polyline4
Inspection 2D		Image & TIF image	Image & TIF image
Mesh		Polyface Mesh (Max: 32767 vertices)	Polyface Mesh (Max: 32767 faces)
Ortho-Image		Image & TIF image	Image & TIF image
Plane(6)	No Holes With Holes	Polyface Mesh Polyface Mesh	Region N Regions
Point		Point	Point
Polyline		(LW)Polyline4	(LW)Polyline4
Pyramid		Polygon Mesh	Polyface Mesh

Rectangular Torus		Polygon Mesh	3D Solid
Segment		Line	Line
Sphere	No Bounds With Bounds	Polygon Mesh Polygon Mesh	3D Solid Polyface Mesh
Volume		N (LW)Polyline ⁴	N (LW)Polyline ⁴

Notes:

- ¹ Export in DXF version R14.
- ² Possibility to export in DWG version R12, R13, R14, R15 (2000/2002), R18 (2004/2005), R21 (2007) and R24 (2010).
- ³ DWG export case: ellipse arcs are exported as segments from start to end point.
- ⁴ LWPolyline by default, Polyline when exporting coplanar polyline as 3D Polyline checked.
- ⁵ DWG export case: Less powerful than DXF export. Use the last R24 version for best result.
- ⁶ DWG export case: Full ellipses well managed, ellipse arcs are exported as segments.

AutoCAD (DWG) Format

DWG - for DraWinG - is a binary file format used by AutoDesk's AutoCAD software. It can contain 2D or 3D objects. Exporting to the **DWG** format means to export a selection from **RealWorks** to the AutoCAD application. AutoCAD includes the notion of layers which can be used as a tool for organizing and gathering information about a drawing. These layers can be considered as an electronic version of traditional layers. The selection hierarchy is preserved during the export; each group or lone object has its own layer. You can only export one project at a time; in this case every type of object (geometries and clouds) in the project can be exported, except frames, measurements, feature code sets and registration entities.

To Export to Solids for AutoCAD (DWG) Format:

1. Select data to be exported from the **Models Tree**.
2. From the **File** menu, select **Export Selection**. The **Export Selection 'Name of Data to be exported'** dialog opens.
3. Click on the **File of Type** pull down arrow.
4. Select **Solids for AutoCAD Files (*.dwg)** as file type.
5. Locate a drive/folder to store the file in the **Look In** field.
6. Enter a name in the **File Name** field.
7. Click **Save**. The **Export as DWG File** dialog opens.
 - **Version:** This option allows you to choose from the various versions of AutoCAD.
 - **Export Of:** This option allows you to choose which kind of object you want to export: **Selected Clouds and Geometries**, **Selected Geometries** or **Selected Clouds**.
 - **Export Frame:** A project may contain several coordinate frames. This option allows you to select which coordinate frame you want to apply to the exported data.
 - **Unit:** This option allows you to select the unit system you want to apply to the exported data.
 - **Cloud Rendering:** This option allows you to select a rendering that will be applied to the exported point cloud.
 - **Export Coplanar Polylines as 3D DXF Polylines:** Coplanar polylines will be exported as 3D DXF polylines. That is to say that all circle arcs will be discretized in segments.
 - **Export Coplanar Polylines in XY Plane:** All nodes of the polyline(s) are exported in the XY plane.
8. Select the kind of object to be exported in the **Export Of** field.
9. Select the coordinate frame to be applied in the **Export Frame** field.
10. Select the unit system to be applied in the **Unit** field.
11. Select a rendering for point clouds.

12. Check the **Export Coplanar Polylines As 3D DXF Polylines** option, if required.
13. Check the **Export Coplanar Polylines in XY Plane** option, if required.
14. Click **Export**. The **Export as DWG File** dialog closes.

Notes:

- You can also select and export a scan/station of a project in the **Models** (or **Scans**) Tree.
- Selecting **Selected Geometries** in the **Export Of** field will gray out the **Cloud Rendering** field.
- Selecting **Selected Geometries** in the **Export Of** field will gray out both the **Export Coplanar Polylines as 3D DXF Polylines** and **Export Coplanar Polylines in XY Plane** options.

AutoCAD (DXF) Format

DXF - for Drawing eXchange Format - is an ASCII file format of an AutoCAD® drawing file. Exporting to the DXF format means to export a selection from **RealWorks** to the AutoCAD application. AutoCAD includes the notion of layers which can be used as a tool for organizing and gathering information about a drawing. These layers can be considered as an electronic version of traditional layers. The selection hierarchy is preserved during the export; each group or lone object has its own layer. You can only export one project at a time; in this case every type of object (geometries and clouds) in the project can be exported, except frames, measurements, feature code sets and registration entities.

To Export to Solids for AutoCAD (DXF) Format:

1. Select data to be exported from the **Models Tree**.
2. From the **File** menu, select **Export Selection**. The **Export Selection 'Name of Data to be exported'** dialog opens.
3. Click on the **File of Type** pull down arrow.
4. Select **Solids for AutoCAD Files (*.dxf)** as file type.
5. Locate a drive/folder to store the file in the **Look In** field.
6. Enter a name in the **File Name** field.
7. Click **Save**. The **Export as DWG File** dialog opens.
 - **Version:** This option allows you to choose from the various versions of AutoCAD
 - **Export Of:** This option allows you to choose which kind of object you want to export: **Selected Clouds and Geometries**, **Selected Geometries** or **Selected Clouds**.
 - **Export Frame:** A project may contain several coordinate frames. This option allows you to select which coordinate frame you want to apply to the exported data.
 - **Unit:** This option allows you to select the unit system you want to apply to the exported data.
 - **Cloud Rendering:** This option allows you to select a rendering that will be applied to the exported point cloud.
 - **Export Coplanar Polylines as 3D DXF Polylines:** Coplanar polylines will be exported as 3D DXF polylines. That is to say that all circle arcs will be discretized in segments.
 - **Export Coplanar Polylines in XY Plane:** All nodes of the polyline(s) are exported in the XY plane.
8. Select the kind of object to be exported in the **Export of** field.
9. Select the coordinate frame to be applied in the **Export Frame** field.
10. Select the unit system to be applied in the **Unit** field.
11. Select a rendering for point clouds.

12. Check the **Export Coplanar Polylines As 3D DXF Polylines** option, if required.
13. Check the **Export Coplanar Polylines in XY Plane** option, if required.
14. Click **Export**. The **Export as DXF File** dialog closes.

Notes:

- You can also select and export a scan/station of a project in the **Models** (or **Scans**) **Tree**.
- Selecting **Selected Geometries** in the **Export Of** field will gray out the **Cloud Rendering** field.
- Selecting **Selected Geometries** in the **Export Of** field will gray out both the **Export Coplanar Polylines as 3D DXF Polylines** and **Export Coplanar Polylines in XY Plane** options.

Advanced Exports

The features in **Specific Exports** are dedicated to the export of an object (or set of objects) created within **RealWorks**.

Object Properties

You can export an item (or a set of items) properties as a report in the **RTF** file format. These properties are those found in the **Property** window. Any selection (or multi-selection) from the **Project Tree** except the **WorkSpace** node (also from the **Project Tree**) swaps the **Export Object Properties** command to enabled.

To Export Object Properties:

1. Select a project (or group, station or other item) from the **Project Tree**.
2. From the **File** menu, select **Advanced Exports Export / Object Properties**. The **Export** dialog opens.
3. Locate a drive/folder to store the file in the **Look In** field.
4. Enter a name in the **File Name** field.
5. In the **Options** panel, choose one of the following:
 - **Selection**: The selected item properties are exported. If there are sub-items under the selected item; all the sub-items properties are exported too. If there is no sub-item under the selected item, only its own properties are exported.
 - **Projection of Selection**: The properties of all items and subitems that belong the project the selection is from are exported.
 - **All Project Trees**: The properties of all items and subitems that belong to the **Scans**, **Targets**, **Models** and **Images** sub-trees are exported.
6. Check the **Show File** option to automatically open the report after exporting.
7. Click **Save**. The **Export** dialog closes.

Images

You can export a single image (or a set of images) to the **JPEG** format (with **JPG** as extension).

To Export an Image:

1. Select an image (or a set of images) from the **Images Tree**.
2. From the **File** menu, select **Advanced Exports / Export Image**. The **Export Image** dialog opens.
3. Locate a drive/folder to store the file in the **Look In** field.
4. Enter a name in the **File Name** field.
5. Click **Save**.
 - If only one image has been selected, the **Export Image** dialog closes.
 - If a set of images has been selected, the **Export Image** dialog displays the next image to save and so on.

Tip: You can right-click on an image in the **Images Tree** and select **Export Image** from the pop-up menu.

Ortho-Images

Tiff (or **Tif**) is the acronym for Tagged Image File Format. It is one of the most popular and flexible current public domain raster file format. Exporting an ortho-image involves saving it in a Tiff (or Tif) format. With the Tiff image, is created a file of the same name and with the TXT extension. This file contains the four corners for the Tiff image: **Top Left**, **Top Right**, **Bottom Left** and **Bottom Right**. These corners are useful for locating an ortho-image in 3D.

To Export an Ortho-Image:

1. Select an ortho-image from the **Images Tree**.
2. From the **File** menu, select **Advanced Exports / Export Ortho-Image**. The **Export Ortho-Image** dialog opens. The **Tag Image Format Files (*.TIF)** type is default-set in the **File of Type** field.
3. Locate a drive/folder to store the ortho-image in the **Look In** field.
4. Enter a name in the **File Name** field.
5. Click **Save**.

Notes:

- You can export an ortho-image as a 3D Tiff. To do this, from the **File** menu, select first **Export Selection**, then **AutoCAD Files (*.dxf)** as file's type.
- If a set of ortho-images has been selected, the export is done one by one from the first to the last. The user only needs to click **Save** for each ortho-image.

Tip: You can right-click on an ortho-image from the **Images Tree** and select **Export Ortho-Image** from the pop-up menu.

Measurements

You can export a measurement (or a set of measurements) as a report in the **Excel** format (*.CSV files). Only measurements from an active project (selected project) are exported. For a **Point to Point Distance Measurement**, its type, name, length, delta X, delta Y, delta Z, extremity 1 and extremity 2 values are exported. For an **Angular Measurement**, its type, name and the angle value are exported. For a **3D Point Measurement**, its type, name and X,Y,Z values. For an **Orientation Measurement**, its type, name and the center, azimuth angle and tilt angle values are exported.

To Export a Measurement:

1. Select a measurement from the **Models Tree**.
2. From the **File** menu, select **Advanced Exports / Export Measurements**. The **Export Measurements** dialog opens.
3. The **Excel Files (*.CSV)** type is default-set in the **File of Type** field.
4. Type a name in the **File Name** field.
5. Locate a drive/folder to store the file in the **Look In** field.
6. Click **Save**. The **Export Measurements** dialog opens.
 - **Export Frame**: This option enables to select a frame that will be applied to the exported data.
 - **Separator***: A separator can be a **Semicolon**, **Comma** or **Tabulation**.
 - **Decimal Char**: A decimal symbol can be either a **Point** or a **Comma**.
 - **Options****: There are four types of measurement to export: **Export Point to Point Distance Measurement**, **Export Angular Measurement**, **Export 3D point Measurement** and **Export Orientation Measurement**. Only one type can be selected at a time.
7. Select a frame to apply from the **Export Frame** field.
8. Select a separator to use from the **Separator** field.
9. Select a decimal char to apply from the **Decimal Char** field.
10. Check an option.
11. Click **Export**.

Notes:

- (*) The required **Separator** when exporting to the CSV format is a **Comma**. If the user selects a **Semicolon** (or **Tabulation**) **Separator** instead, a warning message appears and advises the user that the measurement(s) will be exported as a TXT format report.
- (**) When you select a type of measurement from the **Models Tree**, only the option (from the **Options** panel) of the same type is enabled and the other options are dimmed. When you select a set of measurements that covers all types, the four options are enabled. You need to choose an option; otherwise the Export button remains dimmed.

- You cannot combine a Comma Separator with a Comma Decimal Char.

Feature Sets

Generally, a **Feature Point** is composed of three items: **Point Number**, **Point Coordinates** and **Point Feature Code**. Name and description are optional. Exporting a **Feature Set** from **RealWorks** involves exporting such set of information in an ASCII file format.

To Export a Feature Set:

1. Select a feature sets from the **Models Tree**.
2. From the **File** menu, select **Advanced Exports / Export Feature Sets**. The **Feature Set ASCII Export** dialog opens. The **ASCII Files (*.ASC)** type is default-set in the **File of Type** field.
3. Locate a drive/folder to store the file in the **Look In** field.
4. Enter a name in the **File Name** field. The ASC extension is added automatically.
5. Click **Save**. The **ASCII Feature Set Export** dialog opens.
 - **Export Frame:** This option enables to select a frame that will be applied to the exported data.
 - **Unit:** This option enables to select the unit system that will be applied to the exported data.
 - **Separator:** A separator can be **Semicolon**, **Comma** and **Tabulation**.
 - **Decimal Char:** A decimal char can be either a **Point** or a **Comma**.
 - **Feature Set Export Options:** Options in that panel enables to express a **Feature Set** in either the **Cartesian** coordinate system or the **Global** coordinates system. If required, the user can add a description.
6. Select a coordinate frame to apply from the **Export Frame** field.
7. Select a unit system to use from the **Unit** field.
8. Select a separator to use from the **Separator** field.
9. Select a decimal char to apply from the **Decimal Char** field.
10. Check an option.
11. Click **Export**.

