

# User Guide

## **Trimble® Business Center Heavy Construction Edition**

Version 1.20



**Corporate Office**

Trimble Navigation Limited  
Engineering and Construction Division  
5475 Kellenburger Road  
Dayton, Ohio 45424-1099  
U.S.A.

Phone: +1-937-233-8921

Toll free (in USA): +1-800-538-7800

Fax: +1-937-233-9441

[www.trimble.com](http://www.trimble.com)

**Copyright and Trademarks**

© 2005-2008, Trimble Navigation Limited. All rights reserved.

The Globe & Triangle logo and Trimble are trademarks of Trimble Navigation Limited. All other trademarks are the property of their respective owners.

**Release Notice**

This is the help for version 1.20 of Trimble® Business Center - Heavy Construction Edition software.

# Contents

<b>Welcome to Trimble Business Center - Heavy Construction Edition</b>	<b>1</b>
--	----------

---

<b>Get Started</b>	<b>3</b>
--------------------	----------

---

Register This Software .....	3
Retaining User Settings When Upgrading .....	5
Get Familiar with the Interface .....	6
Project Explorer .....	7
Selection Explorer .....	8
View Filter Manager .....	9
Status Bar .....	10
Device Pane .....	11
Command Pane .....	13
Properties Pane .....	14
Flags Pane .....	14
Coordinates Scroll .....	15
Data Views .....	16
Customize the Menu .....	16
Customize the Toolbar .....	18
Customize the Keyboard .....	20
Customization Options and Tools .....	21
Troubleshoot a Toolbar or Menu Problem .....	21
Find Help Topics .....	22
Help Options .....	23

<b>View, Navigate, and Select</b>	<b>25</b>
-----------------------------------	-----------

---

Graphic Views .....	25
Spreadsheets and Other Views .....	29
Pane and Data View Positioning .....	31
Data View Display Formats .....	32
Tabbed View Arrangement .....	34
2D View Navigation .....	35
3D View Navigation .....	37
3D View Settings .....	38
Keyboard Navigation .....	40
Mouse Modes .....	42
Data Selection .....	42
View and Edit an Object's Properties .....	64
Delete an Object .....	65
Edit an Object .....	66
Undo or Redo an Action .....	67

Manage the Data in Your Views.....	67
Troubleshoot a View or Selection Problem .....	84

**Calculate and Enter Values 86**

---

Understanding COGO Controls .....	86
COGO Expressions, Units, and Entry Formats .....	87
Set the Pick Aperture.....	88
Snaps Modes and Commands.....	89
Enter an Angle.....	92
Enter a Bearing.....	94
Enter a Coordinate .....	97
Enter a Distance .....	120
Enter an Elevation.....	125
Enter an Offset .....	128
Enter a Station .....	133
Measure Values Between Points.....	135
Measure Options .....	136
Measure Angles .....	137
Measure Angle Options.....	138

**Set Up Projects 139**

---

Choose Application Options.....	139
Choose Project Settings.....	147
Choose Local Site Settings.....	159
Create a New Project.....	161
Use a Project Template .....	162
Open an Existing Project .....	164
Save a Project.....	165
Archive a Project.....	166
Print a View or Report.....	166
Troubleshoot a Project Problem.....	167
Troubleshoot a Program Freeze.....	168

**Set Up Geodetic Reference Data 170**

---

Understanding Geodetic Reference Data .....	170
Define the Coordinate System .....	171
Use a Datum .....	174
Define a Projection.....	176
Use a Geoid.....	177
Calibrate a Site.....	181
Using Geoid Models .....	183
Rules for ensuring a useful calibration.....	183

**Import Data 193**

---

Import Data .....	193
-------------------	-----

Drag and Drop to Import .....	194
Importable Data Formats .....	195
Import ASCII Files .....	195
Import Data Collector Files (.dc) .....	196
Import CAD Files (.dxf/.dwg) .....	196
Import GENIO Files .....	198
Import GNSS Job Files (.job) .....	199
Import LandXML Files (.xml) .....	200
Import MicroStation Files (.dgn) .....	204
Import REB Files (.reb) .....	206
Import Trimble Surface Files (.ttm) .....	207
Import Wirth YXZ Files (.yxz) .....	207
Import DiNi Digital Level Files (.dat) .....	207
Use Terramodel (.pro) Data .....	208
Import Data in a Custom Format .....	209
Run an Import Report .....	215
Troubleshoot an Import Problem .....	215

**Transfer/Synchronize Data 217**

---

Prepare to Connect a Field Device .....	217
Office Synchronizer .....	219
Device Pane .....	219
Direct Connection .....	221
Data Synchronization .....	223
Upload Geodetic Reference Data .....	227
Troubleshoot a Data Transfer/Synchronization Problem .....	228

**Work with GNSS Data 229**

---

Planning Utility .....	229
Check GNSS Data .....	229

**Work with Total Station Data 233**

---

Understanding Total Station Data .....	233
Workflow for Total Station Data .....	238
View Total Station Data in Project Explorer .....	240
Total Station Data Errors .....	247
View and Edit Mean Angle Residuals .....	248
Run a Mean Angle Report .....	249

**Work with Level Data 251**

---

Understanding Level Data .....	251
Workflow for Level Data .....	252
Import DiNi Digital Level Files (.dat) .....	253
View and Edit Level Data .....	255
View Level Data in Project Explorer .....	260

Level Data Errors.....	264
Adjust Level Runs .....	265
Merge Level Runs.....	266
Note on Level Runs Without Benchmarks .....	267

**Work with Point Data 269**

---

Understanding Point Types .....	269
Add and Edit Points and Coordinates .....	270
Calculate the Inverse Between Points.....	280
Inverse Options .....	281
Measure Values Between Points.....	282
Measure Options .....	283
Points Spreadsheet .....	284
Troubleshoot an Import Problem.....	285

**Work with Line Data 286**

---

Create and Edit an Alignment .....	286
Create and Edit a Linestring .....	300
Create and Edit a Simple Breakline .....	312
Breakline Options.....	314
Break a Line .....	314
Join Lines .....	315
Delete a Line Segment.....	316
Set a Line Elevation.....	317
Explore an Object .....	318
Explode a Block .....	319

**Work with Surface Data 320**

---

Create and Edit a Surface .....	320
Create a Surface Boundary and Contours .....	339
Add Surface Materials .....	349
Shrink 3D Faces.....	356

**Work with Feature Data 359**

---

Understanding Feature Data .....	359
Workflow for Feature Data .....	361
Feature Code Processing Settings.....	362
Enter, Edit, and Delete Feature Code Strings .....	363
Rules for Merging Feature Attributes .....	365
Split Line Features.....	365
Process Feature Codes.....	367
Export Geodatabase Files (.xml) .....	368
Feature Definition Manager Utility.....	368

**Prepare Data for Site Positioning Systems 370**

---

Prepare Data for Site Positioning Systems ..... 370  
 Understanding SPS and SCS ..... 372  
 Understanding Sites, Designs, Work Orders, and Synchronization ..... 373  
 SCS Data Manager ..... 376  
 Manage a Site and Site Data ..... 377  
 Manage Designs and Design Data ..... 393  
 Manage Work Orders and Results ..... 406  
 Troubleshoot an SCS Site or File Problem ..... 425  
 View Construction Data in Project Explorer ..... 427

**Run Reports 429**

---

Run an Alignment Geometry Report ..... 429  
 Run an Earthwork Report ..... 430  
 Run an Import Report ..... 431  
 Run a Mean Angle Report ..... 431  
 Run a Point Derivation Report ..... 432  
 Run a Point List Report ..... 432  
 Run a Project Computation Report ..... 433  
 Run a Renamed Point List ..... 433  
 Run a Site Calibration Report ..... 434  
 Run a Surface Report ..... 434  
 Run a Work Order Report or Summary ..... 435  
 Customize a Report ..... 436  
 Report View ..... 437  
 Report Options ..... 438

**Export Data 441**

---

Export Data ..... 441  
 Export and Upload Data Formats ..... 442  
 Export Related Files ..... 442  
 Export ASCII Files ..... 443  
 Export CAD Files (.dxf/.dwg) ..... 444  
 Export Geodatabase Files (.xml) ..... 445  
 Export LandXML Files (.xml) ..... 445  
 Export Trimble Data Collector Files (.dc) ..... 448  
 Export Trimble Surface Files (.ttm) ..... 448  
 Export Data in a Custom Format ..... 448

**Troubleshoot Issues 453**

---

Troubleshoot a Coordinate System Problem ..... 453  
 Troubleshoot a Data Transfer/Synchronization Problem ..... 453  
 Troubleshoot a Layer or View Filter Problem ..... 454  
 Troubleshoot an Import Problem ..... 455  
 Troubleshoot a Program Freeze ..... 455

Troubleshoot a Project Problem..... 457  
Troubleshoot a Toolbar or Menu Problem ..... 457  
Troubleshoot a View or Selection Problem ..... 457  
Troubleshoot an Import Problem..... 459  
Troubleshoot an SCS Site or File Problem ..... 460

**Use Related Utilities** **463**

---

Coordinate System Manager ..... 463  
Feature Definition Manager Utility..... 463  
Planning Utility..... 464  
External Tools Manager ..... 464  
SCS Data Manager ..... 465  
Trimble Configuration Utility ..... 465  
Trimble Data Transfer Utility ..... 466

**Index** **467**

---



## Welcome to Trimble Business Center - Heavy Construction Edition

Trimble® Business Center - Heavy Construction Edition is office software designed to help construction contractors prepare construction data for use within Trimble site positioning systems (SPS) that employ SCS900 and SCS900 Tablet Edition site controller software (SCS). For those new to Trimble Business Center, we recommend reading these topics first: [Get Familiar with the Interface](#), the [Project Explorer](#) (on page 7), the [Selection Explorer](#) (on page 8), the [Plan View](#) (on page 25), and [Prepare Data for Site Positioning Systems](#) (on page 370).

### Features:

- Import and export of site and design data in multiple file types
- Graphical views of project data, including plan, profile, 3D, and cross-section
- Hierarchical tree view of project data and list view of selected objects
- Surface modeling and volume calculations
- Alignment creation
- Reports and quality assurance tools
- Site Positioning System management:
  - Management of Trimble SPS operations on the job site
  - Preparation of site and design data for field crews carrying out SPS operations
  - Creation and monitoring of work orders for field crews using SCS900
  - Graphic view of the site and design data associated with a work order, as it will be displayed to field crews by SCS900
  - Import of field measurement data resulting from SPS operations

**Note:** These core features provide the foundation for advanced functionality in the form of future, add-on modules.

### Related topics

- ❑ About Trimble Business Center Command
- ❑ Products Command
- ❑ [Register This Software](#) (on page 3)
- ❑ Start Page Command

## CHAPTER 2

# Get Started

## Register This Software

Register this software so you can receive product upgrades, support, and warranty-related services.

**Note:** The *Product Registration* dialog appears in English only. The following steps guide you through the dialog in your installed language.

### To register:

1. Select **Help > About**. The *About* dialog displays.
2. Click **Register**. The *Product Registration* dialog displays, showing these notes:

Please register your new software with the MyTrimble system. The MyTrimble system provides access to the latest product information from, and helps track your company's products and warranty information.

Enter your email address. Each MyTrimble account has an associated email address. If your company has an existing MyTrimble account, enter that email address.

If your company does not have an existing MyTrimble account, then you will create an account in the next step.

**Note:** We value your privacy. We will not sell, rent, or share this information with third party marketing firms or other manufacturers or products. For further details, please click on the link to see our Privacy Statement.
3. Type your email address in the box, and click **Next**. These notes appear:

To create an account, enter your contact details below. **All fields are required**. If your company already has an account, click **Back** to return to the previous screen and enter the email address for that account.

**Note:** These contact details may be used for the delivery of product upgrades or enhancements, and warranty-related services. Refer to our Privacy Statement for additional details.

4. Type the following in the boxes: (all are required)

- First name
- Last name
- Company name
- Address
- City
- State/Province
- Postal code
- Country
- Phone number

5. Click **Next**. This note appears:

If you have a dongle attached to your computer, its serial number has been read and entered below. If your software did not come with a dongle, refer to the sticker on the software box for your serial number. These boxes should be automatically filled:

- Serial number
- Product name
- Part number
- Ship date

**Note:** Your software product may not require a dongle or other license.

6. If the boxes are empty, type the serial number for your software in the **Serial Number** box, and click **Product Lookup**. Otherwise, skip to step 9.

**Note:** If you receive this warning: Serial Number Not Found! Please make sure you have entered the serial code correctly, click **Reenter Serial Number**, and correct the serial number.

7. If you have a company-specific reference number for software, you can type it in the **Reference Number** box.

8. Click **Submit**. This note appears:

Thank you for registering! Your software has been successfully registered! Your password has been sent to your email. Please check your email to find the details on how to access your MyTrimble account.

This note may also appear: Our system found that you have other products that are not yet registered. Please choose below which product you want to register and click **Register**.

9. Check boxes for any other products you want to register, and click **Register**. Otherwise, click **Not this time**.

**Note:** To login to your MyTrimble account, please visit our web site's *Register* page.

## Retaining User Settings When Upgrading

As you use the program, many of your settings and other customizations are saved as files in an application data folder. These settings, which remain constant regardless of which project is open, include:

### Application settings

These program-wide settings include startup preferences, default file locations, and display properties. Application settings are primarily found in the *Options* dialog.

### Custom Import and Export Format Definitions

These include changes to how file formats are defined in the *Import Format Editor* and *Export Format Editor*.

### Project Templates

These include project settings, coordinate systems, view filters, selection sets, and data that you have saved as project templates.

### Internet Download Configurations

These include new data provider groups and Internet sites that you have added to the *Internet Download* command.

### Baseline Processing and Network Adjustment Styles

These include combinations of project settings that you have defined as templates for baseline processing and network adjustment.

When you upgrade from your current version of the program to a newer version, the installation program searches for previous files containing these settings and customizations. If any are found, the *Copy Settings* dialog appears.

### To retain previous settings and customizations:

1. In the *Copy* column, uncheck the box for each old file that you do not want to retain in the upgrade.
2. Check the box for each old file that you do want to use to overwrite the new file.
3. Click **Copy Selected Files**. The previous settings and customizations that you selected are copied to the new installation.

**Note:** Any files that conflict with the files in the new installation are marked with a red flag and are not selected by default.

**Note:** It is always a good idea to confirm your project and application settings in the new installation to make sure that any new options in the current version are set to the defaults you want.

**Note:** Customized menus and toolbars cannot be saved when you upgrade.

## Get Familiar with the Interface

This software comes with an integrated user interface, including:


## Interface elements

<a href="#">Menu</a> (see "Customize the Menu" on page 16)	Gives you comprehensive access to all available commands.
<a href="#">Toolbar</a> (see "Customize the Toolbar" on page 18)	Gives you quick access to the most commonly used commands and views using icons.
<a href="#">Data views</a> (on page 16)	Allows multiple views of data in the <a href="#">plan view</a> (on page 25), <a href="#">3D view</a> (on page 28), <a href="#">profile view</a> (on page 26), <a href="#">cross-section view</a> (on page 26), and <a href="#">surface slicer view</a> (on page 27). The data view area can be set up as a tabbed interface or a multiple window interface. Navigation and selection can be controlled both graphically and by commands.
<a href="#">Status bar</a> (on page 10)	Includes status information, current units, an error flag (indicating computation errors), an indicator that the project should be computed again, the number of currently selected objects, and a coordinate display.
<a href="#">Project Explorer</a> (on page 7)	Shows a tree view of project data that includes sections for the objects and imported files, enabling you to easily select any object.
<a href="#">Selection Explorer</a> (on page 8)	Shows the currently selected objects, as well as saved sets of objects known as 'selection sets'.
<a href="#">View Filter Manager</a> (on page 9)	Lets you specify what data types and layers are visible and selectable in graphic views.
<a href="#">Command Pane</a> (on page 13)	Provides a consistent place to work through most commands.
<a href="#">Properties Pane</a> (on page 14)	Displays the properties associated with the currently selected object(s), enabling them to be edited.
<a href="#">Flags Pane</a> (on page 14)	Lists objects with import or computation errors.

## Project Explorer


The *Project Explorer* displays your project data organized in a tree structure.

### To display and pin the Project Explorer:



- Do one of the following:
  - Click the  icon on the toolbar.

- Select **View > Project Explorer**.
- Press **[F9]** on the keyboard.

The **Project Explorer** displays, docked on the left side of the application window, or where you positioned it last.

2. If desired, pin the explorer open by clicking the  icon at the top. If the pane is unpinned, the pane can "slide" to the side and out of view. To show it again, click the **Project Explorer** tab.

### Using the Project Explorer

- To expand nodes, click the  icon. To collapse nodes, click the  icon.
- To select a node or a data object, click it.
- To display the properties for an object, double-click it. The **Properties** pane displays.
- To access common commands for an object, right-click for a context menu.

### Related Topics

- ❑ [Select from the Project Explorer](#) (on page 48)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Properties Pane](#) (on page 14)


## Selection Explorer

The **Selection Explorer** is a pane that lists the selection sets in your project in the top section and lists the objects in the active set in the bottom section.

- When you click on a selection set, all the objects in the set are selected in the project.
- When you select one or more objects in the list of objects, those objects are selected in the project.
- When **<Selection Snapshot>** is active, objects selected in the graphics area are listed.

Using selection sets makes accessing and selecting groups of commonly-used objects faster and more consistent.


### To display and pin the Selection Explorer:

1. Do one of the following:
  - Select **View > Selection Explorer**.
  - Select **Select > Selection Set > Selection Explorer**.
  - Click the  icon on the toolbar.



- Right-click in a graphic view and select **Selection Explorer** from the context menu.

The **Selection Explorer** displays, docked on the left side of the application window, or where you positioned it last.

2. If desired, pin the explorer open by clicking the  icon at the top. If the pane is unpinned, the pane can "slide" to the side and out of view. To show it again, click the **Selection Explorer** tab.

### Related topics


- ❑ [Understanding Selection Sets](#) (on page 54)
- ❑ [Create and Use Selection Sets](#) (on page 57)
- ❑ [Modify Selection Sets](#) (on page 58)
- ❑ [Pane and Data View Positioning](#) (on page 31)

## View Filter Manager

The **View Filter Manager** is a pane in which you can select data types and layers to specify what is visible in the current graphic view, helping you reduce and simplify what you see. As you make changes in the manager, the view updates to reflect them.


View filters are saved sets of criteria that control what data and layers are displayed in the views. View filters can be defined separately for each type of view so that only the data that is important for the current phase of your work is displayed. When you change to a different view, the current and available view filters may change as well, because view filters are saved with views. The view filters for each view in your project can be accessed from the view filter list on the toolbar.

### To display and pin the View Filter Manager:


1. Do one of the following:
  - Select **View > View Filter Manager**.
  - Click the  icon on the toolbar.

The **View Filter Manager** displays, docked on the left side of the application window, or where you positioned it last.

**Note:** If the **Project Explorer** or **Selection Explorer** are also active, they may share the same pane, and be accessible as tabs at the bottom of the pane.

2. If desired, pin the manager open by clicking the  icon at the top. If the pane is unpinned, it can "slide" to the side and out of view. To show it again, click the **View Filter Manager** tab on the left edge of the application window.

## Using the View Filter Manager

- When you check and uncheck boxes for data types and layers in the *View Filter Manager*, the current view changes in response.
- Arrange the order of the data type groups in the tree by right-clicking on a group and selecting *Move Up* or *Move Down* from the context menu.
- To set the selectability of data types and layers, click the  icon on the pane's toolbar to display the *Advanced View Filter Settings* dialog.

### Related topics

- ❑ [Create a View Filter](#) (on page 73)
- ❑ [Edit a View Filter](#) (on page 75)
- ❑ [Filter a View](#) (on page 76)
- ❑ [Pane and Data View Positioning](#) (on page 31)

## Status Bar

The status bar, located at the bottom of the application window, displays several useful pieces of information:



<b>Status line</b>	Displays information about the current command.
<b>Snap button</b>	Click this to display the <b>Snap mode</b> dialog, in which you can set running snap modes.
<b>Units button</b>	Displays the current distance units. Click this to display the <b>Units</b> section of the <b>Project Settings</b> dialog.
<b>Flag Pane button</b>	Appears if errors have been detected in the project. Click this to display the <b>Flags</b> pane. Flagged items have associated messages or errors.
<b>Compute Project button</b>	Appears if changes made to the data require that final coordinates for points be recalculated. Click this to start the <b>Compute Project</b> command.
<b>Number of selected objects</b>	Displays the number of objects that are currently selected.
<b>Plan view dimensions icon</b>	Appears when the cursor is not in the data view area.
<b>Plan view dimensions or Coordinates display</b>	Displays the XY dimensions of the current view, when the cursor is not in the view area.  Displays the true northing and easting coordinate of the current cursor location, when the cursor is within view area.
<b>Display Coordinate Pane checkbox</b>	Check this to display the <b>Coordinates</b> pane, which shows the current northing, easting, latitude, and longitude of the cursor.

### Related Topics

- ❑ [Running Snap Modes](#) (see "Set Running Snap Modes" on page 89)
- ❑ [Change Project Units](#) (see "Unit Settings" on page 151)
- ❑ [Flags Pane](#) (on page 14)
- ❑ [Coordinates Pane](#) (see "Coordinates Scroll" on page 15)

## Device Pane

The **Device** pane enables you to directly access Microsoft® Windows® CE-based field devices or the data synchronization area (also known as the **root sync folder** in the Office Synchronizer utility), which contains the files maintained by Office Synchronizer.

### To display the Device pane:

Do one of the following:

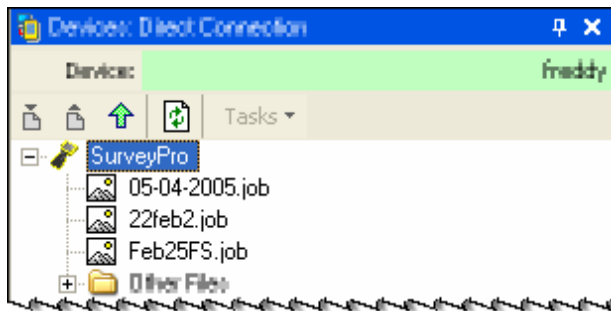
- Click the  icon on the toolbar.

- Select **View > Device Pane**.
- Press **[F10]** on the keyboard.

The **Devices** pane displays, docked on the left side of the application window, or where you positioned it last.

#### To connect to a field device:

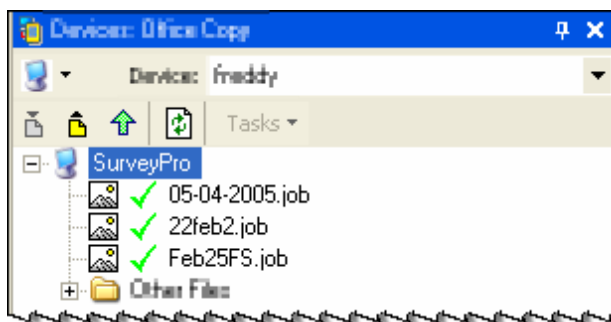
1. Connect the field device to the computer using a USB or serial connection.
2. If the device asks if you want it to be connected, click **Yes**. The **Device** pane displays, showing a list of files on the device.




3. As needed, [upload](#) (see "Upload Files (via Direct Connection)" on page 221) or [download](#) (see "Download Files (via Direct Connection)" on page 221) files via this direct connection.

#### To connect to field data in the data synchronization area:

In Office copy mode, the **Device** pane points to a folder on your office computer that contains the data previously synchronized from the field device, using the [Office Synchronizer utility](#) (see "Office Synchronizer" on page 219).



1. To verify that data in the synchronization area is selected, click the  icon, and verify that **Office Copy** is checked in the drop-down list. The contents of the synchronizer root folder display.
2. As needed, [upload files](#) (see "Upload Files (via data synchronization)" on page 224), [upload tasks](#) (see "Upload Tasks (via data synchronization)" on page 225), or [download files](#) (see "Download Files (via data synchronization)" on page 223) from the data synchronization area.

## Related topics

- ❑ [Office Synchronizer](#) (on page 219)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Prepare to Connect a Field Device](#) (on page 217)

## Synchronizing Data for SCS

In preparing and managing site data, design data, and work orders for site controller software (SCS), you **do not** need to use the **Device** pane to synchronize with the data synchronization area. The synchronization happens automatically when you resolve conflicts and finish SCS commands.

## Command Pane

The **Command** pane gives you access to the **All Commands** list, a comprehensive list from which you can execute most commands. The **Command** pane also provides a place for you to work through many commands.

### To display the Command pane:

- Select **View > Command Pane**.
- Press **[F12]**.

The **Command** pane displays docked on the right side of the application window, or where you positioned it last.

### To run a command from the Command pane:

Do one of the following:

- Enter a command in the **Command** box (command line).
- Double-click a command in the **Recent Commands** list.
- Click a command in the **All Commands** list.

When a command is active in the **Command** pane, these options are available on the pane's toolbar.

### Options



Click this to display a list of the commands that are currently on the stack.



In the default command pane, click this to display the last/current command on the stack.

In any other command pane, click this to display the default command pane.

## Related Topics

- ❑ [Pane and Data View Positioning](#) (on page 31)

## Properties Pane


The **Properties** pane shows properties for selected objects, enabling you to edit certain values. If you select a single object, the properties for that object are displayed. If you select multiple objects, the properties common to all of them are displayed. You can edit the common properties, or select a subset of the selected objects using the drop-down list near the top of the pane.

To view the properties of another object in the **Properties** pane, click the object within any data view or pane. If the **Properties** pane is displayed, selecting any object will show its properties.

**Note:** The toolbar icons and context menu items available in the **Properties** pane depend on the types of objects you have selected. The **Properties** pane also enables you to use COGO controls and snap commands within certain property boxes.

**Note:** An icon for the type of object you have selected appears at the top of the **Properties** pane. If a flag icon appears instead of the object icon, there are import or computation errors associated with one or more of the selected objects. Open the **Flags Pane** (on page 14) for details.

### To display the Properties pane:

- Click the  icon on the toolbar.
- Select **Edit > Properties**.
- Double-click an object in the **Project Explorer**.
- Right-click an object in a view, spreadsheet, or the **Project Explorer** and select **Properties**.
- Press [F11].

The **Properties** pane displays, docked on the right side of the application window, or where you positioned it last.


### Related Topics

- ❑ [View and Edit an Object's Properties](#) (on page 64)
- ❑ [Pane and Data View Positioning](#) (on page 31)

## Flags Pane

The **Flags** pane shows import or computation errors. You can select individual or multiple objects from the **Flags** pane if the objects have been flagged with errors. If there are no objects in the **Flags** pane, no objects have been identified as having errors.

**To display the Flags pane:**

- Click the  icon on the toolbar or the status bar if flags are present.
- Select **View > Flags Pane**.

The *Flags* pane displays at the bottom of the application window, or where you positioned it last.

**To highlight points using the Flags pane:**

- In the *Flags* pane, select a point to view. The selected point(s) highlight in any graphic views and spreadsheets you have open.

**Tip:** [Ctrl] + click to select multiple objects, or [Shift] + click to select a range in the *Flags* pane.

**Related Topics**

- ❑ [Compute Project Command](#)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Select from the Flags Pane](#) (on page 48)
- ❑ [Status Bar](#) (on page 10)

## Coordinates Scroll

The *Coordinates* scroll displays values, such as northing, easting, latitude, longitude, elevation, and offset, based on the position of the cursor in a graphic view. The values shown depend on what type of view the cursor is in.

**To display the Coordinates scroll:**

- Click the checkbox at the right end of the status bar.
- Select **View > Coordinates Scroll**.

The *Coordinates* scroll displays.

**To use the Coordinates scroll:**

1. Display the scroll and right-click in it for options.
2. Select any type of value to show or hide when using the scroll.

**Note:** Although all of the possible values can be selected in the context menu, only certain values will display for each type of graphic view. For example, elevation and offset values will display in profile and cross-section views, but not in the plan view. The coordinates scroll cannot be used in the 3D view.

3. Move the cursor into a 2D view. The values at the cursor's position are displayed in the scroll.

## Related Topics

- ❑ [Coordinate System Manager](#) (on page 173)
- ❑ [Change the Coordinate System](#) (on page 149)
- ❑ [Define a New Coordinate System](#) (on page 149)
- ❑ [Restore the Original Coordinate System File](#) (on page 150)
- ❑ [Pane and Data View Positioning](#) (on page 31)

## Data Views

You can view your project data in a variety of graphical, tabular, and chronological formats, such as:

- [Plan view](#) (on page 25)
- [Profile view](#) (on page 26)
- [3D view](#) (on page 28)
- [Points spreadsheet](#) (on page 29)

## Customize the Menu

You can customize the menus by:

- Rearranging menu commands
- Adding a command to a menu
- Deleting a command from a menu
- Saving a layout

### **Adding external tools to the *Tools* menu To rearrange menu commands:**

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click the **Commands** tab.
3. Click **Rearrange Commands**.
4. In the *Rearrange Commands* dialog, select **Menu Bar** option and then select a menu from the drop-down list.
5. In the *Commands* area, highlight the menu command that you want to move.
6. To move the menu item, do one of the following:
  - Click **Move Up** to move the item up the menu list.
  - Click **Move Down** to move the item down the menu list.
  - Click **Close** to exit, or click **Reset** to return to the default setting.



**To add a command to a menu:**

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click the **Commands** tab.
3. Click **Rearrange Commands**.
4. In the *Rearrange Commands* dialog, select **Menu Bar** and then select a menu from the drop-down list.
5. Click **Add**.
6. In the *Add Command* dialog, select a category and then the command that you want to add to the menu selected in the *Rearrange Commands* dialog.
7. Click **OK**.
8. Do one of the following:
  - Click **Close** to exit.
  - Click **Move Up** or **Move Down** to move the command if you want it in a different position.
  - Click **Reset** to remove the added command and return to the default setting.

**To delete a command from a menu:**

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click the **Commands** tab.
3. Click **Rearrange Commands**.
4. In the *Rearrange Commands* dialog, select **Menu Bar** and then select a menu from the drop-down list.
5. In the *Commands* area, highlight the menu command that you want to delete.
6. Click **Delete**.
7. Click **Close** to exit, or click **Reset** to return to the default setting.

**To save a custom layout:**

After customizing a menu using one of the procedures above, save it so that the new layout appears each time you open the software.

1. Click the **Save/Load** tab.
2. Click **New** and give your layout a name and click **OK**. Your layout now appears in the *Saved Layouts* window.

**To add a new tool to the menu:**

1. Select **Tools > External Tools Manager**. The *External Tools Manager* displays.

2. Click **Add**. *[New Tool]* appears in the *Menu Contents* list.
3. Type a name for the tool in the *Title* box.
4. Next to the **Command** box, click the [browse] icon to browse for a tool file. For example, if you want to add the executable for the calculator, browse to C:\WINDOWS\system32\calc.exe.
5. If needed, click **Move Up** or **Move Down** to change the position of the new item in the menu.
6. Click **OK**. The tool appears in the tools menu.