Notes:

- You can only export one **Feature Set** at a time. If you select a set of **Feature Sets**, only the last (from the selection list) is exported.
- The **Dash-Line Segments** (or **Continuous Segments**) that have been chosen to link each **Feature Point** in the **Feature Set Tool** are not exported.
- You cannot combine a **Comma Separator** with a **Comma Decimal Char**.

Warning: A warning message appears if the selection (as input) is not a feature set and the export is aborted.

Exporting TZF/TZS Luminance Images

Each **TZF** format has a preview which is a **2D Image Data**. You can export this preview as a **Bitmap** image file.

To Export TZF/TZS Luminance Images:

1. Select a project or a station (or a set of stations)*.
2. From the **File** menu, select **Advanced Export / Export TZF/TZS Luminance Images**. The **Select New File Folder** dialog opens.
3. Navigate to the drive/folder where you want the image files to be stored in the **In** field.
4. Select the folder and click **Open**.
 - For each **TZS** (or **TZF**) format file, a Bitmap image file.
 - Each Bitmap file has as name its related **TZS** (or **TZF**) file.

Notes:

- (*) A warning dialog opens if there is no **TZF Scan** within your selection.
- (*) If the **TZF** format file(s) has (have) not been yet processed, the **Processing TZF Scans** dialog opens and prompts you to proceed to do so.

Converting to BSF

This tool provides the user with the ability to convert a station* (or set of stations* or a project) created from files of **TZS** (or **TZF**) format downloaded in **RealWorks** to LaserGen file(s) (*.BSF). If the stations have been registered within **RealWorks**, the registration parameters will be taken into account when converting.

To Convert to BSF Format File:

1. Select a station (or a set of stations or a project) from the **Project Tree**.
2. From the **File** menu, select **Advanced Exports / Convert to BSF**.
 - If a station (or a set of stations) has been selected as input, the **BSF Conversion Using Registered Stations** dialog opens.
 - If a project has been selected as input, a dialog opens and asks you if you want to process all stations or not. Click **Yes**. The **BSF Conversion Using Registered Stations** dialog opens.

Notes:

- The stations can be empty.
- (*) If the **TZF** format file(s) has (have) not been yet processed, the **Processing TZF Scans** dialog opens and prompts you to proceed to do so.



Filtering

The procedures hereafter are in option. When the user decides to not apply the filter (or the sampling); the whole scan data will be taken.

Spatial Sampling

The **Spatial Sampling** method allows you to obtain a point cloud with a homogeneous spatial density that you have to define.



To Sample Spatially:

1. In the **Filtering (Optional)** panel, check the **Spatial Sampling** option. The **Resolution** field becomes enabled.
2. Enter a value in the **Resolution** field.
3. Or use the **Up**  (or **Down** ) button to choose a value.

Filtering by Range

The **By Range** allows you to define a distance (from the center of the FX instrument) beyond which no point will be taken into account. This filter is only applied to the scan data.

To Filter by Range:

1. Check the **Filter by Range** option. The **Max Distance** field becomes editable.
2. Enter a value in the **Max Distance** field.
3. Or use the **Up**  (or **Down** ) button to choose a value.

Filtering by Zone

The **By Zone** option allows filtering by defining a bounding box. The **Min Point** and **Max Point** are the two extremities of a bounding box diagonal.

To Filter by Zone:

1. Check the **By Zone** option. The **Min Point** and **Max Point** fields become editable.
2. Enter a 3D coordinates in the **Min Point** field.
3. Enter a 3D coordinates in the **Max Point** field.

Intensity Mapping

You can define the **Output Intensity** of points for each **TZS** (or **TZF**) format file. This means that all points will be scaled up (or down) to the Intensity defined by the user.

To Define the Output Intensity of Points:

1. Click on the **Output Intensity Max** pull-down arrow.
2. Choose a value from **256** to **8192**.







Output Files

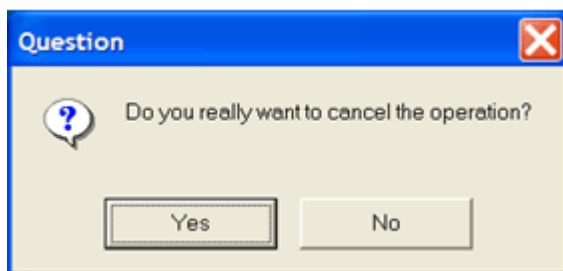
The **Output Files** panel lets the user to choose between two methods for converting **TZS** (or **TZF**) format files to **BSF** format files: **By Volume** or **By Stations**. If required, the user can generate a preview and define a location where the conversion will be placed.

By Volume

This feature allows you to split a single (or a set of) registered station(s) along the **X**, **Y** and **Z** axes of the current coordinate system into a set of cloud data. Each corresponds to a **BSF** format file and as name: **Cloud_Xn_Yn'_Zn''**. All will be put under the **Scan Folder** (defined in the **Output Files** panel). If the cloud data is empty; no **BSF** format file will be created.

To Convert the TZS (or TZF) Format Files into a Set of Cubes:

1. In the **Output Files** panel, check the **By Volume** option (if required).
2. In the **Volume Properties** panel, click on the pull-down arrow.
3. Choose between **Number of Cubes** and **Cube Size**.
 - If the **Number of Cubes** option has been selected:
 - a) Enter a number value in the **Number in X** field.
 - b) Or you can use the **Up**  (or **Down** ) button to select a number value.
 - c) Repeat the steps for the **Number in Y** and **Number in Z** fields.
 - If the **Cube Size** option has been selected:
 - a) Enter a length value in the **Length in X** field.
 - b) Or you can use the **Up**  (or **Down** ) button to select a length value.
 - c) Repeat the steps for the **Length in Y** and **Length in Z** fields.
4. Enter a number value in the **Minimum of Points in Cube for File Creation** field.
5. Or use the **Up**  (or **Down** ) button to select a number value.
6. Click **Convert**.
 - The conversion is done from the first **TZS** (or **TZF**) format file to the last. For each **TZS** (or **TZF**) format file, there are two stages.
 - You can abort the conversion operation by pressing **Esc**, only when the first stage is in progress. In this case, the following dialog appears.



- The Trimble Scan Files to BSF Conversion Using Registered Stations dialog closes on its own (before the end of the conversion).

		Name	Size	Type
+	TZS Files 4			
	+	CAD		
		PROJECT1.rwl		
+	Scans			
		Preview		
		PROJECT1		
		cloud_X0_Y0_Z0.BSF	675 KB	LASERGenViewerDocu...
		cloud_X0_Y0_Z1.BSF	743 KB	LASERGenViewerDocu...
		cloud_X0_Y1_Z0.BSF	4 510 KB	LASERGenViewerDocu...
		cloud_X0_Y1_Z1.BSF	19 207 KB	LASERGenViewerDocu...
		cloud_X1_Y0_Z0.BSF	110 KB	LASERGenViewerDocu...
		cloud_X1_Y0_Z1.BSF	322 KB	LASERGenViewerDocu...
		cloud_X1_Y1_Z0.BSF	2 440 KB	LASERGenViewerDocu...
		cloud_X1_Y1_Z1.BSF	2 409 KB	LASERGenViewerDocu...

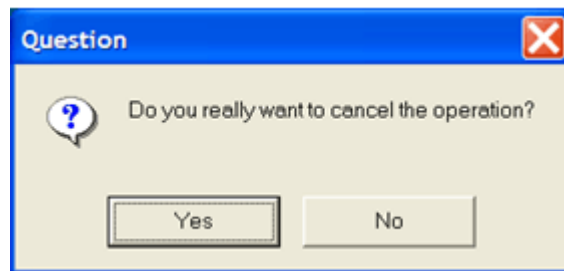
Note: If the Coordinate System in Preferences is "North, East, Elevation"; you will have N, E and El in place of X, Y and Z.

By Station

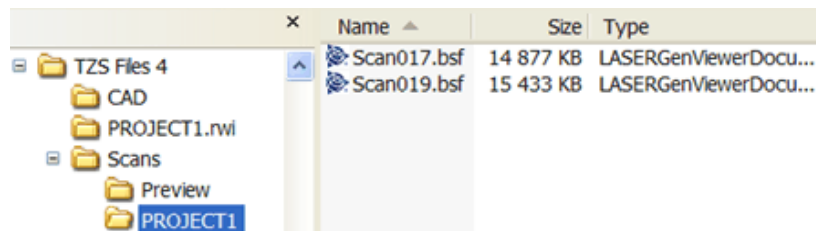
Each **TZS** (or **TZF**) format file will be converted into a unique **BSF** format file (one per file). Each **BSF** format file will have as name its related **TZS** (or **TZF**) file. All will be put under the **Scan Folder** (defined in the **Output Files** panel).

To convert a TZS Format File to a BSF Format File:

1. Check the **By Station** option. The **Volume Properties** panel becomes dimmed.
2. Click **Convert**.
 - The conversion is done from the first **TZS** (or **TZF**) format file to the last. For each **TZS** (or **TZF**) format file, there are two stages.
 - You can abort the conversion operation by pressing **Esc**, only when the first stage is in progress. In this case, the following dialog appears.




- The **Trimble Scan Files to BSF Conversion Using Registered Stations** dialog closes of its own (before the end of the conversion).



Folder

The **Default Folder** is the folder where the **TZS** (or **TZF**) files are located. The **Scan Folder** is the folder where **BSF** files (once converted) will be put. By default, it has the same name as the project created within **RealWorks**.

To Define the Output Files Folder:

1. In the **Output Files** panel, click on the **Default Folder**  button. The **Save In Folder** dialog opens.
2. Locate a drive/folder to store the **BSF** format files in the **Save In** field.
3. Select and open the folder by double-clicking.
4. Click **Save**. The **Save In Folder** dialog closes.

Scan Folder Name

The **Scan Folder** is the folder where the **BSF** format file will be put under. By default, its name is **ProjectX** where **X** is its order.

To Define the Scan Folder Name:

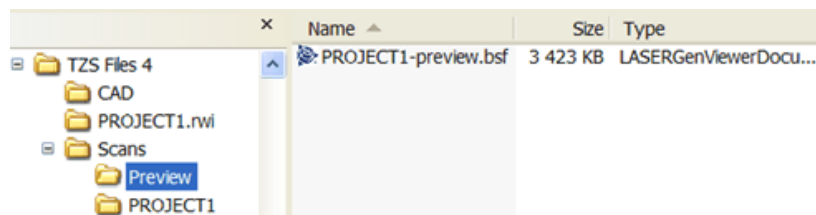
1. Keep the default name which is the project name.
2. Or enter a new name in the **Scan Folder Name** field.

Preview BSF Format File

You can generate a preview of the whole **TZS** (or **TZF**) format files. This preview is a sample* and will have the same name as the **Scan Folder** and will be put under a folder named **Preview**.

To Generate a Preview BSF Format File:

- In the **Output Files** panel, check the **Generate a Preview BSF File** option.

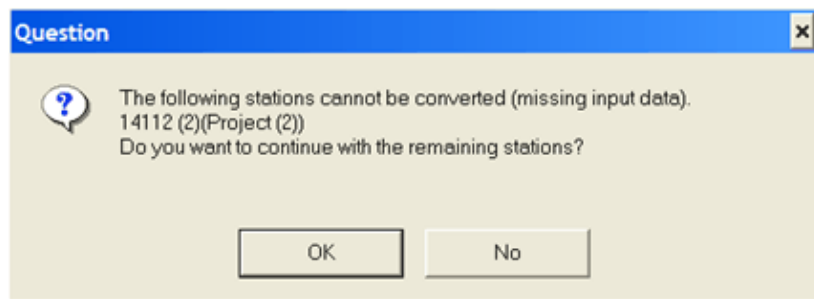


Note: (*) The **Sampling** parameter is set by default to 100 mm.

Converting

Two folders (**CAD** and **Scans**) are created and rooted under the **Default Folder** (the one defined in the **Output Files** panel) as well as a file named **Scan_Centers** (with **BWS** as extension). This file is a list of all **TZS** (or **TZF**) file centers. The preview and the **BSF** (respectively in **Preview** and **Scan Folder**) are all put under the **Scans** folder.

Note: You cannot convert a station (or a set of stations or a project) that is not a **Trimble Scan File** with **TZS** (or **TZF**) extension. If you attempt to so, the dialog below appears.



Converting to E57/PTX/PTS

PTX and **PTS** are extensions for laser scanning files. Both are **ASCII** based.