#### To delete a tool from the menu:

1. In the *External Tools* dialog, highlight the tool to delete.
2. Click **Delete**.
3. Click **OK**.

#### Related topics

- ❑ [Customize the Toolbar](#) (on page 18)
- ❑ [Customize the Keyboard](#) (on page 20)
- ❑ [External Tools Manager](#) (on page 464)

## Customize the Toolbar

You can customize the toolbars by:

- Selecting toolbars to display
- Creating a new toolbar
- Adding a command to a toolbar
- Rearranging toolbar commands
- Deleting a command from a toolbar

**Note:** Click **Reset** to return to the default setting.

#### To select toolbars to display:

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click the **Toolbars** tab.
3. In the *Toolbars* list, select or clear the required toolbar(s).
4. Click **Close**.

**To create a new toolbar:**

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click the **Toolbars** tab.
3. Click **New**.
4. Assign a name to the toolbar, for example `TOOLS`.
5. Select a location from the drop-down list.
6. Click **OK**. The toolbar appears in the project toolbar.

**To add a command to a toolbar:**

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click the **Commands** tab.
3. Click **Rearrange Commands**.
4. In the *Rearrange Commands* dialog, select the **Toolbar** option, then select a toolbar from the drop-down list.
5. Click **Add**.
6. In the *Add Command* dialog, select a category then the command that you want to add to the toolbar selected in the *Rearrange Commands* dialog.
7. Click **OK**.
8. Do one of the following:
  - Click **Close** to exit.
  - Click **Move Up** or **Move Down** to move the command if you want it in a different position.

**To rearrange toolbar commands:**

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click the **Commands** tab.
3. Click **Rearrange Commands**.
4. In the *Rearrange Commands* dialog, select the **Toolbar** option, then select a toolbar from the drop-down list.
5. In the **Commands** area, highlight the toolbar command that you want to move.
6. To move the toolbar item, do one of the following:
  - Click **Move Up** to move the item up the toolbar list.
  - Click **Move Down** to move the item down the toolbar list.
  - Drag and drop items.

7. Click **Close** to exit.

#### To delete a command from a toolbar:

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click the **Commands** tab.
3. Click **Rearrange Commands**.
4. In the *Rearrange Commands* dialog, select the **Toolbar** option, then select a toolbar from the drop-down list.
5. In the *Commands* area, highlight the toolbar command that you want to delete.
6. Click **Delete**.
7. Click **Close** to exit.

#### Related topics

- ❑ [Customize the Menu](#) (on page 16)
- ❑ [Customize the Keyboard](#) (on page 20)

## Customize the Keyboard

#### To customize the keyboard:

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click **Keyboard**.
3. In the *Customize Keyboard* dialog:
  - **To specify a command**, select a category and a command.
  - **To specify a shortcut**, select a shortcut from the drop- down list.
  - **To assign the shortcut** to the command, click **Assign**.
  - **To remove the shortcut** from the command, click **Remove**.
  - **To reset all shortcuts**, click **Reset All**.
4. Click **Close**.

#### Related topics

- ❑ [Customize the Toolbar](#) (on page 18)
- ❑ [Customize the Menu](#) (on page 16)

## Customization Options and Tools

Customization allows you to configure and save a specific layout for your menus, toolbars, display settings and keyboard shortcuts. When more than one person uses the same computer, each user can create their own layout.

**Note:** In the *Options* dialog, if you set the *Window display mode* to *Multiple window views*, the last position of your windows is restored when you reopen a project. The restoration of window positions is not affected by customization. See [Pane and Data View Positioning](#) (on page 31).

### To customize tools:

1. Select **Tools > Customize**. The *Customize* dialog displays.
2. Click the **Options** tab. In this tab you can:
  - Personalize menus and toolbars
  - Select display options
3. Click the **Custom Tools** tab. In this tab, you can:
  - Create a new custom command and add its icon to a toolbar
  - Delete a custom command
  - Customize the keyboard

**Tip:** If you customize, save it so that it is available the next time you use the office software.

4. Click the **Save/Load** tab. In this tab, you can:
  - Save a new layout or load a different custom layout
  - Customize the keyboard
5. When you have made the desired changes, click **Close**.

### Related topics

- ❑ [Customize the Toolbar](#) (on page 18)
- ❑ [Customize the Menu](#) (on page 16)
- ❑ [Customize the Keyboard](#) (on page 20)

## Troubleshoot a Toolbar or Menu Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
Toolbars are not in the same language as the installation.	The program was run in one language, and then reinstalled in a different language.	Reset the toolbars to the reinstalled language by loading the default layout.  Select <b>Tools &gt; Customize</b> . In the <b>Customize</b> dialog, click <b>Save/Load</b> . In <b>Default Layout</b> , select <b>Default Layout</b> and click <b>Load</b> . Click <b>OK</b> in the <b>New layout</b> dialog.
Some text in the user interface (the units on the Status bar for instance) is in a different language.  This only occurs when opening project files that were created in a different language.	The user interface text is being stored in the project file.	There is no solution at this time. Newly created projects will not have the problem, but it will remain in the original project file.

## Find Help Topics

Use this help system to find the information you need on the concepts, procedures, and options used in the software.


**To display the help system:**

- Select **Help > Search** to find topics by entering a keyword.
- Select **Help > Contents** to browse through topics and glossary items in a table of contents.
- Select **Help > Index** to find topics and glossary items alphabetically.
- Press [F1].

**To display context-sensitive help on a specific command:**

- Press [F1] while in the specific command or dialog.
- Right-click on a command in the *Command* pane's *All Commands* list, and select **Help** from the context menu.

**To print a help topic:**

- Click the  icon.
- Right-click on a topic in the Help window, and select **Print** from the context menu. Then select **Print > Print the selected topic** from the context menu.
- To print a topic and all of its associated subtopics, select the topic, and select **Print** from the context menu. Then select **Print > Print the selected heading and all subtopics** from the context menu.

**Related topics**

- [Help Options](#) (on page 23)

## Help Options

Use these options to find information and answers on the concepts, procedures, and options covered in the help system. They are available in the help dialog.

## Tabs

### Contents

Click this tab to show all of the help topics in a tree structure. Click a chapter or topic to view it.

### Search

Type a keyword in the search box. Then, click **List Topics** to search for any occurrence of the word in the help topics. Select a topic in the search results, and click **Display** to open it. The word you search on is highlighted in each help topic. See the **Search highlight on/off** option below.

1. The **Find Setup** wizard may appear. If it does, select the **Minimize database size** option and click **Next**. Then click **Finish**.
2. In the **Type the word(s) you wish to find** box, enter the word or phrase that you want to find.
3. If necessary, in the **Select matching word(s) to narrow the search** box, enter a word to narrow the search.

In the **Choose topic to display** box, select a topic. It will be displayed in the right side of the Help window.

## Menu options

### Hide/show tabs

Click this to hide the **Contents** and **Search** tabs.

### Back

Click this to view the previous topic.

### Print

Click this to display the **Print Topics** dialog. Select an option to either print just the selected topic, or the heading and all subtopics.

### Stop

Click this to end a search function.

### Refresh

Click this to reset the display of the current page.

### Internet options

Click this to display the **Internet Options** dialog, where you can set security, privacy, content, connections, programs, and other settings for the Internet.

### Search highlight on/off

Click this to enable/disable the highlighting of each instance of the keyword in searched topics.

## Related topics

- [Find Help Topics](#) (on page 22)



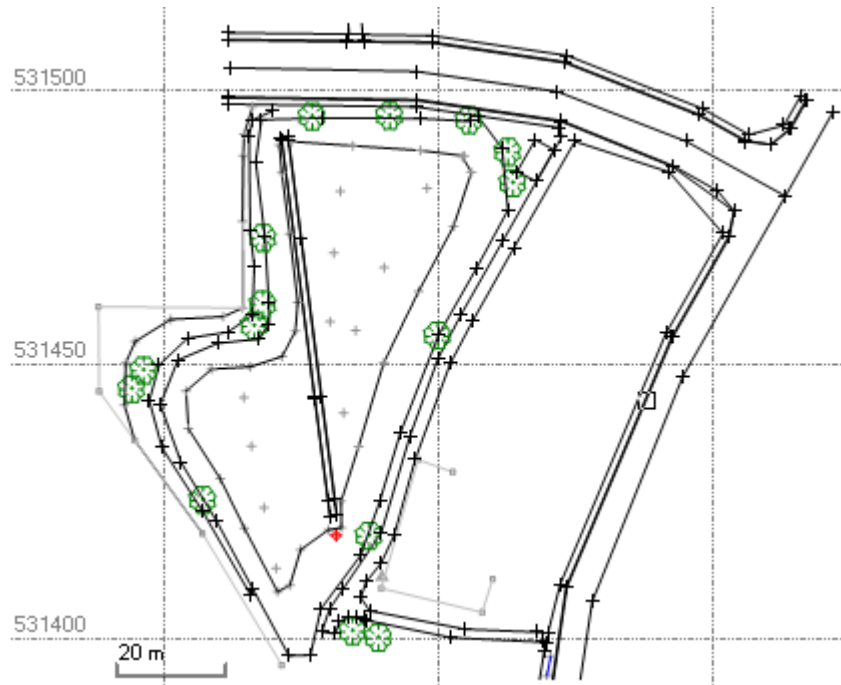
## CHAPTER 3

# View, Navigate, and Select

## Graphic Views

### Plan View

The plan view is the default view for your project data. It displays a graphical view from above, as in a map display. Multiple plan views can be opened at the same time. The data that is visible in graphics views is controlled by the [view filter](#) (see "Filter a View" on page 76).



Within the plan view you can:

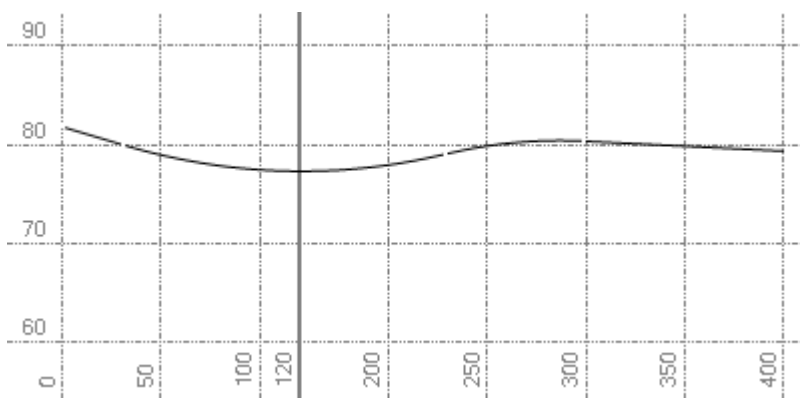
- Pan and zoom to explore the data.
- Select objects to view their properties or start a command.

### Related topics

- ❑ [2D View Navigation](#) (on page 35)
- ❑ [Data View Display Formats](#) (on page 32)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Select from the 2D Views](#) (see "Select from 2D Views" on page 44)
- ❑ [Tabbed View Arrangement](#) (on page 34)

### Profile View

Use the profile view to check the geometry of a vertical alignment. The profile view displays a vertical, graphic view of a single, specific alignment. When the alignment that the view is based on is modified or deleted, the view updates accordingly.



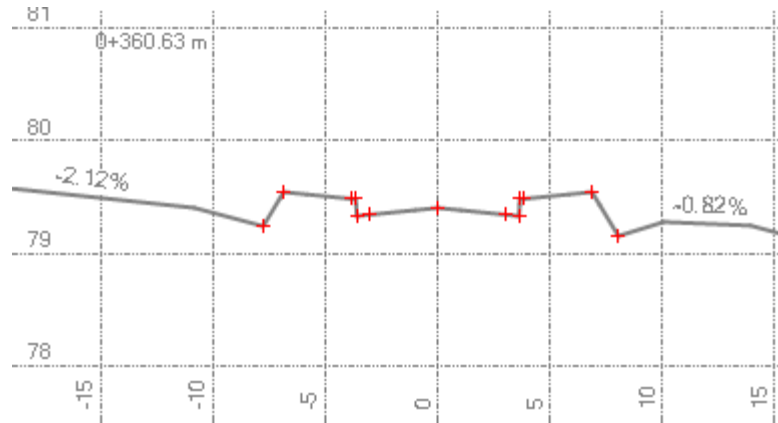
The bold vertical line at 120 denotes a station equation.

### Related topics

- ❑ [2D View Navigation](#) (on page 35)
- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Data View Display Formats](#) (on page 32)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Select from the 2D Views](#) (see "Select from 2D Views" on page 44)
- ❑ [Tabbed View Arrangement](#) (on page 34)

### Cross-Section View

Use the cross-section view to check surface cross-section geometry along a single, specific alignment anywhere it coincides with a single, specific surface. The cross-section view is a vertical, graphic view that changes depending on where you are along the alignment. When the alignment or surface that the view is based on is modified or deleted, the view updates accordingly. Multiple cross-section views can be open concurrently.



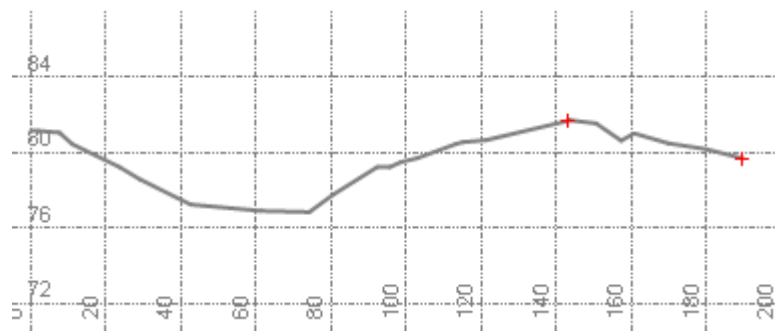
Red tic marks denote where the cross-section crosses points or breaklines. Bold vertical lines (not shown) denote station equations. At some view magnifications, the slope value is shown for each segment of the cross-section.

### Related topics

- ❑ [Create and View a Cross-Section](#) (see "Create and View a Surface Cross-Section" on page 326)
- ❑ [2D View Navigation](#) (on page 35)
- ❑ [Data View Display Formats](#) (on page 32)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Select from the 2D Views](#) (see "Select from 2D Views" on page 44)
- ❑ [Tabbed View Arrangement](#) (on page 34)

### Surface Slicer View

Use the surface slicer view to check any surface by slicing vertically through it to create a 'quick profile'. Multiple surface slicer views can be open at a time, and you can view multiple surfaces in the view concurrently.



Red tic marks denote where the surface slice crosses points or breaklines. At some view magnifications, the slope value is shown for each segment of the profile.

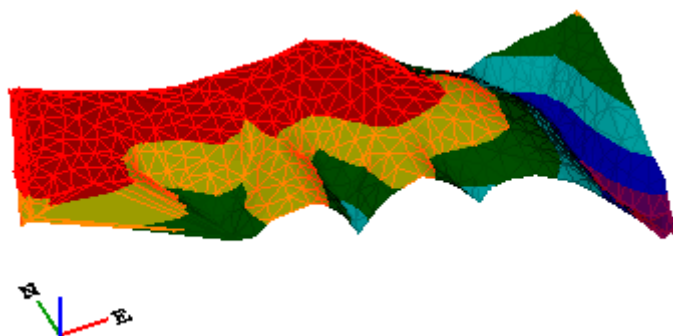
### Related topics

- ❑ [View a Slice of a Surface](#) (on page 327)
- ❑ [Create and View a Cross-Section](#) (see "Create and View a Surface Cross-Section" on page 326)

- ❑ [2D View Navigation](#) (on page 35)
- ❑ [Data View Display Formats](#) (on page 32)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Select from the 2D Views](#) (see "Select from 2D Views" on page 44)
- ❑ [Tabbed View Arrangement](#) (on page 34)

### 3D View

Use the 3D view to visualize your project data from pre-defined viewpoints, or by rotating the view. You can set a point around which the view rotates, and exaggerate the vertical scale to see changes in topography more easily using the [3D View Settings](#) (on page 38). You can also select objects in the 3D view using the standard [graphic selection methods](#) (on page 43).



The 3D view includes a compass triad with north, east, and Z axes to help you stay oriented in the view.

The performance of the 3D view may vary based on your system settings. If you have trouble, try these fixes:

1. Right-click on your desktop and select **Properties** from the context menu. The **Display Properties** dialog displays. Click the **Settings** tab. Select a different/higher setting in the **Color quality** list. Click **OK**.
2. Right-click on your desktop and select **Properties** from the context menu. The **Display Properties** dialog displays. Click the **Settings** tab. Click **Advanced**. The **Plug and Play** dialog displays. Click the **Troubleshoot** tab, and move the **Hardware Acceleration** slider to/near **Full**. Click **OK**, and **OK** again.

**Tip:** To reduce clutter in the 3D view, you can hide 2D objects or objects with a zero elevation. This can be set in **Project Settings** by selecting **Project > Project Settings**. Then click **View** and **3D View** in the left pane to access the settings.

**Note:** Microsoft® DirectX® does not work over Microsoft® NetMeeting®. The 3D view uses DirectX® and will show a blank view when attempting to use it across either communication tool.

### Related topics

- ❑ [3D View Navigation](#) (on page 37)
- ❑ [Data View Display Formats](#) (on page 32)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Select from the 3D View](#) (on page 45)
- ❑ [Tabbed View Arrangement](#) (on page 34)

## Spreadsheets and Other Views

### Points Spreadsheet




The points spreadsheet view lists the survey points in the current project, enabling you to easily edit the data. The plan view and the *Properties* pane reflect all changes made to data in the point spreadsheet view.


#### Using the spreadsheet

- **To select a point**, click in the left column for that row.



101	569588.439	949042.647	1550.108 m
101	569588.439	949042.647	1550.108 m
102	569582.094	949037.776	1549.864 m
102	569582.094	949037.776	1549.864 m
102	569582.094	949037.776	1549.864 m
103	569581.904	949046.519	1550.005 m
103	569581.904	949046.519	1550.005 m

- **To display more detail on a point** in the *Properties* pane, select the point and press [F11], or right-click and select *Properties*.
- **To edit a point's ID, coordinate, elevation, or feature code**, select it by clicking on the cell. You can also tab from cell to cell and simply type over the value in the cell.
- **To sort points based on a criteria**, click on a column heading. Up  or down  icons appear on the selected column heading, indicating the current sort order (ascending or descending).
- **To filter the point data**, click on the  icon at the top of the column and select an option from the drop-down menu.

**Note:** If the filter for a column is active, the icon  appears blue.

- **To copy data to a text editor**, such as Microsoft® Notepad, select data, and copy and paste by using the right-click menu or by pressing **[Ctrl] + C** to copy and **[Ctrl] + V** to paste. You can select all data by pressing **[Ctrl] + A**.
- **To change the order of columns** across the spreadsheet, click and drag the column heading to a new location.

### Related topics

- ❑ [Data View Display Formats](#) (on page 32)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Select from Spreadsheet Views](#) (on page 46)
- ❑ [Tabbed View Arrangement](#) (on page 34)
- ❑ [Create a Point](#) (on page 270)

## Alignment Editor

The **Alignment Editor** enables you to edit the horizontal, vertical, and stationing values of existing alignments. The graphic views reflect all changes made to alignments in the editor.

Edmondson Street Profiles : Edmondson Street										
Edmondson Street Horizontal Alignment By Segment										
Type	North Azimuth	Radius	Lt / Rt	Length	Delta	A Param	Station	North	East	
POB							-0+048.8	289423.659	392776.747	
Arc	83°18'04"	464.425	Right	29.051	3°35'02"		-0+019.8	289426.143	392805.687	
Line	80°45'24"			70.321			0+050.49	289437.439	392875.094	
Spiral	63°23'01"	370.000	Left	65.188		155.305	0+115.68	289468.332	392932.472	
Arc	58°20'10"	370.000	Left	39.451	6°06'33"		0+155.13	289490.790	392964.884	
Spiral	52°13'37"	370.000	Left	65.188		155.305	0+220.32	289533.661	393013.962	
Line	47°10'47"			296.002			0+516.32	289734.854	393231.077	
Spiral	37°49'42"	400.000	Right	60.299		155.305	0+576.62	289781.526	393269.233	
Arc	42°08'49"	400.000	Right	6.879	0°59'07"		0+583.50	289786.586	393273.893	
Spiral	43°07'56"	400.000	Right	60.299		155.305	0+643.80	289828.454	393317.266	
Spiral	47°27'03"	265.000	Left	91.018		155.305	0+734.82	289893.652	393380.605	
Arc	37°36'41"	265.000	Left	79.408	17°10'08"		0+814.23	289962.826	393418.992	
Line	20°26'33"			50.009			0+864.24	290009.686	393436.458	
*										

### Related topics

- ❑ [Edit an Alignment](#) (on page 290)
- ❑ [Data View Display Formats](#) (on page 32)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Tabbed View Arrangement](#) (on page 34)

## Report View

The **Report View** displays when you run certain reports. Along with the content of the report, the view includes a toolbar located along the top of the tab that allows you to:

- Navigate to a specific page in the report
- View and change the print setup information
- View the print layout and print the report
- Export the report to a spreadsheet or PDF document
- Select a magnification to view the report
- Search for text in the report



### Related topics

- [Tabbed View Arrangement](#) (on page 34)

## Pane and Data View Positioning

Control when and where panes and views display by pinning, floating, and docking them. Unpin panes to allow them to slide out of view when they are not being used. Pin panes to keep them open. Float views to move them around the screen for the best arrangement. Dock views to attach them to an edge of the application window.

### To pin and unpin (AutoHide) panes:

- **To pin an open pane**, click the  icon at the top of the pane.
- **To unpin a pane**, click on the  icon. If the pane is unpinned, the pane can "slide" to the side and out of view. To display the pane again, hover the cursor over the vertical tab.

### To float and dock panes and views:

By default, most panes display docked, that is, locked to one side of the application window.

- **To float a docked pane**, right-click on its titlebar and select **Floating** from the context menu. You can also click on the titlebar and drag the pane to float it.
- **To float a spreadsheet, editor, or other view**, right-click beneath the titlebar and select **Float View**. The floated view can even be dragged to an adjacent monitor if you are running dual monitors.

**Note:** Graphic views, such as the plan, 3D, and profile view, cannot be floated.

- To dock a floating pane, right-click on its titlebar and deselect **Floating**. You can also click on its title bar and drag it to a docked position along any edge of the application window.

**Note:** The docking location is determined by the position of the cursor when it intersects the edge of the application window.

- To dock a floating spreadsheet, editor, or view, right-click on the body of the view (not on its titlebar) and select **Unfloat View**.

### Related topics

- [Data View Display Formats](#) (on page 32)

## Data View Display Formats

You can control how data views are displayed, by arranging them as one or two groups of tabbed views, or as one or more tiled or cascaded views. Using multiple views let you easily view different parts of your project concurrently, and from different perspectives

### To change the data view format:

1. Select **Tools > Options**. The *Options* dialog displays.
2. Select a display format in the *Display with* box.

**Note:** There are *Window* menu options for more tabbed view options.



## Data display options

### Tabbed views (SDI)

Displays one view in the view area at a time, with tabs at the top to access additional views.

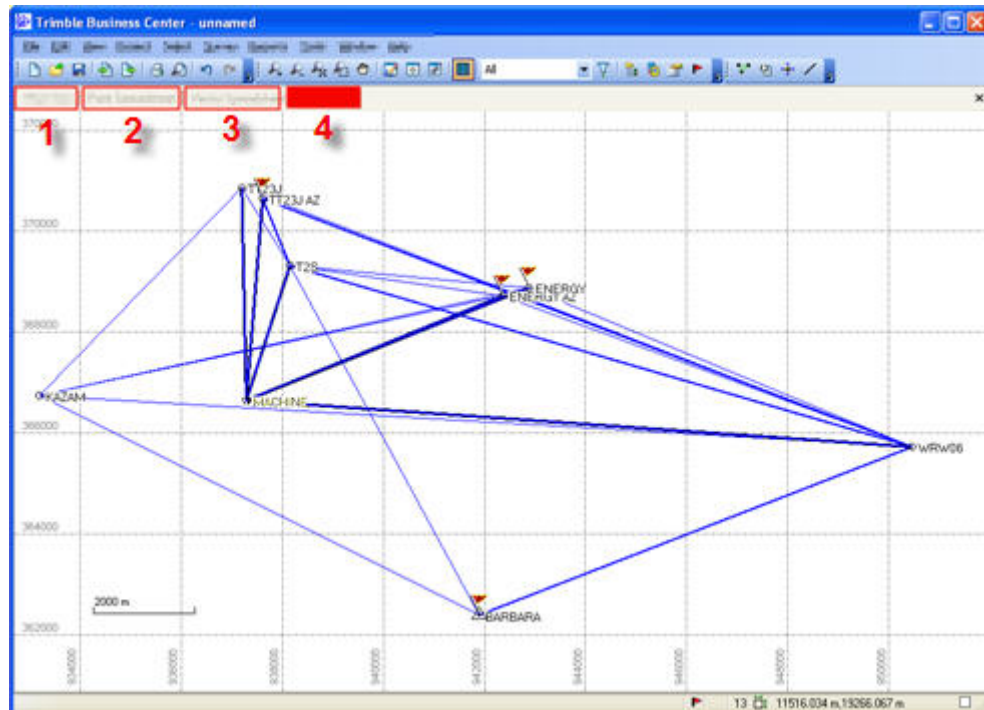
To change views, click a tab.

### Multiple views (MDI)

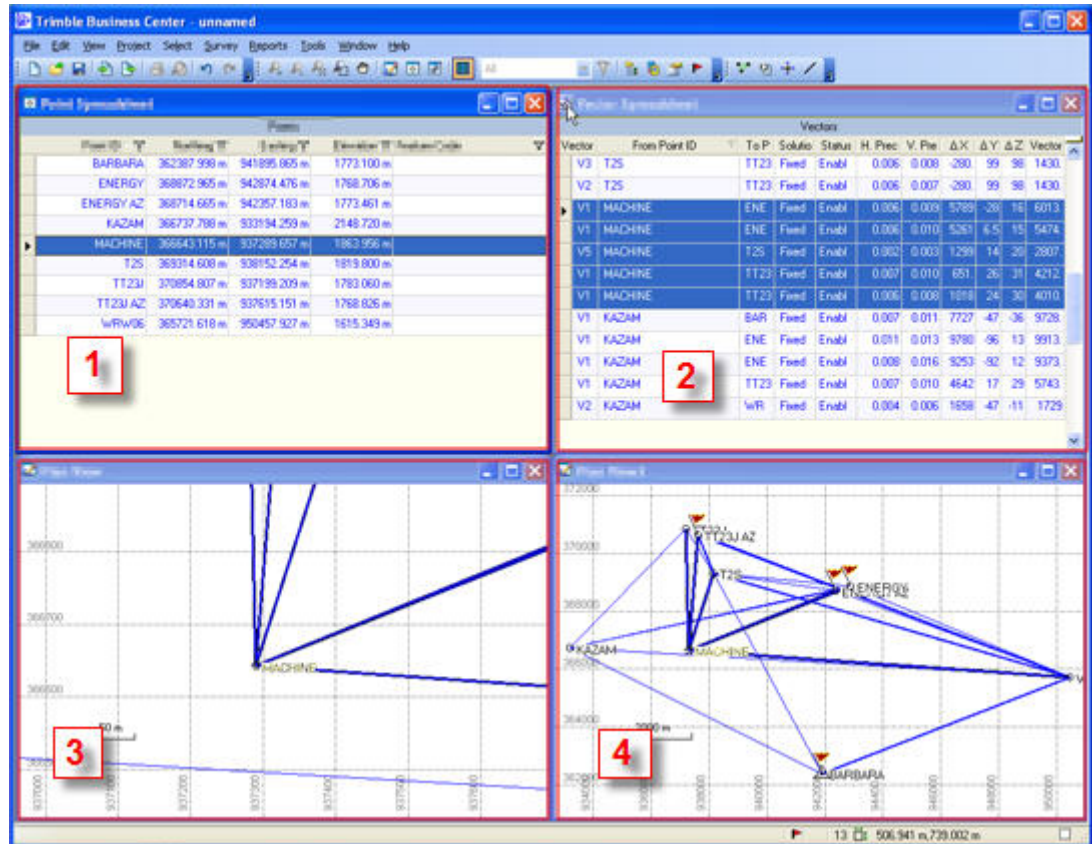
Displays one or more views at a time

- To change views, click on the title bar of the view you want; the active title label is dark blue.
- To tile or cascade the views, select **Window > (option)**.

This image displays the multiple data views displayed in a tabbed format.



This image displays the multiple data views in an MDI format.



### Related topics



- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Startup and Display Options](#) (on page 139)
- ❑ [Tabbed View Arrangement](#) (on page 34)

## Tabbed View Arrangement

When you are working with tabbed views, you can display the views in two or more groups (windows), arranged either horizontally or vertically. The command is active under these conditions:

- Data view display is set to tabbed view (SDI) format in *Options*.
- At least 2 views (tabs) are present.

#### To display tabbed views in multiple groups:

- Select **Window > New Horizontal Tab Group**.
- Select **Window > New Vertical Tab Group**.
- Click the  icon on the toolbar.
- Click the  icon on the toolbar.
- The tabbed views are divided and arranged accordingly.

You can also move tabs from one group to the other group of tabs. It is only active in data view under these conditions:

- Data view display is set to tabbed view (SDI) format in *Options*.
- At least 3 views (tabs) are present (for example, 2 plan views and a point spreadsheet view).
- The 3 views are separated into 2 groups.

#### To move a tabbed view to the next or previous group:

- Click the tab to move, and then select **Window > Move to Next Tab Group**. The tabbed view moves to the next group of tabbed views.
- Click the tab to move, and then select **Window > Move to Previous Tab Group**. The tabbed view moves to the next group of tabbed views.

#### Related topics


- [Data View Display Formats](#) (on page 32)
- [Pane and Data View Positioning](#) (on page 31)
- [Startup and Display Options](#) (on page 139)


## 2D View Navigation

Use these options and keyboard combinations to change what is displayed in the plan and alignment profile views.

#### To pan the view:


Use this to shift a different area of the screen to the center of the view.

- Click and drag the mouse wheel (or middle mouse button).
- Click the  icon on the toolbar, or select **View > Pan**. Click and drag from one point to another point in the plan view.

**Tip:** When using a laptop without a mouse, click the  (left mouse pan) on the toolbar. You can also press and hold both the left-click and right-click buttons while moving the cursor.


#### To zoom in:

Use this to display a smaller area of the plan view, in more detail.

- Click the  icon on the toolbar.
- Click in a view, and roll the mouse wheel forward.
- Select **View > Zoom > Zoom In**.


#### To zoom out:

Use this to display a larger area of the plan view, in less detail.

- Click the  icon on the toolbar.
- Click in a view, and roll the mouse wheel backwards.
- Select **View > Zoom > Zoom Out**.


#### To zoom into a certain area:

Use this to display the data within a box you draw in the view.