To Convert to E57/PTX/PTS Format File:



1. Select a station (or a set of stations or a project)* from the **Project Tree**.
2. From the **File** menu, select **Advanced Export / Convert to E57/PTX/PTS**.
 - If a station (or a set of stations) has been selected as input, the **Convert to E57/PTX/PTS** dialog opens.
 - If a project has been selected as input, a dialog opens and asks you if you want to process all stations of the selected project.
 - Click **Yes**. The **Convert to E57/PTX/PTS** dialog opens.
3. If required, apply a Sampling by Step to the data to export.
4. If required, specify a Folder where to store the file(s).
5. Choose an option between **E57**, **PTX** and **PTS**.
6. Click **Convert**.

Note: The stations can be empty.

Sampling by Step

In the **Sampling by Step** method, one point will be taken into account at each defined **Step** vertically and horizontally in the **2D Image Data**.


To Sample by Step:

1. In the **Filtering (Optional)** field, check the **Sampling by Step** option. The **Step (in Pixels)** field becomes enabled.
2. Enter a value in the **Step (In Pixels)** field.
3. Or use the **Up**  (or **Down** ) button to choose a value.

Folder

The **Default Folder** is the folder where the **TZF** format files are.

To Define the Output Files Folder:

1. In the **Output** panel, click on the **Default Folder**  button. The **Save In Folder** dialog opens.
2. Locate a drive/folder to store the **PTS** (or **PTX**) format files in the **Save In** field.
3. Select the folder and click **Save**. The **Save In Folder** dialog closes.

E57

The **E57** format is a file format specified by the ASTM (American Society for Testing and Materials), an international standards organization. It is compact and vendor-neutral. It was developed for storing data (Point Clouds, images and metadata) produced by 3D imaging systems such as laser scanners. Such format enables data interoperability among 3D imaging hardware and software systems and is not dependent on proprietary formats for storing and exchanging data.

To Convert to E57 Format:

- In the **Output Files** panel, check the **E57** option.

Notes:

- An **E57** format file is created for each **TZF scan**. The **E57** format file is named according to the name of the **TZF** format file.
- Each **TZF Scan** is exported to the **E57** format as a regular grid point sets.

PTS

A **PTS** format file contains a set of lines information. The **Number of Points** in the file is at the first line followed by as many lines as there are points. Each line corresponds to a **Point** with the **X,Y,Z** coordinates, intensity information (from -2047 to +2048).

To Convert to PTS Format:

- In the **Output Files** panel, check the **PTS** option.

Notes:

- The **PTS** option applies the station registrations to point coordinates and then writes them.
- The unit of measurement is in **Meter**.
- A **PTS** format file is created for each **TZF scan**. The **PTS** format file is named according to the name of the **TZF** format file.

PTX

A **PTX** format file also contains a set of lines information. The **Number of Columns** and the **Number of Points per Column** in the file are respectively at the first and the second line followed by a series of sets of lines. The **Number of Lines** in a set corresponds to the **Number of Points**. The first set - which comes after the two first lines of the file - defines the first column, the next set the second column, and so on. Each line corresponds to a **Point** with the following information: x,y,z,i (from 0.0 to 1.0). If there is no scanning data (because of sky e.g.), the **Point** on the column still exists but contains zero.

To Convert to PTX Format:

- In the **Output Files** panel, check the **PTX** option.

Notes:

- The **PTX** option writes the station registration information in the file header and keep point coordinates unchanged.
- The unit of measurement is set to **Meter**.
- A **PTX** format file is created for each **TZF scan**. The **PTX** format file is named according to the name of the **TZF** format file.

Inspection Maps

Each inspection map is a plane which has two directions (**Vertical** and **Horizontal**) whatever the shape (**Plane**, **Tunnel** or **Cylinder**) it has. The two directions are illustrated by the Red and Green axes. You can export an inspection map to **Tiff** format or slices done on the comparison surfaces to **DXF** (or **DWG**) format. Slices can be vertical (parallel to the **Green** axis) or horizontal (parallel to the **Red** axis). **Tiff** is the acronym for Tagged Image File Format. It is one of the most popular and flexible current public domain raster file formats. The **DXF** file format is an **ASCII** file format which describes CAD data defined by AutoDesk. This file format facilitates the exchange of CAD data between two different programs. The **DWG** file format is the binary file format from AutoCAD and AutoCAD LT.

Exporting an Inspection Map

Exporting an inspection map from **RealWorks** involves saving it in the **Tiff** format. With the **Tiff** image, is created a file of the same name and with the **TXT** extension. This file contains four corners: **Top Left**, **Top Right**, **Bottom Left** and **Bottom Right**. These corners are useful for situating an inspection map in 3D.

To Export an Inspection Map:

1. Select an inspection map from the **Project Tree**.
2. From the **File** menu, select **Advanced Exports / Export Inspection Map**.
The **Export Inspection Map** dialog opens.
3. Locate a drive/folder to store the file in the **Look In** field.
4. Enter a name in the **File Name** field.
5. Click **Save**.

Tip: You can select an inspection map from the **Project Tree** (or from the **3D View**) and select **Export Inspection Map** from the pop-up menu.

Notes:

- You cannot export an inspection map that is not already created in the **RealWorks** database.
- If a set of inspection maps has been selected, the export is done one by one from the first to the last. The user only needs to click **Save** for each inspection map.

Exporting Horizontal Slices

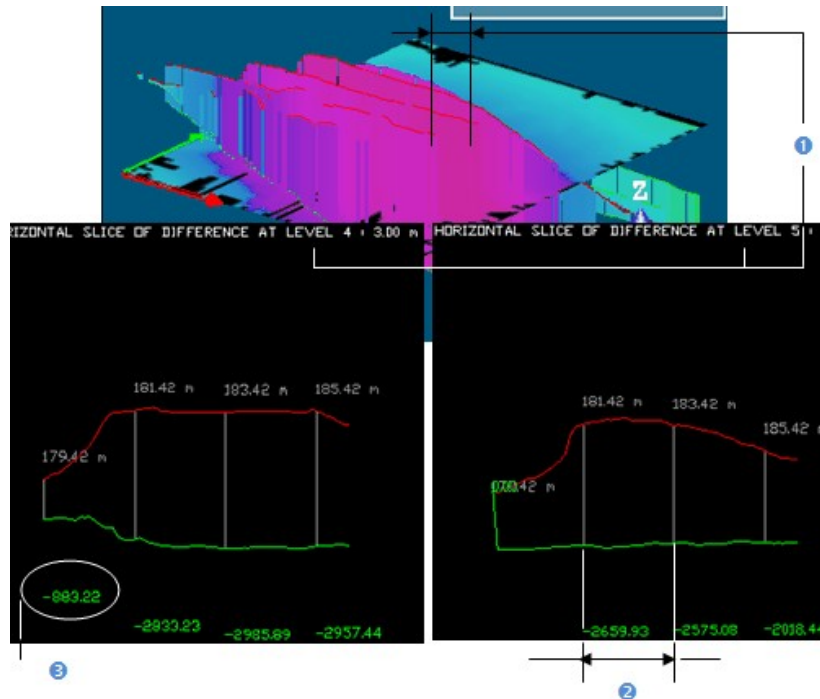
Horizontal slices are obtained by slicing an inspection map along the **Red** axis (of its own frame) with a constant interval. These horizontal slices are those obtained when multi-slicing in **Sections & Shifts** in the **Inspection Map Analyzer** tool. A horizontal slice is a pair of red and green sections. Red sections result from slicing over the **Reference Surface** while the ones in green result from the **Comparison Surface**. When exporting to **DXF** (or **DWG**), the slicing order is preserved; each vertical slice has its own layer (also called **Level**). For a given horizontal slice, the difference of elevations between the red and green sections is displayed with a value (in green) in the current unit of measurement. This difference can be displayed along the slice with a constant interval (value in gray) that the user has to define.

To Export Horizontal Slices:

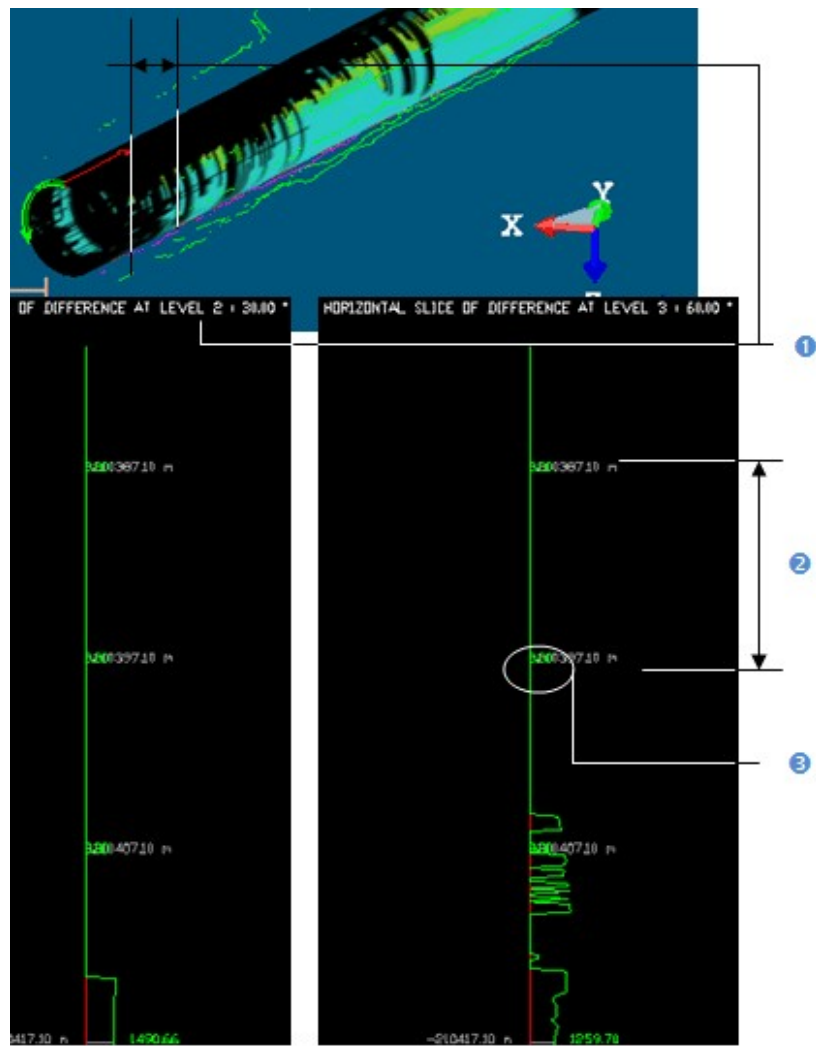
1. Select an inspection map from the **Project Tree**.
2. From the **File** menu, select **Advanced Exports / Export Inspection Map Horizontal Slices**. The **Export Horizontal Slices of Selection** dialog opens.
3. Click on the **Type of File** pull down arrow.
4. Choose from a version of **DWG** (or **DXF**) format.
5. Locate a drive/folder to store the file in the **Look In** field.
6. Enter a name in the **File Name** field.
7. Click **Save**. The **Horizontal Slices Export Parameters** dialog opens.
 - **Horizontal Interval:** This interval is a constant step used for horizontally slicing the inspection map. It is the same as the **Interval** used in **Section & Shifts** when multi-slicing. This constant step is a distance value when the inspection map is a **Plane** (or **Tunnel**). It is an angle* when the inspection map is a **Cylinder**.
 - **Vertical Interval:** This interval is a constant step used for displaying the difference in elevations between the red and green sections. This constant step is a distance value.
 - **Amplification Factor:** This factor is used for magnifying the differences in elevations when they are too small for viewing.
 - **Reference Surface Title:** This option enables to define a name for the **Reference Surface**.
 - **Comparison Surface Title:** This option enables to define a name for the **Comparison Surface**.
 - **Unit:** This option enables to choose a unit of measurement.
8. In the **Horizontal Interval** field, enter a distance value (or angular value) if the inspection map is a **Plane/Tunnel** (or **Cylinder**).
9. In the **Vertical Interval** field, enter a distance value.
10. Give a value in the **Amplification Factor** field.
11. Enter a name in the **Reference Surface Title** field.
12. Enter a name in the **Comparison Surface Title** field.

13. Choose a unit of measurement from the **Unit** drop-down list.
14. Click **Export**. The **Vertical Slices Export Parameters** dialog closes.