- Press **[Ctrl] + [Alt]**, and click and drag around an area.
- Click the  icon on the toolbar, or select **View > Zoom > Zoom**. Click and drag around the area you want to display in the view.

#### To zoom to the extents of your data:

Use this to zoom to the limits of your visible data.

- Click the  icon on the toolbar.
- Double-click the mouse wheel (or middle mouse button).
- Select **View > Zoom > Zoom Extents**.

**To center a selected point in the plan view:**

- Select one or more points in the *Project Explorer* or the plan view. Right-click, and select **Center** from the context menu.
- Select one or more points in the *Project Explorer* or the plan view, and select **View > Center**.
- Select **View > Center**. The *Center* command pane displays. Pick a point in the plan view, or type a point ID in the *Point* box.

**Related topics**

- ❑ [3D View Navigation](#) (on page 37)
- ❑ [Plan View](#) (on page 25)
- ❑ [Mouse Modes](#) (on page 42)

## 3D View Navigation

Use these keyboard and mouse combinations to change your viewpoint in the 3D view.

### 3D view

<b>Pan</b>	Click the mouse wheel (or middle mouse button) and drag.
<b>Zoom in/out</b>	Roll the mouse wheel.
<b>Zoom extents</b>	Double-click the mouse wheel (or middle mouse button).
	<hr/> <b>Note:</b> Zooming extents restores the vertical exaggeration back to its original value. <hr/>
<b>Vertical scale</b>	Press <b>[Ctrl] + [Shift]</b> , and roll the mouse wheel to exaggerate differences between elevations used in a surface.  The exaggeration value displays in the <b>Scale</b> box of the <b>3D View Settings</b> command pane. To restore the original vertical exaggeration, type 1 in the box.
<b>Rotate horizontal</b>	Press <b>[Ctrl]</b> , and roll the mouse wheel to turn the view around the X axis.  The value displays in the <b>Elevation</b> box of the <b>3D View Settings</b> command pane.
<b>Rotate vertical</b>	Press <b>[Shift]</b> , and roll the mouse wheel to turn the view around the Z axis.  The value displays in the <b>Azimuth</b> box of the <b>3D View Settings</b> command pane.
<b>Free rotation</b>	Press <b>[Ctrl]</b> , and click and drag the mouse wheel to rotate the view freely in any direction.

### Related topics

- ❑ [2D View Navigation](#) (on page 35)
- ❑ [3D View](#) (on page 28)
- ❑ [3D View Settings](#) (on page 38)
- ❑ [Mouse Modes](#) (on page 42)

## 3D View Settings

Use these settings to change your viewpoint, and the vertical scale in the 3D view. The compass triad in the bottom, left corner of the view shows the current orientation. By default, the 3D view rotates around the center of the view, which is usually the center of the bounding volume of the data in your project.

**Tip:** To reduce clutter in the 3D view, you can hide 2D objects or objects with a zero elevation. This can be changed in **Project Settings** by selecting **Project > Project Settings**. Then click **View** and **3D View** in the left pane to access the settings.

**To access pre-defined 3D views and settings:**

- Select **View > 3D View Settings**. The *3D View Settings* command pane displays.

**Options**

<b>Preset views</b>	Click an arrow icon to switch to one of nine predefined, orthographic or isometric views.  The view names refer to the direction the view is facing.
<b>Rotation</b>	Use these values to rotate the viewpoint. Entering zero in both boxes makes the view equivalent to the plan view.
<b>Elevation</b>	Drag the vertical slider, or type a value in the box, to rotate the view on the X axis.
<b>Azimuth</b>	Drag the horizontal slider, or type a value in the box, to rotate the view on the Z axis.

**Vertical scale**

<b>Scale</b>	Drag the horizontal slider, or type a value in the box, to increase the difference between elevations. The scale is the factor by which all elevations in the project are multiplied.
<b>Min</b>	Set the minimum and maximum exaggeration values in the boxes at either end of the slider.
<b>Max</b>	

**To change the point around which the view rotates:**

1. Select **View > 3D View Settings**. The *3D View Settings* command pane displays.
2. Click **Top** in the *Preset Views* group.
3. Pan the point around which you want to rotate into the center of the view.
4. Press [**Ctrl**] and roll the mouse wheel until the view is perpendicular to the plan view.

---

**Tip:** You can watch the compass triad to determine when the view is perpendicular.

---

5. Pan the point around which you want to rotate into the center of the view again.
6. Check the new rotation point by pressing [**Ctrl**], and clicking and dragging the mouse wheel to rotate the view.

**Related topics**

- ❑ [2D View Navigation](#) (on page 35)
- ❑ [3D View](#) (on page 28)
- ❑ [3D View Navigation](#) (on page 37)
- ❑ [Mouse Modes](#) (on page 42)

## Keyboard Navigation

Use the keyboard to navigate the application and perform tasks if it is easier for you than using a mouse.

### Shortcut keys

- |             |   |
|-------------|---|
| <b>[F1]</b> | Displays the Help window.   |
| <b>[F2]</b> | Toggles grid cells in spreadsheets and dialogs between editable and uneditable. When a cell is editable, the arrow keys move the insertion point within the cell. When a cell is uneditable, the arrow keys move the focus from cell to cell. |
| <b>[F4]</b> | Computes the current project.   |
| <b>[F5]</b> | Switches the left mouse button to select mode.  |
| <b>[F6]</b> | Switches the left mouse button to rotate mode.  |
|             | <hr/> <b>Note:</b> This mode only works in 3D views. If you select this mode in 3D view, it reverts to the Select mode when in plan view. <hr/>   |
| <b>[F7]</b> | Switches the left mouse button to pan mode.   |
| <b>[F8]</b> | Switches the left mouse button to zoom mode.  |
|             | Displays or hides the <i>Project Explorer</i> .   |
| <b>[F9]</b> |   |



**[F10]** Displays or hides the *Device* pane.

**[F11]** Displays the *Properties* pane.

**[F12]** Displays or the *Command* pane.

---

**Note:** **[F11]** and **[F12]** toggle between the *Command* and *Properties* panes.

---

### Other keys

<b>[Enter]</b>	For commands with an <b>OK</b> button, this initiates the command.
<b>[Esc]</b>	When the command pane has focus, this clears the most recent command from the stack.  In the <i>Properties</i> pane, this cancels an edit, reverting the property to its original value.
<b>[Tab]</b>	Accepts the current value, and advances to the next control or button.
<b>[Shift] + [Tab]</b>	Accepts the current value, and moves to the previous control or button.
<b>[Space]</b>	Accepts the highlighted button. Opens advanced dialogs when you have tabbed into certain controls: <ul style="list-style-type: none"> <li>▪ Color list - Opens the Color dialog.</li> <li>▪ Layer list - Opens the New Layer dialog.</li> <li>▪ Line Style list - Opens the Line Style Manager.</li> <li>▪ Select box - Opens the Advanced Select command pane.</li> </ul>
<b>[Ctrl] + [Tab]</b>	Lets you select any of the active views, panes, or commands.
<b>[Ctrl] + [Shift] + [Tab]</b>	Lets you select any of the active views, panes, or commands.
<b>[Ctrl] + [↑] or [↓]</b>	Navigates within groups.  Press <b>[Ctrl] + [↑]</b> to move up to the previous folder in <i>Project Settings</i> (and similar lists).
<b>[Ctrl] + [←] or [→]</b>	Expands or contracts groups in the <i>Project Explorer</i> and the <i>Properties</i> pane.
<b>[Shift] + [F10]</b>	Displays the context menu when you are in a control (box).
<b>[Ctrl] + [D]</b>	Deletes selected objects from a project.

### Related topics

- [2D View Navigation](#) (on page 35)

- ❑ [3D View Navigation](#) (on page 37)
- ❑ [Mouse Modes](#) (on page 42)

## Mouse Modes

Activate different modes to control what the left-mouse button does in graphic views. These modes are essential when you are using a laptop, or a mouse with no wheel/center button. They are available on the mouse toolbar.

### Options



Click this to pick objects when you click or click and drag the left mouse button. You can also press **[F5]** to switch to the select mouse mode.



Click this to rotate the view when you click and drag the left mouse button. You can also press **[F6]** to switch to the rotate mouse mode.

---

**Note:** This mode only works in 3D views. If you select this mode in 3D view, it reverts to the select mode when in plan view

---



Click this to move the view in a planar way when you click and drag the left mouse button. You can also press **[F7]** to switch to the pan mouse mode.



Click this to zoom in or out when you click and drag the left mouse button. You can also press **[F8]** to switch to the zoom mouse mode.

### Related topics

- ❑ [2D View Navigation](#) (on page 35)
- ❑ [3D View Navigation](#) (on page 37)
- ❑ [Graphic Selection Methods](#) (on page 43)

## Data Selection

Select objects by picking them in graphic views or spreadsheets, selecting menu options, or using keyboard combinations. Objects that you select highlight in graphic views, spreadsheets, and the **Project Explorer**, depending on which you have open. The number of objects selected appears on the status bar at the bottom of the application window.

---

**Note:** To set the visibility and selectability of objects, create a new view filter in the [View Filter Manager](#) (see "Filter a View" on page 76).

---

### Related topics

- ❑ [Select from the 2D Views](#) (see "Select from 2D Views" on page 44)
- ❑ [Select from the 3D View](#) (on page 45)
- ❑ [Select from Spreadsheet Views](#) (on page 46)
- ❑ [Select from the Project Explorer](#) (on page 48)
- ❑ [Selection Methods and Options](#) (on page 43)

## Selection Methods and Options

Use these options to choose a selection method. They are available in the **Select** menu and through the **Options** button, which is available in various commands. Objects set to visible and selectable in the current view filter can be selected.

**Note:** The number of objects currently selected displays on the status bar at the bottom of the window.

### Options

<b>Select All</b>	Use this to select all of the visible and selectable objects (as set in the current view filter) in the views and spreadsheets. You can also press <b>[Ctrl] + [A]</b> to select all objects.
<b>Invert Selection</b>	Use this to deselect the currently selected objects, and select the currently unselected objects visible in the view.
<b>Select Points</b>	Use this to select points with specific properties.
<b>Select Duplicate Point Identifiers</b>	Use this to select points with the same point IDs, often prior to merging duplicate points.
<b>Select Observations</b>	Use this to select survey observations with specific properties.
<b>Select Unprocessed Sessions</b>	Use this to select all sessions for which baselines have not been processed.
<b><a href="#">Select by Elevation Range</a> (on page 52)</b>	Use this to select data within, or outside of, a specific elevation range.
<b><a href="#">Select by Layer</a> (on page 53)</b>	Use this to select data by its layer.
<b><a href="#">Advanced Select</a> (see "Select Using Advanced Criteria" on page 53)</b>	Use this to select data from the entire data set using a custom set of criteria.

### Related topics

- ❑ [Select from the 2D Views](#) (see "Select from 2D Views" on page 44)
- ❑ [Select from the 3D View](#) (on page 45)
- ❑ [Select from Spreadsheet Views](#) (on page 46)
- ❑ [Select from the Project Explorer](#) (on page 48)
- ❑ View Filter Manager Command

## Graphic Selection Methods

Select objects by clicking and dragging in graphic views in these ways:

## Options

- Windowing** Click and drag a box from left to right. All of the objects **entirely within the box** are selected.
- Crossing** Click and drag a box from right to left. All of the objects **within the box and crossed by the box** are selected.

In graphic views, there are also two context menu options that you can access by right-clicking when you have data selected.

## Options

- Previous selection** Select this to clear the current selection and reselect the objects last selected.
- Clear selection** Select this to deselect all objects.

The appearance of the cursor will change in graphic views, depending on the control you are using. When you are using a COGO control, the name of the control will also appear on the status bar. The status line's tooltip lists ways in which you can use the control.

## Cursor types



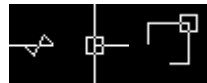
Pick any point.



Pick any point or applicable object (arc, line, segment, point).



Pick any applicable object.



No anchor point has been defined.



The pick function is unavailable, or the object you are trying to pick is invalid. Often, you can click a blank space in the **Project Explorer** to refresh the pick cursor for the graphic views.

## Related topics

- ❑ [Select from the 2D Views](#) (see "Select from 2D Views" on page 44)
- ❑ [Select from the 3D View](#) (on page 45)
- ❑ [Selection Methods and Options](#) (on page 43)

## Select from 2D Views

Select objects in the plan and profile views using these standard methods, or graphic selection methods.

**To select all objects:**

- Select **Select > Select All**.
- Press **[Ctrl] + [A]**.

**To select individual objects:**

- Move the cursor over an object in the view and click. If there is more than one object within the pick aperture, a list of objects from which you can select appears. Select the object you need.
- Click the name of the object in the *Project Explorer*.

**To select a group of points or observations:**

- **[Ctrl] + click** - to add an object to the selection set
- **[Shift] + [Ctrl] + click** - to remove an object from the selection set
- **Click and drag (from left to right)** - to select objects within the window.
- **Click and drag (from right to left)** - to select all objects within or crossed by the window.

**To undo a selection:**

- Right-click and select **Clear Selection** from the context menu - the last object selected with be removed from the selection.
- Click any blank space in a graphic view. This deselects everything.

**To clear the selection:**

- Right-click the selected point or group of points and select **Clear Selection**.
- Click in an empty area of a graphic view.

**Related topics**

- ❑ [2D View Navigation](#) (on page 35)
- ❑ [Graphic Selection Methods](#) (on page 43)
- ❑ [Filter a View](#) (on page 76)

**Select from the 3D View**

Select objects in 3D views using these standard methods, or graphic selection methods.

**To select all objects:**

- Select **Select > Select All**.
- Press **[Ctrl] + [A]**.

**To select individual objects:**

- Move the cursor over an object in the view and click. If there is more than one object within the pick aperture, a list of objects from which you can select appears. Select the object you need in the list.
- Click the name of the object in the *Project Explorer*.

**To select a group of points or observations:**

- **[Ctrl] + click** - to add an object to the selection set.
- **Click and drag (from left to right)** - to select objects within the window.
- **Click and drag (from right to left)** - to select all objects within or crossed by the window.
- **[Shift] + [Ctrl] + click** - to remove an object from the selection set.

**To undo a selection:**

- Right-click and select **Clear Selection** from the context menu. The last object selected is removed from the selection set.
- Click any blank space in a graphic view.

**To clear a selection:**

- Right-click the selected point or group of points and select **Clear Selection**.
- Click in an empty area of a graphic view.

**Related topics**

- ❑ [3D View](#) (on page 28)
- ❑ [3D View Navigation](#) (on page 37)
- ❑ [3D View Settings](#) (on page 38)
- ❑ [Graphic Selection Methods](#) (on page 43)

**Select from Spreadsheet Views**

To select individual or multiple points or vectors in a [spreadsheet](#) (see "Points Spreadsheet" on page 29) view, use the mouse. Selected objects also highlight in graphic views, and the *Project Explorer*. The number of selected points or vectors appears on the status bar at the bottom of the application window. Edits in a spreadsheet cell are reflected in the *Properties Pane* after you exit from the cell.

**To select a single point or vector:**

- Click the gray box on the left of the row:



101	569588.439	949042.647	1550.108 m
101	569588.439	949042.647	1550.108 m
102	569582.094	949037.776	1549.864 m
102	569582.094	949037.776	1549.864 m
102	569582.094	949037.776	1549.864 m
103	569581.904	949046.519	1550.005 m
103	569581.904	949046.519	1550.005 m

**Note:** When you edit northing, easting, and elevation values in a point spreadsheet view, the quality of the point is upgraded to **Control**.