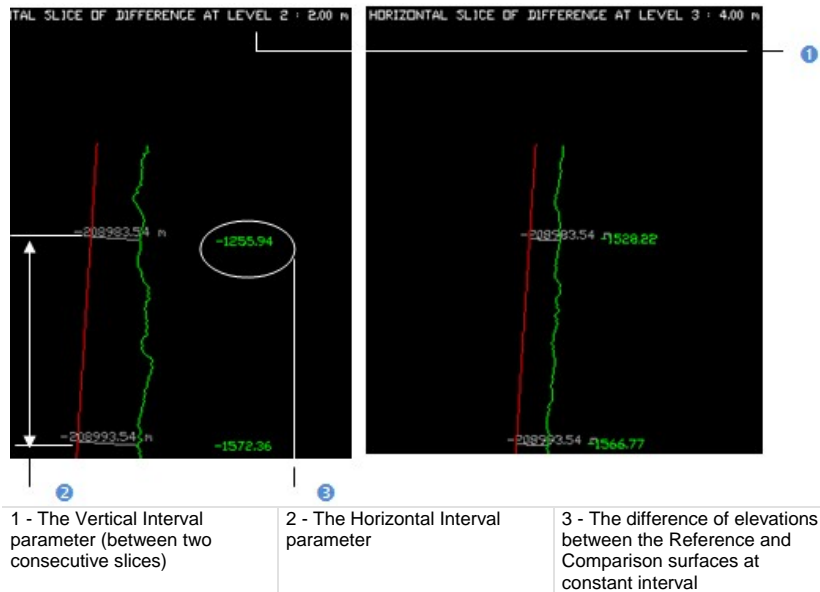
If the inspection map is a **Plane**; a vertical slice has the shape shown below:



If the inspection map is a **Cylinder**; a vertical slice has the shape shown below:



If the inspection map is a **Tunnel**; a vertical slice has the shape shown below:



Notes:

- The unit of measurement for a distance value is by-default set to **Meter**; you do not have to enter “m” and you can change it when necessary (refer to the **Preferences** options for more details).
- The unit of measurement for an angular value is by-default set to **Degree**; you do not have to enter “°” and you can change it when necessary (refer to the **Preferences** options for more details).
- * In **Sections & Shifts** in the **Inspection Map Analyzer** tool, this constant step can be a distance or an angle while in **Export Inspection Map Vertical Slices** it can only be a distance.
- ** In **Sections & Shifts** in the **Inspection Map Analyzer** tool, you cannot slice an inspection map of tunnel shape horizontally while in **Export Inspection Map Vertical Slices** you can do it.

Exporting Vertical Slices

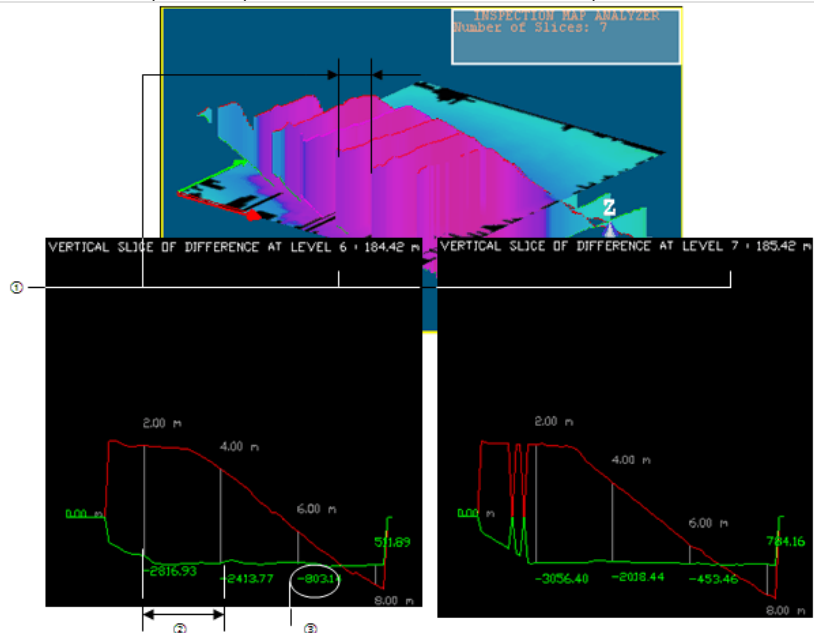
Vertical slices are obtained by slicing an inspection map along the **Green** axis (of its own frame) with a constant interval. These vertical slices are those obtained when multi-slicing in **Sections & Shifts** in the **Inspection Map Analyzer** tool. A vertical slice is a pair of red and green sections. Red sections result from slicing over the **Reference Surface** while the ones in green result from the **Comparison Surface**. When exporting to **DXF** (or **DWG**), the slicing order is preserved; each vertical slice has its own layer (also called **Level**). For a given vertical slice, the difference of elevations between the red and green sections is displayed with a value (in green) in the current unit of measurement. This difference can be displayed along the slice with a constant interval (value in gray) that the user has to define.

To Export Vertical Slices:

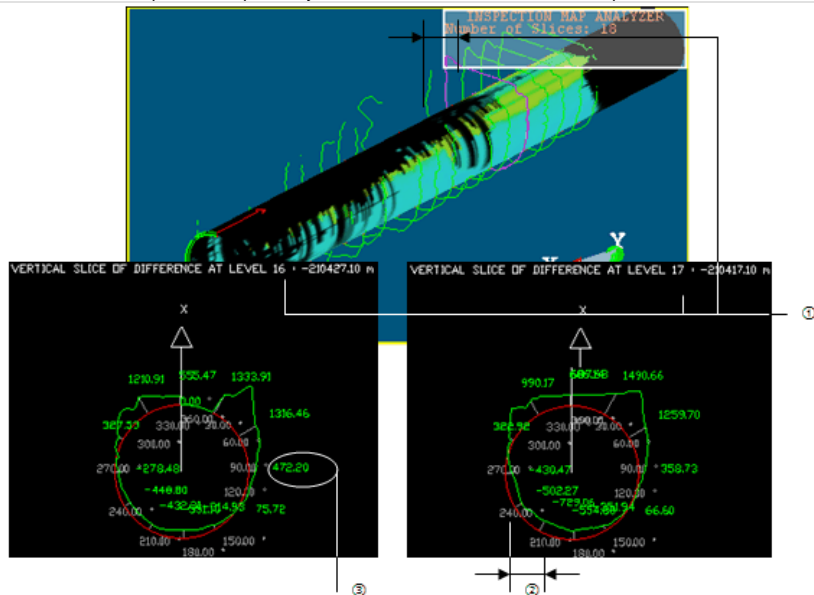
1. Select an inspection map from the **Project Tree**.
2. From the **File** menu, select **Advanced Exports / Export Inspection Map Vertical Slices**. The **Export Vertical Slices of Selection** dialog opens.
3. Click on the **Type of File** pull down arrow.
4. Choose between **DWG** and **DXF** format.
5. Locate a drive/folder to store the file in the **Look In** field.
6. Enter a name in the **File Name** field.
7. Click **Save**. The **Vertical Slices Export Parameters** dialog opens.
 - **Horizontal Interval:** This interval is a constant step used for displaying the difference in elevations between red and green sections. This constant step is a distance value when the inspection map is a **Plane** (or **Tunnel**). It is an angle when the inspection map is a **Cylinder**.
 - **Vertical Interval:** This interval is a constant step used for vertically slicing the inspection map. It is the same as the **Interval** used in **Section & Shifts** when multi-slicing. The constant step is distance value.
 - **Amplification Factor:** This factor is used for magnifying the differences in elevations when they are too small for viewing.
 - **Reference Surface Title:** This option enables to define a name for the **Reference Surface**.
 - **Comparison Surface Title:** This option enables to define a name for the **Comparison Surface**.
 - **Unit:** This option enables to choose a unit of measurement.
8. In the **Horizontal Interval** field, enter a value (distance or angle).
9. In the **Vertical Interval** field, enter a distance value.
10. Give a value in the **Amplification Factor** field.
11. Enter a name in the **Reference Surface Title** field.
12. Enter a name in the **Comparison Surface Title** field.
13. Choose a unit of measurement from the **Unit** drop-down list.

14. Click **Export**. The **Vertical Slices Export Parameters** dialog closes.

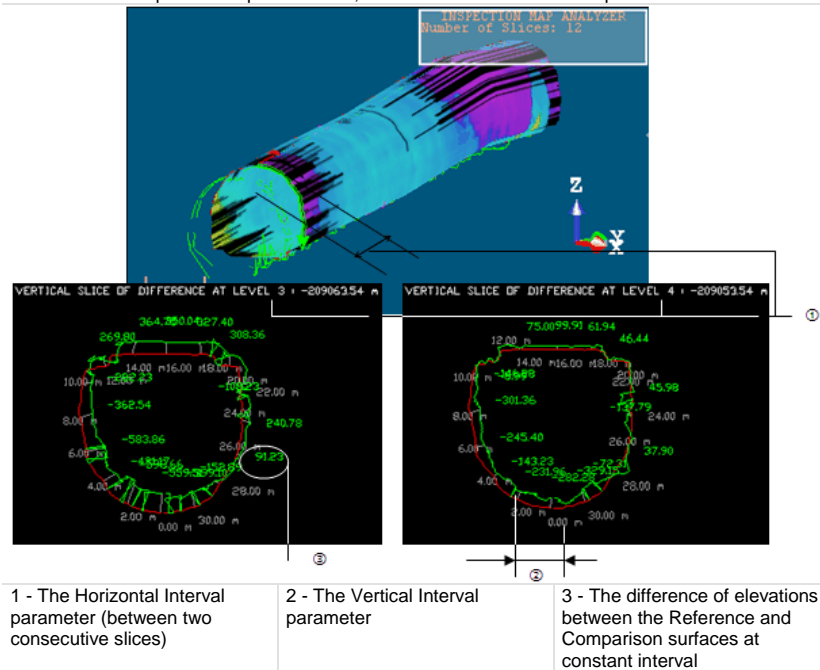
If the selected inspection map is a Plane; a horizontal slice has the shape shown below:



If the selected inspection map is a Cylinder; a horizontal slice has the shape shown below:



If the selected inspection map is a Tunnel; a horizontal slice has the shape shown below**:



Notes:

- The unit of measurement for a distance value is set by default to **Meter**; you do not have to enter "m" and you can change it when necessary (refer to the **Preferences** options for more details).
- The unit of measurement for an angular value is set by default to **Degree**; you do not have to enter "°" and you can change it when necessary (refer to the **Preferences** options for more details).

CHAPTER 15

Storage Tank Application

The **Storage Tank** module is a **RealWorks** application designed for computing the **Volume** of vertically positioned storage tanks. The **Storage Tank** menu is present in all **RealWorks** products except in the **Viewer**, **Base** and **Plant**.

Tools (in)	Viewer	Base	Advanced Advanced - Modeler Advanced - Plant
Tank Calibration Tool			✓
Tank Calibration Check			✓

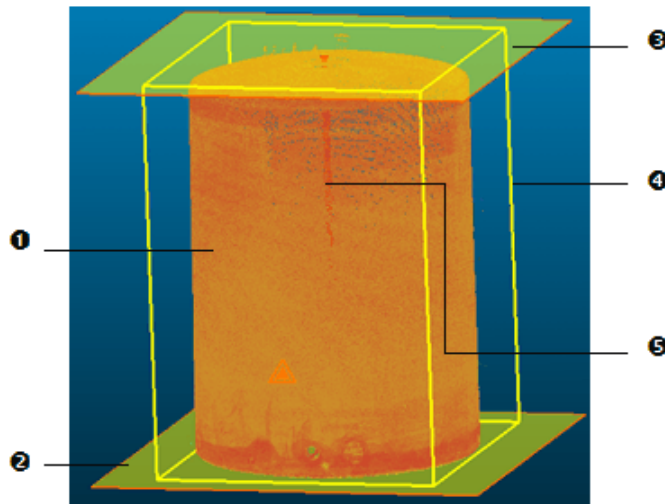
Tank Calibration Tool

The **Tank Calibration Tool** computes data for the calculation of volumes of vertically positioned storage tanks.

Activating

To Activate the Tool:

1. Select an object (**Point cloud** or **Fitted Mesh**) from the **Project Tree**.
2. From the **Storage Tank** menu, select **Tank Calibration Tool**. The **Tank Calibration** dialog opens.



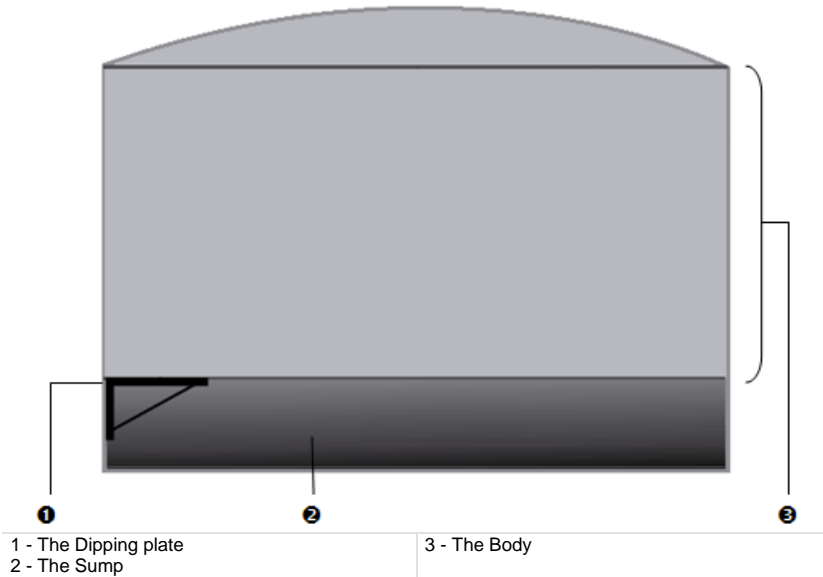
1 - A set of points
2 - The bottom plane

3 - The top plane
4 - The bounding box
5 - The cross section center line

This dialog opens as the third tab of the **WorkSpace** window and is composed of several parts. If the input is a **Point Cloud**, you can clean it by removing parasite points (or reduce its size by simplifying it). If the input is a **Fitted Mesh**, only a **Point Cloud** is displayed. By default, two planes and a scale appear.