**To select multiple points or vectors:**

- To select a series of rows, click the first row in the series, press **[Shift]**, and click the last row in the series. All rows in-between are selected.
- To select multiple, separate rows, press **[Ctrl]** + click on each individual row to add to the selection.
- To select all rows, right-click anywhere in the spreadsheet and select **Select All** from the context menu.

**Note:** To edit the feature code for multiple points, use the **Properties** pane.

**To undo a selection:**

- Click any cell.
- Right-click and select **Undo Selection** from the context menu - the last object selected will be removed from the selection set.

**To delete a selected row:**

- Right-click the selected row and select **Delete** from the context menu.

**Related topics**


- ❑ [Point Spreadsheet](#) (see "Points Spreadsheet" on page 29)
- ❑ Select Observations
- ❑ [Select Points](#) (on page 49)
- ❑ Select Unprocessed Sessions
- ❑ Vector Spreadsheet


## Select from the Flags Pane

Select one or more objects from the **Flags** pane if the objects have been flagged with import or computation errors. If there are no objects in the **Flags** pane, no objects have been identified as having errors.

**Tip:** The number of objects selected appears on the status bar at the bottom of the application window.

### To display the Flags pane:

- Select **View > Flags Pane**.
- Click the  icon (appears in the toolbar and on the status bar)

**Note:** Press **[F11]** or click the  icon to display the **Properties** pane. Within the properties dialog, you can edit the errors. Click on a data object in the **Flags** pane to display its properties.

### To select flagged objects:

- To select an individual object, click on the left edge of the row.
- To select multiple objects, do one of the following:
  - Click the first row of the series, hold **[Shift]**, and click the ending row of the series.
  - Hold **[Ctrl]** and click individual rows to add these rows to the selection.

The selected points now appear highlighted in the plan view.

### Related topics

- ❑ Compute Project Command
- ❑ [Flags Pane](#) (on page 14)
- ❑ [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)

## Select from the Project Explorer

The **Project Explorer** displays your project data organized in a tree structure. The number of objects selected appears on the status bar at the bottom of the application window.

### To select objects:

- To select an individual object, click on the object.
- To select multiple objects, do one of the following:
  - Click the first object, hold **[Shift]**, and click the ending object of the series.



- Hold **[Ctrl]** and click individual objects to add these objects to the selection.

The selected points appear highlighted in the graphic and spreadsheet views.


### Related topics

- ❑ [Project Explorer](#) (on page 7)
- ❑ [Properties Pane](#) (on page 14)

## Select Points

Select individual, multiple, ranges, or sets of points based on specific criteria.

### To select points:

1. Do one of the following:
  - Select **Select > Select Points**.
  - Click the  icon on the toolbar.
  - Click **Options** next to a select box in a command, and select **Select Points** from the drop-down list.

The **Select Points** command pane displays. Each tab (*General*, *GPS*, and *Occupation*) shows a subset of possible selection criteria.

1. Set the selection criteria you wish to use to select points. You can set multiple criteria and check more than one box in each group.
  - To add points selected to the current selection, check the **Add to current selection** box.
  - To preview the selection in an open graphic view, *Project Explorer*, or points spreadsheet, click **Apply**. All points meeting all of the criteria are selected.
2. Refine the criteria if needed, and click **OK** to make the selection and close the dialog. The number of selected points appears on the status bar.

## Options

### General tab

#### Point ID

To select a single point, enter the point ID.

To select a range of points, enter the ID of the first point in the range followed by three dots (...) and the ID of the last point in the range. For example, you would enter 1 . . . 5 to select points 1, 2, 3, 4, and 5.

To select two or more non-contiguous points, enter the ID of each point separated by a comma. For example, you would enter 1, 3, 6 to select points 1, 3, and 6.

Note that you can enter multiple ranges by separating each range with a comma. For example, you could enter 1 . . . 5, 101 . . . 105.

---

**Note:** Alpha characters used in point IDs are not case sensitive.

---

#### Feature code

Enter the abbreviation you have given to a feature associated with the points you want to select.

#### Observed from

Enter the name of a point from which the points you want to select were observed.

#### Layer

Select the layer that the points you want to select reside on.

#### Coordinate quality (horizontal, elevation, height)

**Fixed in adjustment** - Check this box to select control quality points that were designated as 'fixed' during the last network adjustment.

**Adjusted** - Check this box to select points with final coordinates resulting from the last network adjustment.

**Control** - Check this box to select NGS surveyed coordinates of the highest quality.

**Survey** - Check this box to select surveyed coordinates of the second highest quality.

**Mapping** - Check this box to select coordinates of the low to average quality.

**Unknown** - Check this box to select coordinates of the lowest or unverified quality.

### GPS tab - Points observed via GPS vectors

#### Horizontal/ vertical precision

Select operators, and enter precisions in the format shown in the **Properties** pane for vectors in your project.

<b>Solution type</b>	<p><b>Code</b> - Check this box to select points with autonomous positions.</p> <p><b>Fixed</b> - Check this box to select coordinates for which the baseline processor was <b>able</b> to resolve the integer ambiguity with enough confidence to select one set of integers over another during baseline processing.</p> <p><b>Float</b> - Check this box to select coordinates for which the baseline processor was <b>unable</b> to resolve the integer ambiguity search with enough confidence to select one set of integers over another.</p>
<b>Field method</b>	<p><b>Continuous</b> - Check this box to select points observed during a continuous trajectory.</p> <p><b>Event</b> - Check this box to select points observed in a Real-Time Kinematic (RTK) GPS mode in which the resulting vectors include an 'event' marker.</p> <p><b>Observed control</b> - Check this box to select control points observed in an RTK GPS mode.</p> <p><b>Rapid</b> - Check this box to select points observed in an RTK GPS mode was collected at a fast data rate, such as every second.</p> <p><b>Static or fast static</b> - Check this box to select points observed in a PPK GPS mode collecting up to 20 minutes or several hours (respectively) of raw data, and then postprocessing to achieve sub-centimeter precisions.</p> <p><b>Stop and go</b> - Check this box to select points observed in short PPK or RTK Stop and Go occupations, while maintaining lock, and then postprocessing to achieve centimeter precisions.</p> <p><b>Topo</b> - Check this box to select points observed in a GPS survey mode defined as topographic.</p>

#### Occupation tab

<b>Antenna heights between</b>	Enter antenna height parameters in the format shown in the Properties pane for points in your project.
<b>Add to current selection</b>	Check this to add the results of current selection to any previously selected data.

#### Related topics

- Select Observations
- Select Unprocessed Sessions
- [Selection Methods and Options](#) (on page 43)

## Select Duplicate Points

Select all of the points with duplicate IDs in your project if you need to review or merge them into single points. Merge duplicate points if you know that they are the same physical point, and you do not need them to be separate points.

### To select points with duplicate IDs:

1. Select **Select > Select Duplicates**. The *Select Duplicate Points* command pane displays.
2. If you do not want to set a distance tolerance between the points selected, uncheck *Within the following distance*.
3. Otherwise, click two points in a graphic view, or type a distance in the box. Points within the distance will be selected as duplicates, depending on what you set in the next group.
4. Select an option to ignore point IDs, or include them only if they are identical or different.
5. Click **Apply** if you want to see and refine the selection, or click **OK** to use the current selection criteria. All points meeting all of the criteria are selected. The number of selected points appears on the status bar.

### Related topics

- ❑ [Merge Duplicate Points](#) (on page 278)
- ❑ [Selection Methods and Options](#) (on page 43)

## Select by Elevation Range

Use an elevation range to select data, such as a set of surveyed points, that lies between high and low points in your project. You can also select objects, such as surfaces, that cross the elevation range you specify.

### To select data using an elevation range:

1. Select **Select > Select by Elevation Range**. The *Select by Elevation Range* command pane displays.
2. Click in the *Max elevation* box, and pick a point in a graphic view to use its elevation, or type an elevation in the box.
3. Click in the *Min elevation* box, and pick a point in a graphic view, or type an elevation in the box.
4. Specify whether to include only data that falls completely within the range, or data that falls within or crosses the range.
5. If you want to add consecutive selections to your selection set, leave *Add to current selection* checked.

6. Click **Apply** if you want to see and refine the selection, or click **OK** to use the current selection criteria. All data meeting all of the criteria are selected. The number of selected points appears on the status bar.

**Note:** Check the *Project Explorer* to easily see what you have selected.

### Related topics

- [Selection Methods and Options](#) (on page 43)

## Select By Layer

Select data that resides on specific layers in your project.

### To select data by its layer:

1. Select **Select > Select by Layer**. The *Select by Layer* command pane displays.
2. Simply check or uncheck the boxes next to any layer names in the list. The graphic views and *Project Explorer* update in real-time to show the selected data.
3. Click **Close**.

### Related topics

- [Create and Edit a Layer](#) (on page 68)
- [Selection Methods and Options](#) (on page 43)

## Select Using Advanced Criteria

Select objects by specifying a type of data (baseline, coordinate, point, etc.) and a specific property of that data. If needed, continue to build or modify your selection by specifying additional data types and their properties.

### To select data using specific criteria:

1. Select **Select > Advanced Select**. The *Advanced Select* dialog displays, showing your current selection in the *Current Status* group.
2. In the *Apply This Selection To* group, specify whether to select from the currently selected objects or from all data, and whether to replace or add to the current selection.
3. Designate the kind of objects you want to select in the *Data type* list.
4. To narrow your selection by specifying a property, click *Data with the following property* and select a property in the list. Otherwise, click **Apply** to preview the results, or **OK** to make the selection and close the *Advanced Select* dialog.
5. If you are specifying a property of the data, select a mathematical operator in the *That is* box. Then type or select a value in the *This value* box.

**Note:** If you are entering a duration in the *This value* box, it must be in the time format used by your computer's operating system, which is likely HH:MM:SS (Hours:Minutes:Seconds).

6. To select the inverse of the criteria you specify, check the ***Invert selected objects*** box. This deselects all objects that would have been selected and selects all of the previously unselected objects.
7. Click **Apply** to preview the results, or **OK** to make the selection and close the **Advanced Select** dialog.

### Related topics



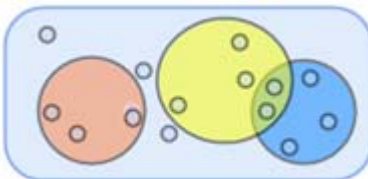
- ❑ [Selection Methods and Options](#) (on page 43)

## Selection Explorer and Selection Sets

### Understanding Selection Sets

If you need to select the same objects over and over, it can be frustrating to manually do it and it is easy to make mistakes. Instead, use the **Selection Explorer** to create, modify, and reuse selection sets.

Selection sets can be created and modified in a variety of ways, including Boolean operations using two or more sets.

Action	Selection Explorer Command	Resulting Sets
Project with data		
Group project data into selection sets.	 <b>Save as...</b> <ul style="list-style-type: none"> <li>▪ Red(3)</li> <li>▪ Yellow(5)</li> <li>▪ Blue(5)</li> </ul>	

Add currently selected objects to a selection set.

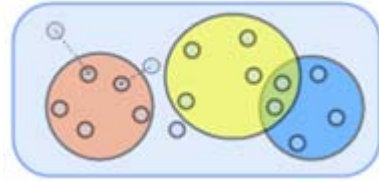
 **Add Current Selection**

current selection(2)

+ Red(3)

--> Red (5)

**Note:** Don't forget to **Save as**.



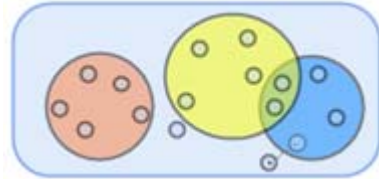
Subtract currently selected objects from a selection set.

 **Subtract Current Selection**

Blue(5) – current selection (1)

--> Blue (4)

**Note:** Don't forget to **Save as**.

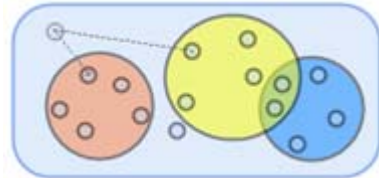


Add objects to a more than one selection set.

Pick current selection(1)

**Add To** Red(4), Yellow(5)

--> Red(5), Yellow(6)



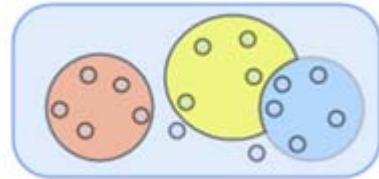
Subtract a selection set from another selection set.

**Select** Blue(5)

**Subtract From** Yellow(6)

--> Yellow(4)

**Note:** Only the two objects that Blue and Yellow have in common are subtracted from Yellow.

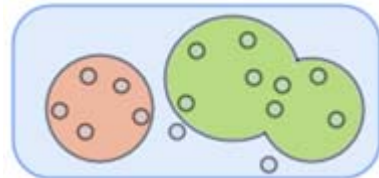


Create a new set from two or more selection sets.

**Select** Yellow(6), Blue(5)

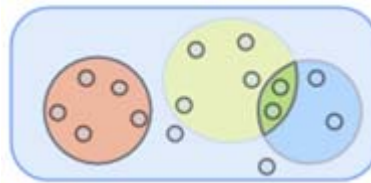
**Save as** Green(8)

**Note:** Use [Ctrl] or [Shift] to multi-select selection sets from the Selection Explorer list.



Create a selection set of only the common objects from two or more sets.

**Select** Yellow(6), Blue(5)  
**Save Common Items as**  
 Green(8)




## Selection Explorer

The **Selection Explorer** is a pane that lists the selection sets in your project in the top section and lists the objects in the active set in the bottom section.


- When you click on a selection set, all the objects in the set are selected in the project.
- When you select one or more objects in the list of objects, those objects are selected in the project.
- When **<Selection Snapshot>** is active, objects selected in the graphics area are listed.

Using selection sets makes accessing and selecting groups of commonly-used objects faster and more consistent.

### To display and pin the Selection Explorer:

1. Do one of the following:
  - Select **View > Selection Explorer**.
  - Select **Select > Selection Set > Selection Explorer**.
  - Click the  icon on the toolbar.
  - Right-click in a graphic view and select **Selection Explorer** from the context menu.

The **Selection Explorer** displays, docked on the left side of the application window, or where you positioned it last.

2. If desired, pin the explorer open by clicking the  icon at the top. If the pane is unpinned, the pane can "slide" to the side and out of view. To show it again, click the **Selection Explorer** tab.

### Related topics

- ❑ [Understanding Selection Sets](#) (on page 54)
- ❑ [Create and Use Selection Sets](#) (on page 57)
- ❑ [Modify Selection Sets](#) (on page 58)
- ❑ [Pane and Data View Positioning](#) (on page 31)



## Create and Use Selection Sets


Use the **Selection Explorer** to create, modify, and reuse selection sets for easier selection and editing. Using selection sets makes accessing and selecting groups of commonly-used objects faster and more consistent.

The **Selection Explorer** contains two lists: the **Selection Sets** list at the top of the pane, and the object list at the bottom of the pane. The object list shows the objects that are contained in the selected selection set(s).


**Note:** The selection sets you create, and those created from some types of imported files, also appear in the **View Filter Manager** (on page 9). They can be used as a starting point in creating view filters.

### To create a temporary list of the currently selected objects:

1. Open the **Selection Explorer**.
2. Click **<Selection Snapshot>** on the list of selection sets.
3. Select objects that you want to list in a graphic view, spreadsheet view, or the **Project Explorer**. This list of objects displays in the object list of the Selection Explorer.

**Note:** If the list does not update automatically, click the  (**Refresh**) icon in the Selection Explorer.

### To create a selection set (with the Selection Explorer open):

1. Select objects that you want to include in the set in a graphic view, spreadsheet view, or the **Project Explorer**.
2. Click the  icon on the **Selection Explorer's** toolbar. The **Save As** dialog displays.
3. Type a name for the selection set, and click **OK**. The name of the new set appears in the **Selection Sets** list.