Defining a Dipping Plate

A **Dipping Plate** is a level plate which defines the separation between the **Sump** and the **Body** part of a **Vertical Tank**. Its **Height** can be either picked in the **3D View** or a value can be entered, if known by the user.



Entering a Dipping Plate Height

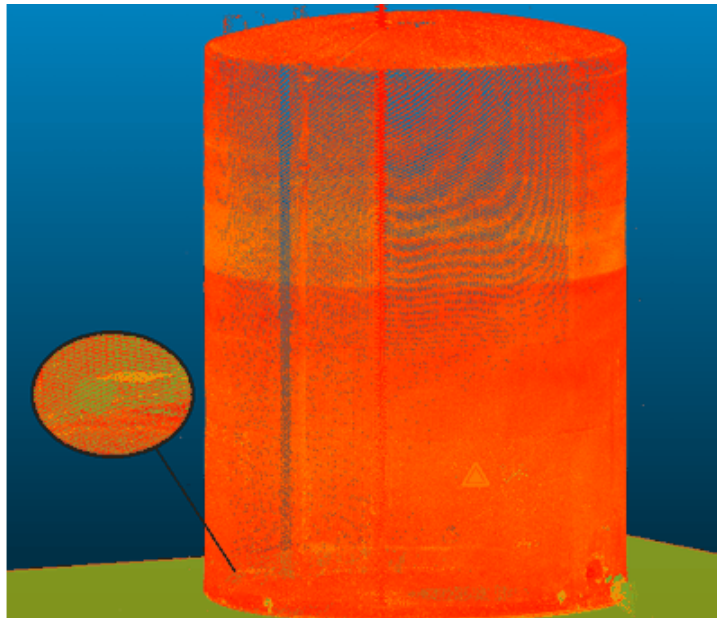
To Enter a Dipping Plate Height:

- Enter a height value in the **Dipping Plate Height** field.

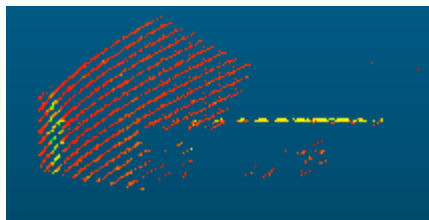
Picking a Dipping Plate Height




To Pick a Dipping Plate Height:

1. First, navigate within the set of points to visually locate the **Dipping Plate**.
2. Then, click the **Segmentation** icon in the **Tank Calibration** dialog. The **Segmentation Tool** toolbar opens.



3. Isolate the **Dipping Plate** from the whole set of points by segmenting.
4. Bring the view to **Front**.



5. Close the **Segmentation Tool** toolbar by clicking **Close Tool** . The two initial planes and the cross section center line appear again.
6. Click the **Pick Dipping Plate**  icon.
7. Pick a point on the **Dipping Plate** in the **3D View**. The **Bottom Plane** will be then set to that point.
8. If required, reload the initial points that make up the tank by clicking the **Reload Points**  icon (within the **Tank Calibration Tool**).

Defining Body Parameters

A **Body** is the part of a **Vertical Tank** above its **Dipping Plate**, from which **Sections** will be computed. Its **Maximum Height** from the **Dipping Plate** can be either picked in the **3D View** or the value can be entered if known by the user.

Picking a Maximum Height Level

To Pick a Maximum Height Level:

1. Click on the **Pick Maximum Level** icon.
2. Bring the view to **Front**.
3. Pick a point on the set of points in the **3D View**.

Entering a Height Value

To Enter a Height Value:

- Enter a height value in the **Maximum Level Plate** field.

Section Interval

An **Interval** is the distance between two consecutive **Sections**. It needs to be at least of 5 mm.

To Define a Section Interval:

1. Enter a distance value in the **Interval** field.
2. Or use the **Up**  (or **Down** ) button to set a value.

Defining Sump Volume Calculation Settings

A **Sump** is the part of a **Vertical Tank** below its **Dipping Plate**, from which a **Volume** will be computed. The computation is based on a 2D grid projection. The default projection plane is the plane passing through the **Dipping Plate**. The projection's **Resolution** is square - the same in both of the default projection plane directions (**Length** and **Width**). It needs to be at least of 10 mm.

To Define the Sump Volume Calculation Settings:

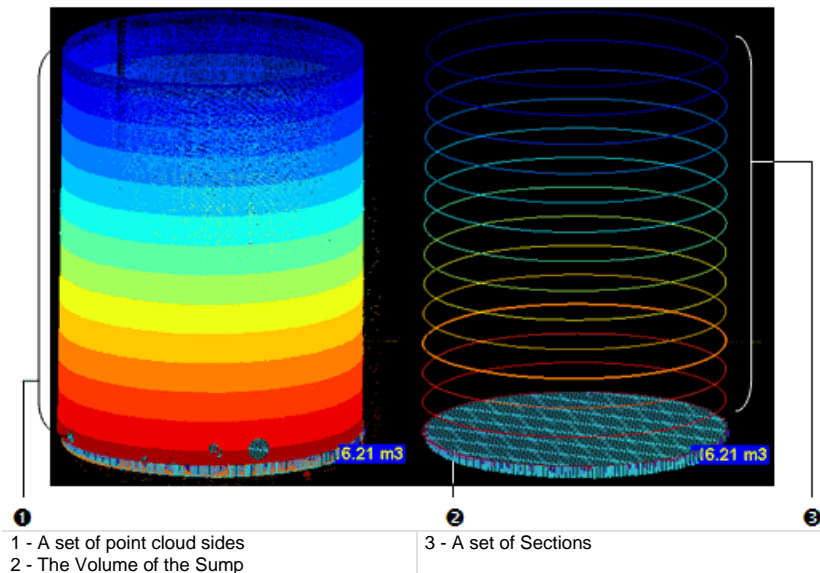
1. Enter a value in the **Resolution** field.
2. Or use the **Up**  and **Down**  buttons to select a value.

Generating Results

You can now generate and visualize the **Sections** and the **Volume** of the **Sump** obtained from the set of points (before creating them in the database).

To Generate Results:

1. Click **Preview**. In the **Tank Calibration** dialog, the **Display Sump** and **Display Sections** options become enabled. By default, the two **Geometry** options are checked. In the **3D View**, the **Sump's Volume** is represented by a graph of vertical color lines and the **Sections** are represented by a set of closed and fitted (with points) **Polylines**. The information box at the top right corner of the **3D View** displays in text the **Sump Volume**, **Body Volume** and **Full Volume**.
2. If required, check the **Cloud** option to display points in the **3D View**.
3. If required, uncheck the **Geometry** option to hide the **Sump's Volume** (or **Sections**) in the **3D View**.
4. Perform visual checks on the **Sump Volume** data to ensure that the entire area has been taken into account for the volume calculation. If you detect "holes" in the **Sump Volume** display, choose a different resolution setting (see Defining Sump volume calculation settings... above). Reiterate this step until you achieve a satisfactory result.



Exporting

The results can be exported into a **TXT** format file. There are two columns of information inside the file. The first column contains all **Section Heights** (above the **Dipping Plate**). The second column lists the **Area** information at each **Section** level. The first line displays the **Sump** value.

To Export the Tank Calibration Results:

1. Click **Export**. The **Save As** dialog opens.
2. Enter a name in the **File Name** field.
3. Find a location in your disk in the **Look In** field.
4. Click **Save**. The **Save As** dialog closes.

Note: If the **Storage Tank Application** option has been chosen in the **RealWorks** installation, two files are installed in X:\Program Files\Trimble\RealWorks 8.0\Tables\Tank. The **Excel** sheet is a sample template that allows the importing of cross section data (from the above mentioned **TXT** file) generated by the **Tank Calibration Tool**. Formulas allow the user to apply compensations and to then create capacity tables. The **Word** format file contains detailed instructions for importing and processing the cross section data.

Warning: You must export the results before closing the **Tank Calibration Tool**, otherwise they will be lost.

Creating

Once you are satisfied with the results, you can create the cross sections and sump volume in the database. A group named **Tank-Interval** "Interval value" is created and rooted under the **Models Tree**. All computed **Sections** and **Volume** are put under that group. A **Section** is a **Fitted Polyline** and is named **Section-TankX-Alt** "Altitude Value" where X is its order in three or more digits. The **Volume** is named: **Sump-Dipping Plate** "Height value". You can create as many sets of **Sections** and **Volume** as required without leaving the tool. If you decide to leave this tool without saving the results, a message appears and prompts you to confirm, undo or cancel the action you are going to perform.

To Create:

1. Click **Create**.
2. Click **Close**.

Tip: You can leave the **Tank Calibration Tool** by pressing **Esc** or by selecting **Close** from the pop-up menu.

Tank Calibration Check

The **Tank Calibration Check** allows the user to first check, and then if required, to modify the **Sections** previously extracted with the **Tank Calibration Tool**.

Activating

To Activate the Tool:

- In the **Tank Calibration** dialog, click **Check**. The **Tank Calibration Check** dialog appears in place of the **Tank Calibration** dialog. The first **Section** with fitted points is displayed in the **3D View** with a **2D Grid** superimposed (if not previously hidden).

Notes:

- You can hide the **2D Grid** or change its size by selecting its related command from the pop-up menu or from **3D View** menu bar.
- The user needs to first generate a preview of **Sections** and **Volume** from the set of points; otherwise the **Check** button remains dimmed.

Filtering Sections

This step is optional, though recommended if you expect, or visually detect, significant differences from one cross section to the next. It allows the user to filter by comparing cross sections from one to the next. The filtering setting called **Tolerance** corresponds to a degree of change from one cross section to the next at a percentage rate ranging from 0% to 10%. The **Sections** for which the difference (in percent) is higher than the defined rate are considered as potentially defective, and can then be edited.

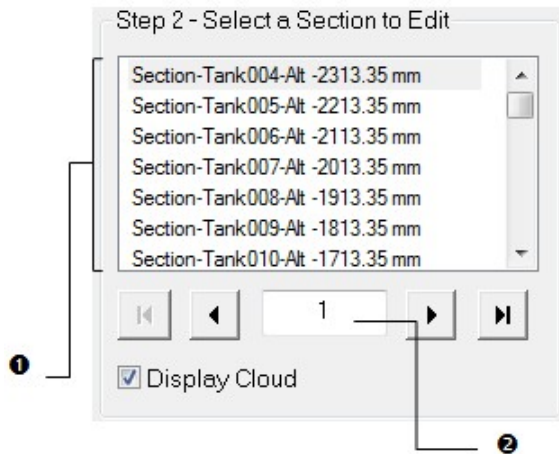
To Filter Sections:

1. Check the **Filter Sections** options. The **Tolerance** slider and its field become enabled.
2. Move the **Filter Sections** slider from **Left** to **Right** to set a value.
3. Or scroll the mouse wheel
4. Or enter a rate in the field.

Note: The rate increments in steps of 0.2%.

Displaying Sections

If the **Filter Section** step has been skipped, all extracted **Sections** (from the **Tank Calibration Tool**) are listed in **Step 2**. If the **Filter Section** step has been performed, only extracted **Sections** that are considered as out of tolerance are listed here. By default, the first **Section** from the list is the selected one and is displayed with its associated points in the **3D View**.



1 - A list of Sections

2 - The order of the Sections in that list

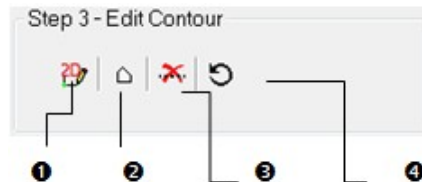
If the active **Section** is other than the first **Section**, you can use the **Up** and **Down** keys of your keyboard (or the **Display Previous Contour** and **Display Next Contour** buttons in **Step 2** of the **Tank Calibration Check** dialog) to display the next and the previous **Section** in the **3D View**.

Clicking the **Display First Contour** and **Display Last Contour** buttons will set the first and last **Section** as active (selected). You can key in a **Section's** order in **Step 2** to select it. Validate by pressing the **Enter** key.

Tip: You can hide the slice of points that is associated with the selected **Section** by un-checking the **Display Cloud** option.

Editing Sections

This step is dedicated to the editing of defective **Sections**. Only one **Section** can be edited at a time; the selected one. As a **Section** is a fitted **Polyline** which is composed of a set of **Segments**, you can use the **Polyline Drawing Tool** to modify it by hand by moving vertices. You can also delete a part and connect extremities with a **Segment**.







1 - Polyline Drawing Tool
2 - Standard Selection Mode


3 - Delete and Connect Extremities
4 - Reload Initial Section


Tip: All features can be selected from the pop-up menu.