### To create a selection set (from the menu):

1. Select objects that you want to include in the set in a graphic view, spreadsheet view, or the **Project Explorer**.
2. Select **Select > Selection Set > Save As**. The **Save As** dialog displays.
3. Type a name for the selection set, and click **OK**.

**Note:** When you open the **Selection Explorer**, the selection set displays in the **Selection Sets** list.


**To select all objects in a selection set:**

1. Open the *Selection Explorer*.
2. Select one or more sets in the *Selection sets* list. All of the objects in the sets are selected in your project.

**Note:** Use [Ctrl]-click and [Shift]-click to multi-select in the *Selection Sets* list.

3. Deselect, modify, and use the objects as needed.

**To copy a selection set:**

1. Open the *Selection Explorer*.
2. Select the selection set to copy in the *Selection sets* list.
3. Click the  icon on the *Selection Explorer's* toolbar. The *Save As* dialog displays.
4. Type a new name for the selection set, and click **OK**.


**To remove a selection set (from the menu):**

1. Select **Select > Selection Set > Remove**. The *Remove* dialog displays.
2. Select one or more sets to remove in the list, and click **OK**.

**Note:** *Remove* does not delete the objects in the selection set from your project. If you delete objects from your project, they are also removed from any selection sets they were in.

**Note:** <Selection Snapshot> is a default selection set that cannot be removed.


**To remove a selection set (with the Selection Explorer open):**


1. Select one or more sets to remove in the *Selection Sets* list
2. Click the  icon. The selection set disappears from the list.

**Related topics**



- ❑ [Selection Explorer Options](#) (on page 60)
- ❑ [Understanding Selection Sets](#) (on page 54)
- ❑ [View Filter Manager](#) (on page 9)

**Modify Selection Sets****To add objects to an existing selection set (from the Selection Explorer):**

1. Open the *Selection Explorer* and select the selection set from the *Selection Sets* list.
2. Select the objects to add.
3. Click  icon on the *Selection Explorer*. The objects appear in the list below.

4. (optional) Click  to save the updated selection set.

#### **To subtract objects to an existing selection set (from the Selection Explorer):**

1. Open the *Selection Explorer* and select a selection set from the *Selection Sets* list.
2. Select the objects to subtract.
3. Click  (or right-click and select **Subtract Current Selection** from the context menu). The objects are subtracted from the set.
4. (optional) Click  to save the updated selection set.

#### **To add objects to an existing selection set (from the menu):**

1. Select objects that you want to add to a set.
2. Select **Select > Selection Set > Add To**. The *Add To* dialog displays.
3. Select the set to edit in the list and click **OK**. The objects are added to the set.

#### **To subtract objects from an existing selection set (from the menu):**

1. Select objects that you want to remove from a set.
2. Select **Select > Selection Set > Subtract From**. The *Subtract From* dialog displays.
1. Select the set to edit in the list and click **OK**. The objects are removed from the set.

#### **To edit multiple selection sets at once:**

1. Open the *Selection Explorer*.
2. Select one or more sets in the *Selection sets* list. All of the objects in the sets are selected in your project.

**Note:** Use [Ctrl]-click and [Shift]-click to multi-select in the *Selection Sets* list.

3. Select objects that you want to add; then right-click and select **Add Current Selection**.
4. Select objects that you want to remove in the objects list; then right-click and select **Subtract Current Selection**.

#### **To combine multiple selection sets:**

1. Open the *Selection Explorer*.
2. Select each of the selection sets that you want to combine in the *Selection Sets* list.

**Note:** Use [Ctrl]-click and [Shift]-click to multi-select in the list.

3. Click the  icon on the *Selection Explorer's* toolbar. The **Save As** dialog displays.

4. Type a name for the new selection set, and click **OK**. The name of the combined set appears in the *Selection Explorer*.

**Note:** Objects that were in more than one set are not duplicated in the new combined set.

### Related topics

- [Selection Explorer Options](#) (on page 60)
- [Understanding Selection Sets](#) (on page 54)

### Selection Explorer Options

Use these *Selection Explorer* options to create, edit, and use selection sets.

## Options



(save the selection list as)

## Context menu

**Save as**

The icon and the context menu option work in slightly different ways:

Click the icon to display the **Save Selection List As** dialog, where you can save all of the objects in the **Selection List** as a new selection set, regardless of whether they are selected or not.

Select the context menu option to display the **Save As** dialog, where you can save just the objects that are selected in the **Selection List** as a new selection set.

Use **Save As** to copy and combine selection sets as well. You can also enter an existing set name to overwrite that set with the current selection.



**Refresh**

Click this to refresh the **Selection Explorer's** list of currently selected objects after you have selected or deselected objects in another view, spreadsheet, or the **Project Explorer**.



**Properties**

Click this to display the **Properties** pane, where you can view and edit the properties common to all the objects in the current selection set.

---

**Tip:** Using the <**Selection Snapshot**>, you can select all of the objects you need to modify and then view and edit properties for each one without losing the others in the list.

---



**Remove**

Click this to remove the selected selection sets.

---

**Note:** This does not delete the members of the selection set from your project.

---



**Add Current Selection**

Click this to add newly selected objects to the current selection set.



**Subtract Current Selection**

Click this to subtract newly selected objects from the current selection set.

**Selection sets**

This column lists the <**Selection Snapshot**> (always present), selection sets automatically generated by some types of imported files, and selection sets you have created.

---

**Note:** Selection sets also appear in the **View Filter Manager** (on page 9). They can be used as a starting point in creating view filters.

**Note:** Selection sets can contain objects that have no visible display, such as coordinates. In the **Selection Explorer**, all selection sets are available. In the **View Filter Manager**, however, only selection sets that contain at least one visible object are available.

---

**Objects**

This column shows the number of objects that are members of the selection set.

**Selection List**

This list shows the members of the currently selected set.



Click this to sort the set by selection status, either 'selected' or 'unselected'.

This column also denotes the last object selected with the ► icon, and displays the icon for each member's object type.

**Name**

Click this column heading to sort the members of the set by object name.

Unnamed objects are shown with '?' for the name, but they can be partially distinguished by the object type icon in the next column.

**Type**

Click this column heading to sort the objects in a set by their object type.

**Additional options (context menu)**

Right-click on any selection set or object to access these options. Options vary, depending on what is selected.

**Save Common Objects As** - Select this to display the **Save As** dialog, where you can create a new selection set containing all of the objects that were in each of the selected sets.

**Lock/Unlock Selection Sets** - Select these to prevent or allow changes to the selected sets.

---

**Note:** Locked selection sets can only be changed by deleting objects from the project.

---

**Select all** - Select this to select every object in the selected selection set.

**Select This Type** - Select this to select all the objects in the set whose type (such as 'point') matches the selected object.

**Invert** - Select this to switch the selection state in the object list. This deselects the currently selected objects in the set, and selects the currently unselected objects.

**Add To** - Select this to display the **Add To** dialog, where you can select one or more sets to which you want to add the selected objects. If you add objects that already exist in the set, they are not duplicated.

**Subtract From** - Select this to display the **Subtract From** dialog, where you can select one or more sets from which you want to subtract the selected objects.

**Delete** - Select this to remove the selected object from the current selection set.

**Rename** - Select this to put the selection set name into edit mode in the **Selection Sets** list.

**Related topics**

- ❑ [Create and Use Selection Sets](#) (on page 57)
- ❑ [Understanding Selection Sets](#) (on page 54)

## View and Edit an Object's Properties

Review and edit the properties for selected objects to ensure they have the correct attributes. If you select a single object, the properties for that object are displayed. If you select multiple objects, the properties common to all of them are displayed. You can edit the common properties, or select a subset of the selected objects using the drop-down list near the top of the pane.

To view the properties of another object in the **Properties** pane, click the object within any data view or pane. If the **Properties** pane is displayed, selecting any object will show its properties.

**Note:** The toolbar icons and context menu items available in the **Properties** pane depend on the types of objects you have selected. The **Properties** pane also enables you to use COGO controls and snap commands within certain property boxes.

### To edit properties:

- Click in a property box, highlight the value to edit, and type a new value, or (if applicable) select a new value from the list.

**Note:** You can enter a value in units different from the project units; it will be converted to the project units. For example, if an elevation is displayed in meters and you enter a value as "10 ft", it will convert to "3.048" (meters).



- Press [**Esc**] to cancel an edit; the property will revert to its original value.
- Press [**Tab**] to accept an edit and move to the next property box.
- Press [**Shift**] + [**Tab**] to accept an edit and move to the previous property box.
- Press [**Enter**] to accept an edit and close the *Properties* pane.

### Sections



Click this on the pane's toolbar to select any associated point, coordinate, baseline, vector, or trajectory when survey data is selected.

#### Sub-selection list

This displays all of the selected objects by type. If you have selected multiple objects, you can accept the default selection of **All**, or click the drop-down arrow to narrow the selection to a specific type of object.

#### Properties

This displays groups of properties. Click a property label to see additional information, or click in a property box to edit the value, when available.

---

**Note:** If you are in a [COGO control](#) (see "Understanding COGO Controls" on page 86) in the grid, you can pick a point(s) or object(s) in the view, or right-click for snap options.

---

#### (information box)

This displays a description of the selected property.

### Related Topics

- ❑ [Properties Pane](#) (on page 14)
- ❑ [Pane and Data View Positioning](#) (on page 31)

## Delete an Object

Remove objects that you no longer need from the project database.

#### To select and then delete data:

1. Select the data to delete in a graphic view, spreadsheet, or the *Project Explorer*.  
The number of objects selected appears in the status bar.
2. Do one of the following:
  - Select **Edit > Delete**.
  - Press [**Ctrl**] + [**D**].
  - Right-click and select **Delete** from the context menu.

The selected objects are removed from graphic views, spreadsheets, and the *Project Explorer*. The *Delete* command pane does not display.

#### To start Delete and then select the data:

1. Select **Edit > Delete**. The *Delete* command pane displays.
2. Select data using one of the following:
  - Click **Options**, and select an option from the menu.
  - Pick objects in a graphic view, spreadsheet, or the *Project Explorer*.

The number of objects selected appears in the status bar.

3. Click **OK**. The selected objects are removed from graphic views, spreadsheets, and the *Project Explorer*.

#### To restore deleted data:

- Select **Edit > Undo Delete** immediately after using the *Delete* command.


#### Related topics

- ❑ [Filter a View](#) (on page 76)
- ❑ Undo and Redo Commands
- ❑ [Selection Methods and Options](#) (on page 43)

## Edit an Object

When you need to edit an object, the program will open the editor appropriate to the object's type.

#### To edit an object:

- Right-click an object, and select **Edit** from the context menu.
- Select **Edit > Edit**.
- Click the  icon on the toolbar, or on the *Properties* pane toolbar.

Depending upon whether you have an object selected or not, either the **Edit** command pane displays (prompting you to select an object) or the editor appropriate to the selected object displays.

#### Related topics


- ❑ [Edit an Alignment](#) (on page 290)
- ❑ [Edit a Linestring's Horizontal Segments](#) (on page 301)
- ❑ [Edit a Surface by Adding and Removing Members](#) (on page 328)

## Undo or Redo an Action

Any action that affects the project database, such as creating an object, can be redone or undone. Commands that don't affect the database, such as opening a file, cannot be redone or undone.

### To undo an action:

Use the **Undo** command to revert to the state prior to the last action.

- Select **Edit > Undo** command.
- Press **[Ctrl] + Z**.
- Click the  icon on the toolbar.

Multiple actions can be undone one at a time. The most-recently performed action is undone first. Once you close the project or the software, no actions can be undone.

### To redo an action:

Use the **Redo** command to reverse the Undo command.

- Select **Edit > Redo** command.
- Click the  icon on the toolbar.

Once you close the project or the software, no actions can be redone.

**Note:** Viewing actions, such as zoom, pan, and rotate, do not appear in the Undo / Redo list.

## Manage the Data in Your Views

### Understanding Layers and View Filters

Layers enable you to separate *different* types of data, and group *related* types of data, in the same way that transparent overlays in drawings used to.

View filters let you set the visibility of each data type and layer to control what is displayed in graphic views, helping you reduce and simplify what you see. View filters can be customized for each kind of view so that only the useful types of data are shown in each view.

With view filters, you can also set the selectability of data types and layers. For example, if you have a layer containing just background reference data that you will always need to see, but never need to use, simply make it unselectable.

Layers and view filters serve unique purposes, but work powerfully in conjunction with each other. Once you have set up layers and view filters, you can save them in project templates, making them ready to use each time you start a new project.

### Related topics


- ❑ [Create and Edit a Layer](#) (on page 68)
- ❑ [Create a View Filter](#) (on page 73)
- ❑ [Layer Options](#) (on page 71)
- ❑ [View Filter Manager](#) (on page 9)


## Create and Edit a Layer

Use layers to keep your data organized by type. You can create and delete individual layers, purge all empty layers, and select objects by choosing their layer. Layer properties allow you to change layer colors and line styles to help you distinguish more easily between data on different layers.

**Note:** Layers can only be imported or exported as part of CAD (.dxf, .dwg, and .dgn) files.

### To create a new layer:

1. Do one of the following:
  - Select **Project > Layer Options**.
  - Click the  icon on the toolbar.

The **Layer Options** command pane displays.
2. Click the  icon on the command pane's toolbar. The **New Layer** dialog displays.
3. Type a unique name in the **Layer name** box to indicate what type of data the layer will contain.
4. Select layer display options in the **Color**, and **Line style** lists.

**Note:** If you have a separate line styles file, you can import it using the line styles list.



**Note:** If the scale of your lines is small in a graphic view, some line styles display as solid lines to improve performance.

5. Click **OK**. The new layer appears in the pane's **Layers** list.

### To edit properties of a layer:


1. Starting after step 1 above, select a layer in the **Layers** list. The layer's properties appear in the **Layer Properties** group.
2. Click in a box and edit the property or select a new property. The edits are reflected in graphic views immediately.

**To delete one or all empty layers:**

1. Select one or more empty layers in the *Layers* list.
2. Click the  icon on the pane's toolbar. The layer is deleted.
3. To delete all empty layers, click the  icon on the pane's toolbar.

**Note:** Empty layers are those that have no objects assigned to them. If there are no empty layers selected or in your project, the icons are unavailable.

**To select the objects that reside on a layer:**

1. Select one or more layers in the *Layers* list.
2. Click the  icon on the pane's toolbar. The objects on the selected layers highlight in graphic and spreadsheet views and in the *Project Explorer*.



**To change the layer on which objects reside:**

1. Select the objects for which you want to change the layer.
2. Select **Edit > Properties**. The *Properties* pane displays.
3. If needed, select a subset of the objects in the data type list at the top of the pane.

**Note:** To change the layer, all of the selected objects must have *Layer* as a property. Generally, survey data and survey points do not have a layer property.



4. Click in the *Layer* box and select the new layer.
5. Click **Close**.

**To set the visibility or selectability of objects by layer:**

1. Do one of the following:
  - Select **View > View Filter Manager**.
  - Click the  icon on the toolbar.
 The *View Filter Manager* displays.
2. Click the tab at the top of the view for which you want to edit the layers.
3. Select an unlocked view filter in the  list at the top of the pane.

**Note:** This sets the visibility and selectability of the layers only for the view filter you have selected.

**Note:** To unlock a locked view filter, you can modify or copy it. See [Create and Edit a View Filter](#) (see "Create a View Filter" on page 73).