Modifying

To Manually Modify a Section:

1. Click the **Polyline Drawing Tool**  icon. The **Drawing Tool** and **Picking Parameters** (in 2D constraint mode) toolbars appear.
2. Place the cursor over the **Section**. A solid square  appears if you are on a node,  if you are on a middle node and  if you are on a middle node to insert.
3. Drag the node to a position. The selected node turns to yellow.
4. Drop the node to that position.
5. Click **Validate**.


Note: If required, reload the initial **Section** by clicking .


Tip: The **Polyline Drawing Tool**  icon can also be selected from the pop-up menu.

Selecting

A **Section** is a fitted **Polyline** which is composed of **Segments**.

To Select Items:



1. Click the **Standard Selection Mode**  icon.
2. Draw a polygonal fence.
3. Double-click to end and validate the polygonal fence.

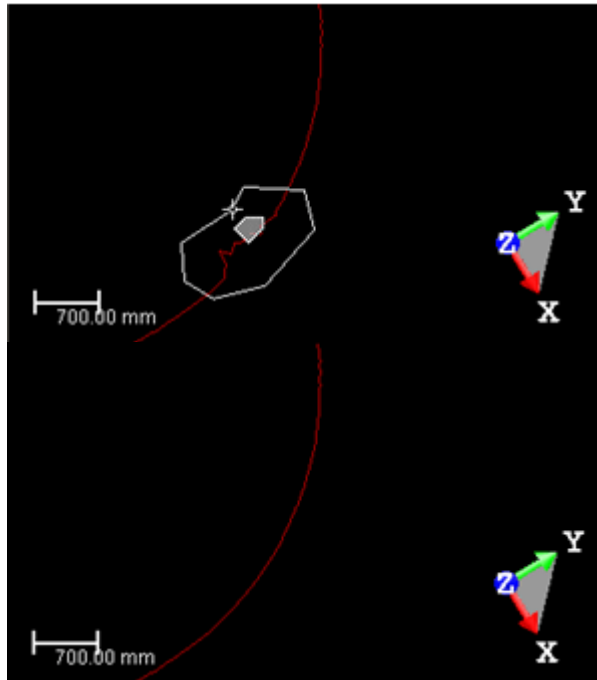
Note: To undo the previously drawn fence and start a new one, select the **Standard Selection Mode**  icon again.


Tip: The **Standard Selection Mode**  icon can also be selected from the pop-up menu.

Deleting and Connecting Extremities

To Delete and Connect Extremities:

1. Perform a selection as described previously.
2. Click the **Delete and Connect Extremities**  icon. **Segments** inside the previous selection are deleted and the extremities are connected together with a **Segment**.
3. If required, reload the initial **Section** by clicking .



Tip: The **Reload Initial Section**  and **Delete and Connection Extremities**  icons can be selected from the pop-up menu.

Applying

If modifications have been applied to the **Sections**, click **Apply**. The **Tank Calibration Check** dialog will save the changes and then close. Otherwise, click **Close**. This will close the **Tank Calibration Check** dialog too.

Index

2

- 2D Grid • 52
- 2D Preview Image • 178
- 2D Sections • 1007
- 2D-EasyLine Tool • 482
- 2D-Polyline Inspection Tool • 667

3

- 3D Axis Tool • 978
- 3D Direction Tool • 966
- 3D Inspection Analyzer Tool • 765
- 3D Inspection Tool • 761
- 3D Picking Tool • 963
- 3D Plane Tool • 989
- 3D Point • 840, 907
- 3D Point Tool • 971
- 3D Radius Tool • 982
- 3D Secant Tool • 986
- 3D View • 50, 57
- 3D View Window • 145

A

- A Single Ortho-Image • 554
- A Single Polyline • 587
- Activating • 1089, 1097
- Activating/Deactivating the Selection Mode • 957
- Active Group • 79
- Adding • 658, 664
- Adding Keyframes • 315
- Adjusting • 614
- Adjusting Stations • 356
- Advanced Exports • 1062
- Advanced Features • 1029
- Alias/WaveFront (OBJ) Format • 1050
- Aligning a Geometry (Z-Axis) Along a 2-Point-Defined Axis • 891

- Aligning to Join to an Existing Cylinder • 905
- Aligning to Join to two Secant Cylinders of Same Radius • 790, 839
- Aligning to Join to Two Secant cylinders of Same Radius • 906
- Aligning to Join Two Existing Secant Boxes of Same Section • 814, 853, 912
- Aligning to Joint to an Existing Cylinder • 789
- All Ortho-Images • 554
- All Polylines • 588
- Along a Line • 918
- Along a Polyline • 925
- Angular Measurements • 250
- Applying • 365, 1101
- Applying a New Texture • 518
- Applying and Grouping • 367
- Applying Bounds • 960
- Applying Constraints • 892
- Applying Filters • 495
- Applying Operations • 519
- Applying the Cut Positions • 645, 649
- Applying the Matching • 616
- Applying the Result • 398
- Applying Transformation • 418
- Around an Axis • 922
- As a DGN Format File • 1027
- As a DWG Format File • 1025
- As a DXF Format File • 1026
- ASCII Files • 104
- ASCII Format • 1054
- Assigning Coordinates to Picked Points • 425
- Assigning Coordinates to Targets • 423
- AutoCAD (DWG) Format • 1058
- AutoCAD (DXF) Format • 1053, 1060
- AutoCAD Files • 112
- AutoCAD PCG format (AutoDesk Revit MEP) • 1049

- Auto-Duplicating Polylines • 593
- Auto-Extract Targets and Register • 336
- Auto-Match All • 364
- Auto-Match Station • 364
- Automatic Axis Definition • 417
- Auto-Pairing Targets • 354
- Auto-Register Using Planes (Target-Less)
 - 342
- Auto-Splitting a 3D Inspection Cloud in a Cluster of Clouds • 768

B

- Background Color • 126
- Basic Tools • 235
- Basics of Operation • 83
- Bin Folder • 130
- Black and White Flat Targets • 393
- Bottom Align all Planes • 546
- Box • 855, 867, 913
- Browsing Iso-Curves • 730
- Browsing Keyframes • 329
- Browsing Through Stations • 197
- BSF Format • 1046
- Building a Frame with Constraints • 289
- Building a Frame without Constraints • 283
- Building Polylines • 469
- Bundle Adjustment • 428
- By Interpolation • 464
- By Offset • 463
- By Station • 1073
- By Volume • 1071

C

- Calculating a Geometry • 857
- Calculating Contours • 475
- Capturing Screen (High Resolution) • 333
- Capturing the Screen • 149
- Centering on Point • 201
- Chains of Segments/Arcs • 576
- Changing • 402
- Changing a Keyframe Direction • 327
- Changing a Keyframe Position and Orientation • 320
- Changing a Size • 53
- Changing Color • 220
- Changing Dimensions • 530
- Changing Name • 222
- Changing Sub-View Configuration • 51

- Changing the 2D Section Position and Orientation • 1009
- Checking a Projection Plane • 531
- Checking a Volume to Keep • 629
- Checking for Checkout Support • 37
- Checking the Adjustment • 357
- Checking the Inspections • 685, 698
- Checking the Registration Results • 412
- Checking-out • 38
- Choosing a Geometry Type • 821
- Choosing a Method • 917
- Choosing a Section Type • 1005
- Choosing a Station • 556
- Circles • 581
- Circular Arc • 842
- Circular Selection • 264
- Circular Torus • 836, 865, 903
- Clearing Selections • 145
- Close • 148
- Close All • 148
- Closing • 170
- Closing Projects • 148
- Closing the Tool • 419
- Cloud-Based Modeler Tool • 816
- Cloud-Based Registration Tool • 399
- CMF Files • 115
- ColorBars • 205
- Colored Meshes • 733
- Coloring Points • 616
- Coloring Points Using Station Images • 1036
- Computing Cross-Sections • 650
- Cone • 864
- Cone to Cone • 938
- Cone to Plane • 938
- Configuring a Computer to Allow License Checkout • 37
- Connecting a Plane to a Series of Planes • 942
- Connecting to a Series of Entities • 941
- Connecting to Mobile Device • 87
- Connecting Two/Three Entities • 934
- Constraining Picking on a Line • 225
- Constraining Picking on a Plane • 225
- Constraining Picking on a Point • 226
- Constraining to a Pair • 614
- Constraining to Two Pairs • 615
- Contacting Trimble • 39
- Continuing in Tracking Cylinders • 1019

Continuing Polylines • 586
 Continuing Tracking • 754
 Contouring Tool • 471
 Converting • 1075
 Converting to BSF • 1069
 Converting to E57/PTX/PTS • 1075
 Converting to Meshes • 297
 Converting to the TZF Format • 94
 Copying and Pasting an Item • 216
 Creating • 206, 392, 395, 397, 1096
 Creating 3D Points • 382, 449
 Creating 3D Points From Matched Targets • 450
 Creating 3D Points From Targets • 449
 Creating a 3D Inspection Cloud • 764
 Creating a Box • 803
 Creating a Fitted Geometry • 760
 Creating a Geometry • 858
 Creating a Merged Mesh • 298
 Creating a New Group Node • 219
 Creating a Polyline • 927
 Creating a Rectangular Torus • 813
 Creating a Rectified Image • 572
 Creating a Registration Report (Target-Based) • 451
 Creating a Video • 331
 Creating an Extruded Entity • 802
 Creating an Ortho-Image • 540
 Creating Beams • 1010
 Creating Colored Meshes • 735
 Creating Contours • 480
 Creating Extracted Cylinders • 1023
 Creating Feature Sets • 666
 Creating Iso-Curves • 732
 Creating Meshes • 504
 Creating Ortho-Images • 553
 Creating Point Cloud from Topo Points • 448
 Creating Points • 447
 Creating Polylines • 598
 Creating Profile and Cross-Sections • 656
 Creating Profiles • 755
 Creating Sampled Scans • 136
 Creating the Built Frame • 295
 Creating the Extracted Cloud(s) • 769
 Creating the Profile • 748
 Creating the Segmentation Results • 267
 Creating Thumbnails • 141
 Creating Topo Points • 447

Creating/Editing Targets • 379
 Customizing a Size • 54
 Customizing the User Interface • 64
 Cutting and Pasting an Item • 214
 Cutting Plane Tool • 455
 Cylinder • 827, 863, 897
 Cylinder Shape • 695
 Cylinder to Circular Torus • 936
 Cylinder to Cone • 937
 Cylinder to Cylinder • 935
 Cylinder to Sphere • 935
 Cylinder-Based Projection • 500, 674, 676, 689

D

Data Folder • 131
 Data Size Reduction • 129
 Database • 56
 Define a 3D Plane • 1007
 Define a Segment • 798
 Define Horizontal Axis by Picking Two Points • 416
 Defining • 642, 660
 Defining a 3D Axis • 924
 Defining a 3D Direction • 921
 Defining a 3D Direction Using Precise Methods • 967
 Defining a 3D Direction Using Visual Methods • 969
 Defining a 3D Plane • 574
 Defining a 3D Plane (in all Navigation Modes (Examiner, Walkthrough and Station-Based)) • 776
 Defining a 3D Plane in all Navigation Modes • 991
 Defining a 3D plane in the Examiner/Walkthrough/Station-Based Mode • 558
 Defining a 3D Plane in the Station-Based Navigation Mode • 559, 778, 993
 Defining a 3D Point • 791
 Defining a Bounding Polyline • 947
 Defining a Circular Torus • 788
 Defining a Color Range • 722
 Defining a Cutting Plane • 740
 Defining a Cylinder • 784
 Defining a Dipping Plate • 1090
 Defining a Duration • 330

- Defining a Grid Resolution • 626
- Defining a Length • 1009
- Defining a Navigation Path • 308
- Defining a Path • 647
- Defining a Plane • 618, 775
- Defining a Polyline • 541, 847
- Defining a Profile • 742
- Defining a Projection • 674, 688
- Defining a Projection Plane • 522, 557
- Defining a Regular Cone • 786
- Defining a Set of Points on the Cloud Data
 - 819, 1003, 1014
- Defining a Slice • 465
- Defining a Slider Direction • 710
- Defining a Slider Position • 711
- Defining a Sphere • 782
- Defining a Width • 644
- Defining a Zone of Interest • 534, 544, 565
- Defining an Elevation Range • 472
- Defining Body Parameters • 1092
- Defining Horizontal Face By Picking One 3D Point, Then Four Screen Points (Two Horizontal Directions and Depth) • 809
- Defining Image Parameters • 548
- Defining Parameters • 928
- Defining Principal Contours • 478
- Defining Principal Iso-Curves • 731
- Defining Sump Volume Calculation Settings • 1093
- Defining the Horizontal • 527
- Defining the Normal Z Direction • 856
- Defining the Orientation of a Plane • 457
- Defining the Position of a Plane • 462
- Defining the Vector X Direction • 856
- Defining Vertical Face By Picking One 3D Point, Then Four Screen Points (Horizontal Direction, Vertical Direction and Depth) • 811
- Defining Video Parameters • 330
- Delete Selection • 493
- Delete Selection Filling Holes • 494
- Deleting • 207, 609, 659, 660, 662
- Deleting a Bounding Polyline • 957
- Deleting a Keyframe • 319
- Deleting a Node • 954
- Deleting all Keyframes • 319
- Deleting an Element from a Mesh • 513
- Deleting an Item • 218
- Deleting and Connecting Extremities • 1101
- Deleting Displayed Cloud • 820, 1004, 1015
- Deleting Extracted Cylinders • 1020
- Deleting Geometries • 1037
- Deleting Items • 492
- Deleting Nodes • 584
- Deleting Polylines • 587
- Deleting Sections • 753
- Deleting Targets • 382
- Density • 549
- Depth • 550
- Detaching • 36
- Detecting Edges • 277
- Determining a Resolution • 680, 691
- Discontinuity-Based Sampling • 275
- Displacement Modes • 187
- Displacing the Moving Cloud • 403
- Display • 56
- Displaying • 64, 67, 161, 163, 166, 171, 172, 173, 176, 177
- Displaying Contours • 479
- Displaying Cross-Sections • 655
- Displaying Sections • 1098
- Distance Measurements • 239
- Docking • 66
- Dragging and Dropping an Item • 213
- Drawing • 535, 543, 566, 640, 649
- Drawing a 2D Section • 1008
- Drawing a Circle • 501, 923
- Drawing a Circular Bounding Polyline • 952
- Drawing a Cylinder • 501
- Drawing a Fence • 263
- Drawing a Line by Defining Two Points • 919
- Drawing a Polygonal Bounding Polyline • 948
- Drawing a Rectangular Bounding Polyline • 950
- Drawing and Creating a Path • 325
- Drawing Circle • 677
- Drawing Cylinder • 677
- Drawing in a Plane Parallel to the Screen View • 847
- Drawing in a User-Defined Plane • 848
- Drawing Polylines • 575, 706
- Duplicating Items • 932
- Duplicator Tool • 917

E

E57 • 1077
 E57 Format • 1047
 EasyPipe Tool • 1011
 EasyProfile Tool • 749
 Editing • 209, 661, 683, 697
 Editing a Bounding Polyline • 953
 Editing a Mesh • 511
 Editing a Volume • 630
 Editing Cross-Sections • 656
 Editing Data • 211
 Editing Keyframes • 318
 Editing Parameters • 462, 502, 534, 624, 678, 781, 783, 785, 787, 797, 801, 812, 815, 977, 981, 982
 Editing Planes • 547
 Editing Polylines • 488, 583
 Editing Sections • 1099
 Editing Targets • 356
 Editing the Selected Plane Bounds • 946
 Editing the Target Height • 383
 Enable Extract • 130
 Enhancing With Break Lines Using Polyline(s) • 515
 Entering a Dipping Plate Height • 1090
 Entering a Height Value • 1092
 Entering a Trimble Power, Process and Plant License • 33
 Equalizing Image Color • 1034
 Equalizing Point Cloud Color • 1033
 Equalizing Point Cloud Luminance • 1032
 Examiner • 188
 Expanding and Shrinking the Project Tree • 155
 Exploring Data • 153
 Exploring in the 3D View • 158
 Exploring in the Images Tree • 159
 Exporting • 208, 659, 1095
 Exporting an Inspection Map • 1079
 Exporting Data • 1039
 Exporting Horizontal Slices • 1080
 Exporting Pipe Center Lines • 1024
 Exporting Selection • 1041
 Exporting to RMX Files • 434
 Exporting to TZF Files • 433
 Exporting TZF/TZS Luminance Images • 1068

Exporting Vertical Slices • 1084
 Extending (or Stretching) • 601
 Extracted Targets • 179
 Extracting • 389
 Extracting a Couple of Section and Shift • 713
 Extracting a Set of Sections and Shifts • 714
 Extracting an Initial Cylinder by Picking • 1016
 Extracting Clouds From 3D Inspection Clouds • 766
 Extracting Colored Meshes • 734
 Extracting Iso-Curves • 728
 Extracting to a New Mesh • 514
 Extrusion • 846, 867, 910

F

Feature Code Libraries • 657
 Feature Codes • 659
 Feature Points • 661
 Feature Set Tool • 657
 Feature Sets • 1067
 Fencing • 391, 394
 Fencing a Set of Points • 757
 Fencing an Area • 510, 632, 684, 697, 717
 Fill Line Breaks • 496
 Filling Holes • 633
 Filtering • 1069
 Filtering (Optional) • 138
 Filtering a 3D Inspection Cloud • 763
 Filtering Altitudes • 726
 Filtering an Area • 684, 698
 Filtering by Range • 138, 1070
 Filtering by Zone • 138, 1070
 Filtering From an Elevation Range • 631
 Filtering Sections • 1097
 Filtering the 2D Inspection Results • 670
 Filtering the Inspections • 685
 Finding an Item in the Project Tree • 157
 Finding the Best Cross Plane • 461, 621
 Fitting a Plane • 461, 621
 Fitting an Axis • 285
 Fitting Geometry to Cloud [From Scan Items] • 381
 Fitting Geometry to Cloud [From Unfitted Items] • 384
 Fitting Tool • 756

- Fitting With a Geometry • 460, 620, 758
- Fixing to an Axis • 830, 835, 899, 902, 904
- Flipping Edges • 516
- Focusing on Targets • 377
- Folder • 1074, 1076
- Font Color • 127
- Forcing Leveled • 436
- Forcing Unleveled • 441
- Frame Creation Tool • 281
- From Current View • 303
- From Path • 637
- From Segments • 645
- From Station View • 359
- From Target View • 362
- From the Menu Bar • 221
- From the Property Window • 220
- From TZF Scans • 302

G

- General • 150
- Generating Key Plan • 299
- Generating Point Color-Coding by Height • 142
- Generating Preview Scans • 349
- Generating Results • 1094
- Geometry • 162, 183
- Geometry Creator Tool • 773
- Geometry Modifier Tool • 859
- Georeferencing Tool • 420
- Getting all Points • 77
- Getting Remaining Points • 78
- Global Views • 203
- Going to a Shooting Position • 202
- Google Earth (KMZ) Format • 1042
- Group and Object Nodes • 80
- Grouping • 366

H

- HASP • 34
- Hiding • 64, 67, 162, 164, 167, 171, 172, 174, 176, 177
- Hiding (or Showing) • 600
- Hiding all • 164
- Hiding all Except those Selected • 165
- Hiding and Displaying Sub-Views • 52
- Hiding/Displaying Input Data • 477
- Hiding/Displaying the Input • 548
- Hiding/Opening • 210

- Hiding/Showing • 53
- Horizontal Pan • 191, 193
- Horizontal Rotation • 190, 194
- Horizontally • 594

I

- Image • 165
- Image Matching Tool • 599
- Image Rectification Tool • 555
- Images • 100, 1063
- Images Tab • 82
- Images Tree • 75, 121, 144
- Importing • 110, 207, 658
- Importing from TZF Files • 432
- Importing Images • 143
- Importing Projects • 86
- Importing SteelWorks Catalogs • 998
- Improving • 613
- In Plane/Cylinder-Based Projection • 680, 691
- In the List Window • 80
- In the Workspace Window • 80
- In Tunnel-Based Projection • 680, 691
- Including a Media File • 128
- Including Another Media Files • 128
- Inserting • 663
- Inserting a Middle Node • 956
- Inserting Nodes • 585
- Inspection Map • 169
- Inspection Map Analyzer Tool • 699
- Inspection Maps • 682, 693, 1078
- Installation of RealWorks • 23
- Installing RealWorks • 26
- Intensity Mapping • 1070
- Intensity-Based Sampling • 273
- Interactive Pan • 404
- Interactive Rotation • 405
- Intersect Tool • 933
- Interval • 474
- Iso-Curves • 727
- Isometric • 187
- IXF Files • 114

J

- JobXML, JOB and RAW Files • 102

K

Keeping In (or Out) • 265
 Keeping Only Displayed Cloud • 820,
 1004, 1015

L

Labels • 171
 LandXML Format • 1055
 LAS Format • 1048
 Length & Quantity • 931
 License Files • 31
 Lighting Directions • 185
 Lines • 606
 Links • 129
 List • 47
 List Window • 145
 Loading Data • 231
 Loading Existing Parameters • 562
 Loading Keyframes • 317
 Loading Markers • 607
 Local Views • 204
 Locating a License File • 35
 Locating an Item in the Project Tree • 156
 Locking a Center • 826, 895
 Locking a Radius • 826, 829, 895, 899
 Locking an Axis • 292
 Locking Center Line Radius • 837, 905
 Locking Center on Line • 826, 896
 Locking on Plane • 841, 843, 907, 908
 Locking on Primitive • 226
 Locking Pipe Radius • 838, 904
 Locking the Origin • 290
 Locking the Origin and an Axis • 293
 Locking to Line (or Axis) • 841, 907
 Logo • 125

M

Main • 55
 Making Horizontal • 893
 Making Parallel • 823, 828, 834, 845, 893,
 898, 902, 903, 909
 Making Parallel to a Direction • 850, 914
 Making Parallel to a Plane • 837, 843
 Making Perpendicular • 823, 829, 834,
 845, 894, 898, 902, 904, 909

Making Perpendicular to a Direction • 837,
 843
 Making Perpendicular to a Plane • 851
 Making Secant to a Box (With Same
 Section) • 856, 916
 Making Secant to a Cylinder • 831, 900
 Making Vertical • 893, 897, 901, 908, 910
 Managing SteelWorks Catalogs • 998
 Manipulating Labels • 481
 Manually Duplicating Polylines • 597
 Mapping With a Texture • 517
 Matched Station Tab • 360
 Matched Target Tab • 363
 Matched Targets • 361
 Matching an Image • 611
 Matching Targets • 443
 Matching With • 442
 Maximizing (or Minimizing) • 601
 Measurement Tool • 237
 Measurements • 1065
 Measuring a Between-Geometry Angle •
 256
 Measuring a Distance • 240
 Measuring a Distance Along a Vertical
 Axis • 243
 Measuring a Distance in a Horizontal
 Plane • 241
 Measuring a Distance to a Fitted Plane •
 247
 Measuring a Fitted Cylinder Diameter •
 246
 Measuring a Geometry Slope Angle • 255
 Measuring a Horizontal Angle • 252
 Measuring a Point-to-Geometry Distance •
 249
 Measuring a Position • 257
 Measuring a Slope Angle • 254
 Measuring a Vertical Clearance
 (Downward) • 245
 Measuring a Vertical Clearance (Upward) •
 244
 Measuring an Angle • 251
 Measuring an Orientation • 258
 Media • 127
 Media Tools • 305
 Menu Bar • 44, 62
 Merging Point Clouds • 223
 Merging Projects • 118
 Mesh Creation Tool • 497

- Mesh Editing Tool • 505
- MicroStation (DGN) Format • 1051
- Modeling • 771
- Modeling Automatically Polylines • 485
- Modeling Extracted Cylinders • 1022
- Modeling Manually Polylines • 487
- Models Tab • 81
- Models Tree • 74, 119, 144
- Modifying • 568, 662, 1099
- Modifying a 3D Plane's Size • 996
- Modifying a Projection Plane • 526
- Modifying a Projection Plane's Position • 564
- Modifying a Projection Plane's Size • 563
- Modifying a Shape • 860
- Modifying Built Elements • 752
- Modifying Markers • 607
- Modifying Stations • 436
- Modifying Target Matching • 442
- Modifying Target Position • 388
- Modifying the Instrument Height • 437
- Modifying the Path for Input TZF Scan Files • 139
- Modifying, Repairing and Removing RealWorks • 30
- Moving • 67, 601, 608, 665
- Moving a Bounding Polyline • 958
- Moving a Geometry • 869
- Moving a Geometry Along a User Defined Vector • 889
- Moving a Geometry by Picking Entities • 875
- Moving a Geometry Using a 2-Point Defined Vector • 890
- Moving a Node • 955
- Moving Along a Direction • 311, 328
- Moving in a Plane • 312, 329
- Moving Nodes • 585
- Moving Polylines • 588
- Moving the Profile • 743
- Multi-Ortho-Projection Tool • 541
- Multiple Slice • 465
- Multi-Selection Mode • 490
- Multi-User Licenses • 36

N

- Name-Based Bundle Adjustment • 429
- Navigating Through Data • 186

- Navigating Within a Station • 198
- Navigation Constraint Tools • 54
- Network Visuals of a Single Station • 172
- Network Visuals of all Stations • 175
- No Projection • 502
- Normal • 619

O

- Object Properties • 1062
- OfficeSurvey • 453
- Offset • 626
- Oil, Gas & Chemical Files • 31
- Opening • 109, 168, 169
- Opening Projects • 85
- Opening Scan Explorer • 134
- Opening the Tool • 238, 262, 268, 282, 307, 337, 342, 351, 368, 374, 399, 415, 421, 456, 471, 482, 497, 506, 521, 541, 555, 573, 599, 617, 636, 657, 667, 673, 687, 700, 738, 749, 756, 761, 765, 774, 816, 859, 917, 933, 945, 1011
- Options • 349
- Organization of Data • 69
- Orientation Tool • 414
- Ortho-Images • 1064
- Ortho-Projection Tool • 520
- Output Files • 1070
- Output Folder • 126
- Overwriting an Existing Texture • 519

P

- Page Configuration • 125
- Pan • 188, 196, 589
- Pan Along Home Frame Axes • 870
- Pan Along Own Axes • 872
- Panning the Profile • 745
- Partial Deselection Mode • 491
- Partial Reselection Mode • 492
- Passing Axis Through Point • 829, 846, 898, 909
- Passing Through a Point • 824, 894, 896
- Passing Through an Axis • 894
- Path Mode • 323
- PDMS Macro Format • 1046
- Perspective • 186
- Pick a Plane, and then Pan • 881
- Pick a Point from Other Geometry, then Rotate • 880

-
- Pick an Axis from Other Geometry, then Pan • 876
 - Pick an Axis from Other Geometry, then Rotate • 878
 - Picking • 390, 393, 396, 643, 661
 - Picking a Color • 720
 - Picking a Cylinder to be Secant With • 987
 - Picking a Dipping Plate Height • 1091
 - Picking a Maximum Height Level • 1092
 - Picking a Plane and a Segment • 794, 974
 - Picking a Plane and Selecting Another Object • 884
 - Picking a plane and selecting another object. • 888
 - Picking a Point • 791, 971
 - Picking a Radial Entity • 983
 - Picking a Triangle • 509
 - Picking a Vertex • 508
 - Picking an Axial Entity • 978
 - Picking an Axial Geometry • 800
 - Picking an Axis and a Point • 984
 - Picking an Axis From Object • 501
 - Picking an Edge • 508
 - Picking an Element • 507
 - Picking an Entity With a Direction • 804
 - Picking an Entity with Center • 795, 975
 - Picking an Entity with Direction • 964
 - Picking an Object Local Frame • 458, 622
 - Picking an Object's Local Frame • 288
 - Picking and Panning a Plane • 882
 - Picking and Selecting a Plane • 886
 - Picking Axis from Object • 678
 - Picking Four Screen Points • 807
 - Picking in the 2D Constraint Mode • 227
 - Picking in the 3D Constraint Mode • 224
 - Picking Items • 224
 - Picking Low and High Elevations • 474
 - Picking Markers • 603
 - Picking Points • 284, 557, 704
 - Picking the Bottom Left Corner of a Box • 806
 - Picking the Origin • 418
 - Picking Three Pairs of Points • 407
 - Picking Three Planes • 792, 972
 - Picking Three Points • 287, 459, 523, 623, 785, 787
 - Picking Two Axial Entities • 797
 - Picking Two Planes • 799, 980
 - Picking Two Points • 460, 622, 783, 789, 798, 979
 - Picking Two Points of a Diameter • 983
 - Picking Two Points of a Radius • 983
 - Plane • 822, 892
 - Plane Bounding Tool • 944
 - Plane Parallel to the Screen View • 624
 - Plane Shape • 694
 - Plane to Circular Torus • 939
 - Plane to Extruded Entity • 940
 - Plane to Sphere • 941
 - Plane-Based Projection • 499, 675, 688
 - Point • 625
 - Point Cloud • 160, 181
 - Point Loading Manager • 229
 - Point Targets • 396
 - Points • 98, 604
 - Polygonal Selection • 263
 - Polyline Drawing Tool • 573
 - Positions • 170
 - Preview BSF Format File • 1074
 - Previewing • 476, 612, 651
 - Previewing a 3D Inspection Cloud • 762
 - Previewing a Multiple Slice • 467
 - Previewing a Rectified Image • 570
 - Previewing a Single Ortho-Image • 551
 - Previewing a Single Slice • 466
 - Previewing a Video • 330
 - Previewing a Volume • 627
 - Previewing an Ortho-Image • 538
 - Previewing Meshes • 503
 - Previewing the Inspections • 681, 692
 - Print • 152
 - Printing • 653, 683, 696
 - Printing an Ortho-Image • 539, 553
 - Printing Inspection Maps • 736
 - Processing Data • 232
 - Profile Matcher Tool • 737
 - Profile/Cross-Section Tool • 636
 - Project Cloud • 76
 - Project Data • 71
 - Project File Formats • 89
 - Projecting • 615
 - Projecting a 3D Point on a Plane • 796, 976
 - Projecting a Box onto a Plane • 805, 915
 - Projection Modes • 186
 - Property • 48
 - PTS • 1077

PTS Files • 117
PTX • 1078
PTX Files • 116
Publish • 124
Published Projects • 130
Publishing Projects • 124

Q

Quick Mode • 309

R

Random Sampling • 270
RealWorks Files • 90
RealWorks Plant Tables • 27
Rectangles • 579
Rectangular Selection • 264
Rectangular Torus • 852, 868, 911
Redoing Operations • 147
Reference Cloud and Moving Cloud • 400
Reference Station • 340, 343, 369
Refine Registration Using TZF Scans • 368
Refining a Measurement • 259
Refining a Mesh • 512
Refining Registration • 370
Refining the Initial Registration • 410
Re-Fitting Geometry to Cloud [From Fitted Items] • 386
Refusing to Convert to the TZF Format • 93
Register Stations With Imported RMX Files • 435
Registered Stations • 361
Registering Stations • 341, 344
Registering the RealWorks Power, Process and Plant Software • 32
Registration • 335
Registration Details • 358
Registration Report • 347, 371
Regular Cone • 833, 901
Removing a Known Point • 440
Removing an Existing Texture • 519
Removing Peaks from a Mesh • 512
Removing SteelWorks Catalog List • 999
Renaming • 658
Renaming Targets • 446
Rendering • 537, 549
Rendering Data • 180
Reordering • 610

Reporting the Volume and Surface Information • 726
Resetting • 610
Resizing • 536
Resizing a Path • 928
Resizing the Circular Path • 313
Resolution • 548
Reversing • 644
Reversing a Path Direction • 328
Reversing the Path Direction • 931
Reversing the Profile • 747
Reversing Triangles • 513
Rotate • 188, 591, 874
Rotating 90° Around Vertical Axis • 533
Rotating Around an Axis • 294
Rotating Counterclockwise 90° • 417, 532
Rotating the Profile • 744

S

Sampling • 137
Sampling by Step • 137, 1076
Sampling Methods • 268
Sampling Spatially • 137
Sampling Tool • 268
Save • 123
Save As • 123
Saving a Measurement • 260
Saving a Volume in a Report • 630
Saving a Volume in the Database • 635
Saving in RTF • 348
Saving Keyframes • 318
Saving Markers • 611
Saving Projects • 123
Saving Results • 496
Saving the 2D Inspection Results • 671
Saving the Adjustment Result • 365
Saving the Cutting Result(s) • 470
Saving the Extracted Data • 715
Saving the Inspections • 686, 698
Saving the Registration Results • 413
Saving the Results • 427, 708
Scaling • 654
Scaling the Profile • 746
Scan Creation • 339
Scan Folder Name • 1074
Scan-Based Sampling • 271
Scans • 97
Scans Tab • 81

- Scans Tree • 72, 118, 144
- Screen Rotation • 192, 195
- Screen View-Based Projection • 502
- Section Interval • 1092
- Segment • 844, 866, 908
- Segmentation Tool • 261
- Selecting • 402, 542, 638, 648, 1100
- Selecting a Catalog File • 1005
- Selecting a Frame Axis • 458, 500, 620
- Selecting a Method • 637
- Selecting a Model for Inspection • 668
- Selecting a New Cloud Data • 817, 1001, 1012
- Selecting a Path • 324
- Selecting a Polyline • 849, 926
- Selecting a Polyline for Bounding • 946
- Selecting a Projection Mode • 498
- Selecting a Projection Plane • 552
- Selecting a Reference • 1006
- Selecting a Reference Frame • 283
- Selecting a Reference Station • 352
- Selecting a Segment Group • 646
- Selecting a Station • 375
- Selecting a Table • 1006
- Selecting a TZF Scan • 376
- Selecting an Element • 507
- Selecting an Existing Profile • 750
- Selecting an Image • 600
- Selecting an Initial Cylinder for Tracking • 1017
- Selecting Items • 489
- Selecting Markers • 602
- Selecting Points & Polylines • 702
- Selecting Polylines • 582
- Selecting Sections & Shifts • 709
- Selecting Stations • 422
- Selecting Volumes & Surfaces • 716
- Selection List • 49
- Selection Mechanism • 144
- Setting a Direction • 326
- Setting a Position • 529
- Setting a Resolution • 536
- Setting a Thickness • 650, 742
- Setting a Tolerance • 650
- Setting an Axial Entity Axis as Axis • 289
- Setting an Image Resolution • 569
- Setting as Home Frame • 1031
- Setting Coordinates • 457
- Setting Corners • 524
- Setting Display Mode • 666
- Setting From Frame • 677
- Setting Low and High Elevation Values • 473
- Setting Over a Known Point • 438
- Setting Preferences • 150
- Setting the Interval Parameter • 644
- Setting the Section Size • 751
- Setting the Thickness Parameter • 668
- Shifting a Project • 296
- Shortcut Keys • 63
- SIMA ASCII Files • 111
- Simplification • 495
- Single Slice • 465
- Single-User License • 35
- Smoothing a Mesh • 512
- Smoothing a Path • 328
- Smoothing Cells • 634
- Smoothing Extracted Cylinders • 1021
- Solids for AutoCAD • 1056
- Spatial Sampling • 269, 1069
- Specifying Coordinates • 284
- Sphere • 825, 861, 895
- Spherical Targets • 389
- Splitting a Rectified Image • 571
- Splitting an Ortho-Image • 539
- Standard Selection Mode • 490
- Standard Views • 203
- Start and End Positions • 641
- Starting the Tool • 1000
- Starting Tracking Cylinders • 1018
- Station • 170
- Station Registration Parameters • 431
- Station-Based • 196
- Station-Based projection • 502
- Stations • 96, 444
- SteelWorks Creator Tool • 1000
- Step & Length • 930
- Step & Quantity • 929
- Step-by-Step Mode • 314
- Storage Tank Application • 28, 1087
- Surface to Model Inspection Tool • 687
- Surveying Network ASCII Files • 108
- Swapping a Local License for a Network License and Vice Versa • 34
- System Requirements • 25

T

- Tank Calibration Check • 1097
- Tank Calibration Tool • 1089
- Target Analyzer Tool • 373
- Target Types • 338
- Target-Based Registration Tool • 350
- Targets • 443
- Targets Tab • 82
- Targets Tree • 73, 120, 145
- Three Sub-View Configuration • 52
- Tilt (or Rotate) • 195
- Title • 125
- Tolerance • 475
- Toolbars • 45, 55, 66
- Tools [in Modeling] • 61
- Tools [in OfficeSurvey] • 60
- Tools [in Registration] • 59
- Tools and Commands • 55
- Top Aligning all Planes • 545
- Topography-Based Sampling • 279
- Trimble 3D Scanning Files • 89
- Trimble Survey Project Files • 95
- Trimble TX5 and Other FLS Files • 107
- Tunnel Shape • 696
- Tunnel-Based Projection • 679, 690
- Twin Surface Inspection Tool • 672
- Two Sub-View Configuration • 52
- TZF Files • 91
- TZF Scan • 177
- TZF Scans • 135
- TZS Files • 93

U

- Undocking • 65
- Undoing Operations • 146
- Units • 152
- Unmatched Station Tab • 361
- Unmatched Target Tab • 364
- Unmatched Targets • 361, 362
- Un-matching Targets • 445
- Unregistered Stations • 362
- Updating • 38
- Updating Network • 397
- Updating RealWorks • 29
- Upgrading • 36
- User Defined • 1043
- User Interface • 43

- Using Constraints • 821
- Using Existing Parameters • 525
- Using Segment Width • 649
- Using Surface Values • 724
- Using the Current Camera View • 524
- Using the Current View as 3D Plane • 1008
- Using the Fit Axis Mode • 291
- Using the Pick Axis Mode • 291
- Using the Three Point Pick Mode • 290
- UTM to WGS84 Latitude and Longitude • 1045

V

- Validating a 3D Direction • 970
- Validating a 3D Plane • 997
- Validating a 3D Point • 977
- Validating an 3D Axis • 981
- Validating Bounds • 962
- Vertical Pan • 190, 193
- Vertical Rotation • 191, 194
- Vertically • 596
- Video Creator • 307
- View Alignment • 58, 199
- View Manager • 51
- Viewer • 151
- Viewing • 132
- Viewing from a Projection Plane' Side • 531, 553
- Viewing Inwards/Outwards • 310
- Viewing the 2D Inspection Results • 669
- Visualizing Data • 160
- Volume Calculation Tool • 617

W

- Walkthrough • 195, 196
- Welcome • 15
- What's New in RealWorks • 17
- Window • 61
- Windows • 46, 64
- With Permanent Constraints • 192
- With Temporary Constraints • 189
- With Wizard • 105
- Without Constraints • 188
- Without Wizard • 106
- Working Environment • 41
- Working Frame Tool • 61
- Workspace • 46

WorkSpace Window • 144

Z

Zoom In (or Out) • 189

Zooming All • 200

Zooming In (or Out) • 199

Zooming on Selection • 200