4. Click the  icon on the pane's toolbar. The **Advanced View Filter Settings** dialog displays.
5. Scroll down and click the  icon next to **Layers** to expand the group.
6. Check and uncheck boxes in the **Visible** and **Selectable** columns as needed.
7. Click **OK**.






### Related topics

- ❑ [Layer Options](#) (on page 71)
- ❑ [Isolate or Exclude a Layer](#) (on page 70)
- ❑ [Create and Edit a View Filter](#) (see "Create a View Filter" on page 73)
- ❑ [Filter a View](#) (on page 76)



## Isolate or Exclude a Layer

Turn individual layers on or off to better understand what data is on each layer in your project, and to help you create view filters using just the relevant layers.



### To isolate individual layers:

1. Do one of the following:
  - Select the **View > View Filter Manager**.
  - Click the  icon on the toolbar.  
The **View Filter Manager** displays.
2. Click the tab at the top of the view in which you want to isolate the layer.
3. Select a view filter in the  list at the top of the pane. If you want to create a new view filter from an unlocked view filter, click the  icon to make a copy so you do not modify the original view filter.
4. If desired, click the  icon on the pane's toolbar to turn on the **Auto Zoom** (to the extents of the visible data) mode.
5. Click the  icon on the pane's toolbar to turn on the **Isolate Layers** mode. The groups in the **View Filter Manager** collapse so that only the **Layers** group shows.
6. Click the name of each layer to see the data that resides on it in the graphic view. Only objects on the selected layer are visible.
7. If you are creating a view filter and want it to show the selected layer, make sure its box is checked. If you do not want to include the layer in the view filter, uncheck the box.

**Tip:** You may find it faster to use the keyboard to isolate layers. Click a layer name, and then use the up and down arrow keys to move through the list. Press the spacebar to check and uncheck the boxes that control the layer visibility.

8. Once you are done, click the  and  icons again to return to normal mode. The new view filter shows only the data on the layers for which the boxes are checked.

#### To hide (exclude) layers by selecting objects:

1. Starting after step 4 above, select one or more objects (in the graphic view) of the types you want to hide or that reside on the layers you want to hide.
2. Click the  icon on the *View Filter Manager's* toolbar. The layers that the objects were assigned to are hidden.
3. Click the  icon again to return to normal mode.

**Tip:** This method will not hide unlayered data types, such as survey data. If you are primarily interested in CAD data, you may want to hide raw data types before using this method.

#### Related topics

- ❑ [Create and Edit a Layer](#) (on page 68)
- ❑ [Layer Options](#) (on page 71)
- ❑ [Create and Edit a View Filter](#) (see "Create a View Filter" on page 73)
- ❑ [Filter a View](#) (on page 76)
- ❑ [View Filter Manager](#) (on page 9)

#### Layer Options

Use these options to create and delete layers, select objects by choosing their layers, and edit layer properties. They are available in the *Layer Options* command pane.

## Options



Click this to open the **New Layer** dialog, where you can create a new layer.



Click this to delete one or more empty layers that you have selected in the **Layers** list.

Empty layers are those that have no objects assigned to them. You cannot delete layers that have objects assigned to them.



Click this to delete all empty layers from the **Layers** list and your project.



Click this to select all of the objects on the layers you have selected.

## Layers

This lists all of the layers in your project, enabling you to select one or more layers or the objects on them.

## Layer properties

This shows the attributes of the selected layer. Click in the box for any property to edit it.

## Related topics


- ❑ [Create and Edit a Layer](#) (on page 68)
- ❑ [Isolate or Exclude a Layer](#) (on page 70)

## View Filter Manager

The **View Filter Manager** is a pane in which you can select data types and layers to specify what is visible in the current graphic view, helping you reduce and simplify what you see. As you make changes in the manager, the view updates to reflect them.

View filters are saved sets of criteria that control what data and layers are displayed in the views. View filters can be defined separately for each type of view so that only the data that is important for the current phase of your work is displayed. When you change to a different view, the current and available view filters may change as well, because view filters are saved with views. The view filters for each view in your project can be accessed from the view filter list on the toolbar.


### To display and pin the View Filter Manager:

1. Do one of the following:
  - Select **View > View Filter Manager**.
  - Click the  icon on the toolbar.


The **View Filter Manager** displays, docked on the left side of the application window, or where you positioned it last.

**Note:** If the **Project Explorer** or **Selection Explorer** are also active, they may share the same pane, and be accessible as tabs at the bottom of the pane.



2. If desired, pin the manager open by clicking the  icon at the top. If the pane is unpinned, it can "slide" to the side and out of view. To show it again, click the **View Filter Manager** tab on the left edge of the application window.

### Using the View Filter Manager

- When you check and uncheck boxes for data types and layers in the **View Filter Manager**, the current view changes in response.
- Arrange the order of the data type groups in the tree by right-clicking on a group and selecting **Move Up** or **Move Down** from the context menu.
- To set the selectability of data types and layers, click the  icon on the pane's toolbar to display the **Advanced View Filter Settings** dialog.


### Related topics


- ❑ [Create a View Filter](#) (on page 73)
- ❑ [Edit a View Filter](#) (on page 75)
- ❑ [Filter a View](#) (on page 76)
- ❑ [Pane and Data View Positioning](#) (on page 31)

## Create a View Filter


Use the **View Filter Manager** to select data types and layers to specify what is visible in graphic views, helping you reduce and simplify what you see. As you make changes in the manager, the current view updates to reflect them. You can create, copy, and delete view filters in the manager. View filters allow you to select data types and layers to control what is visible in graphic views, helping you reduce and simplify what you see. In creating a view filter, you set criteria for what to show in views.

### To create a view filter:

1. Do one of the following:
  - Select the **View > View Filter Manager**.
  - Click the  icon on the toolbar.




The **View Filter Manager** displays.
2. Click the tab at the top of the view for which you want to create the view filter.
3. In the  list, select a **locked** view filter that is close to the one you are trying to create; a copy of the original view filter is created as you make changes. If you are concerned about accidentally changing your existing view filters, and have most of them locked, use this method.

Or

Select an **unlocked** view filter that is close to the one you are trying to create and click the  icon on the pane's toolbar. A copy of the view filter is created. If you do not want to be bothered with locking/unlocking view filters, and have most of them unlocked, use this method.


**Note:** There is one pre-defined view filter (*All*) which cannot be deleted.

**Tip:** If you have project templates, you may want to create and add custom view filters to them.

4. If desired, narrow the view filter by selecting a selection set in the  list (for more on selection sets, see [Create and Use Selection Sets](#) (on page 57)).
5. If necessary, click the  and  icons to expand and collapse groups.
6. Check and uncheck boxes to show and hide different data types and layers as you watch the changes in the graphic view. All of the changes you make are automatically saved to the current view filter for the current view.
7. For additional display style options, click the tabs at the bottom of the pane and change the settings as needed.

**Note:** The display styles tabs are only available for unlocked view filters, and generally only affect raw data, such as survey data.



#### To create a view filter using advanced settings:

1. Starting after selecting an unlocked view filter in step 3 above, click the  icon on the pane's toolbar. The **Advanced View Filter Settings** dialog displays.
2. Use the process described in steps 4 - 6 above to create a view filter. Additional options (such as *Include newly created layers*, *Locked*, *Show display styles*, and *Selectable*) are also available. See [Advanced View Filter Options](#) (on page 81) for details.

**Note:** Since you are working in a dialog, you will not see the graphic view update as you make changes to the view filter. For projects with a lot of data, this is more efficient because the screen does not have to refresh between changes. You can, however, click **Apply** at the bottom of the dialog to manually refresh the graphic view at any time.



3. Click **OK**. The new view filter is associated with the current graphic view; other views have the option of using the new view filter as well.

**To copy a view filter:**

- Select a view filter in the  list, and click the  icon on the pane's toolbar. The copy of the view filter appears in the list. The original view filter's name is appended with " - *Copy*" in the new view filter's name. The copy is unlocked and is associated with the current graphic view.

**Note:** The *Copy*, *Delete*, and other commands can also be accessed by clicking in the view filter list and selecting from the context menu.

**To delete a view filter:**

- Select a view filter in the  list, and click the  icon on the pane's toolbar. The view filter disappears from the list, and the current graphic view becomes associated with the default view filter. If the default view filter does not exist, the view is associated with the original default view filter named *All*.

**Note:** The *All* view filter cannot be deleted.


**Related topics**



- ❑ [Edit a View Filter](#) (on page 75)
- ❑ [Filter a View](#) (on page 76)
- ❑ [View Filter Manager Options](#) (on page 79)
- ❑ [Create and Edit a Layer](#) (on page 68)

**Edit a View Filter**

Use the *View Filter Manager* to edit, copy, and rename existing view filters. View filters can be customized for each view so that you see only the types of data you need for your current task.




**To unlock/lock a view filter:**

1. Do one of the following:
  - Select the **View > View Filter Manager**.
  - Click the  icon on the toolbar.


The *View Filter Manager* displays.
2. Click the tab of the view associated with the view filter you want to unlock.
3. Select the view filter in the  list.
4. Click the  icon on the pane's toolbar. The *Advanced View Filter Settings* dialog displays.
5. Check or uncheck the **Locked** box, and click **OK**.

**Note:** If you modify a locked view filter, an unlocked copy is automatically made. If you copy a locked view filter, the copy is unlocked.

#### To edit a view filter:

1. Starting after step 3 above, click the  and  icons to expand and collapse groups if necessary.
2. Check and uncheck boxes to show and hide different data types and layers as you watch the changes in the graphic view. All of the changes you make are automatically saved to the current view filter for the current view.
3. For additional display style options, click the tabs at the bottom of the pane and change the settings as needed. For advanced options, click the  icon on the pane's toolbar.

#### To rename a view filter:

1. Select an unlocked view filter in the list and click the  icon on the pane's toolbar. Alternately, you can click in the view filter list and select **Rename View Filter** from the context menu.
2. Type a new name for the view filter in the **New name** box, and click **OK**.

#### Related topics

- ❑ [Create and Edit a Layer](#) (on page 68)
- ❑ [Filter a View](#) (on page 76)
- ❑ [Troubleshoot a Layer or View Filter Problem](#) (see "Troubleshoot a Layer or View Filter Problem" on page 83)
- ❑ [Layers and View Filters](#) (see "Manage the Data in Your Views" on page 67)

## Filter a View

Use view filters to reduce and simplify what you see in graphic views. The software comes with one pre-defined, default view filter that shows all of the visible data in your project. Use this view filter, named **All**, as the starting point, and adapt it to create a set of basic view filters to fit your needs. Then, you can modify those view filters to build more subtle and complex ones.

#### Filter a view to inspect your project data: (simple)

1. Click the tab of the graphic view you want to filter.
2. Click in the view filter list on the toolbar, and select the view filter that meets your needs for the current view. Depending on the data in your project, the view changes.

The view filter list on the toolbar looks like this:




**Note:** The view filter list shows all of the view filters that have been created in the *View Filter Manager*. If no additional view filters have been created for the project, only the default *All* view filter is available.




**Note:** The *All* view filter cannot be deleted.

3. Select other view filters in the list to try other view filters. If the view filter you need has not been created yet, and you want to modify the existing view filter, use the procedure below.




### Filter a view to inspect your project data:

1. Click the tab of the graphic view you want to filter.
2. Do one of the following:
  - Select the **View > View Filter Manager**.
  - Click the  icon on the toolbar.



The *View Filter Manager* displays.

3. Select a view filter in the  view filter list at the top of the pane.
4. If you selected an unlocked view filter, and want to preserve the original, click the  icon on the pane's toolbar to create a copy.
5. If desired, narrow the view filter by choosing a selection set in the  list.

**Note:** Selection sets are automatically created for many types of imported files. For more information on selection sets, see [Create and Use Selection Sets](#) (on page 57).

6. If necessary, click the  and  icons to expand and collapse the groups.
7. Check and uncheck boxes to show and hide different data types and layers as you watch the changes in the graphic view.
8. For additional display style options, click the tabs at the bottom of the pane and change the settings as needed. Continue to experiment by making different combinations of data visible. This will help you understand what is in your project.
9. If an unneeded view filter was created, click the  icon on the pane's toolbar to delete it. If you do not delete the view filter, it will be available in the view filter list on the toolbar.

### Filter a view to see only one type of data:

1. Starting after step 4 above, click the  and  icons to expand and collapse the groups if necessary.

2. Right-click the name of a data type you want to view, and select **View Only This** from the context menu. All of the other boxes in the group are automatically unchecked so that only the selected object type is visible.

Or

3. Right-click on the tree view and select **Hide All** from the context menu. All of the boxes in the group are automatically unchecked.
4. Check the box for the one object type you want to see.

**Note:** When using **View Only This** or **Hide All**, the check boxes in other groups are not affected. If you want to see only one data type from all groups, uncheck the boxes next to the other group names before selecting **View Only This** or **Hide All**.

**Note:** There are two things that affect whether an object is visible in the graphic views: whether the box for the object type is checked, and whether the box for the layer that the object resides on is checked. If an object does not display as expected, make sure both boxes are checked. Generally, this only applies to **Points** and **CAD Blocks, Lines, and Text** (only available in the **Advanced View Filter Settings** dialog).


#### Filter a view to see only one layer:

- See [Isolate or Exclude a Layer](#) (on page 70).

#### Filter a view to see only objects in a selection set:

Selection sets are automatically generated by some types of imported files. You can also define them using selected objects in the **Selection Explorer** (on page 8). Use selection sets to narrow the data included in a view filter.

**Note:** Selection sets can contain objects that have no visible display, such as coordinates. In the **Selection Explorer**, all selection sets are available. In the **View Filter Manager**, however, only selection sets that contain at least one visible object are available.

1. Select the **All** view filter in the  list.
2. Select the selection set in the  list. Only data that is in the selection set is visible.

#### Filter a view to see everything but one data type or layer:

1. Select the **All** view filter in the **View Filter Manager**. Alternately, you can also check the box for each group or right-click in each group and select **View All** from the context menu to see all of the data.
2. Uncheck the box for just the data type or layer you want to hide.

#### Related topics

- [Create and Edit a View Filter](#) (see "Create a View Filter" on page 73)

- [Create and Edit a Layer](#) (on page 68)
- [Troubleshoot a Layer or View Filter Problem](#) (see "Troubleshoot a Layer or View Filter Problem" on page 83)

### **View Filter Manager Options**

Use these options to select data types and layers to control what is visible in graphic views, helping you reduce and simplify what you see. View filters can be customized for each kind of view so that only the useful types of data are shown for each view. They are available in the *View Filter Manager*.

## Options



Click this to make a copy of the currently selected view filter. The original view filter's name is appended with "- **Copy**" in the new view filter's name.



Click this to rename the currently selected view filter. Then type a unique name for the view filter in the **New name** box, and click **OK**.



Click this to delete the currently selected view filter. The view filter disappears from the list, and the current graphic view becomes associated with the default view filter. If the default view filter does not exist, the view is associated with the view filter named **All**.

Only unlocked view filters can be deleted.



Click this to put the **View Filter Manager** into **Auto Zoom** mode in which the current graphic view automatically zoom to the extents of the data that is visible in the current view filter.



Click this to put the **View Filter Manager** into **Isolate Layers** mode. All of the groups collapse except for the **Layers** group. Select a single layer to see only the data on that layer.



Pick objects in the graphic view, and click this to hide the layers on which the selected objects reside.



Click this to open the **Advanced View Filter Settings** dialog, where you can set additional view filter and layer settings, as well as the standard settings found in the **View Filter Manager**.



**(view filter list)**

Select a view filter to apply it to the current view. Then you can edit it or copy it.

This list matches the view filter list found on the toolbar.



**(selection set list)**


Select from selection sets automatically generated by imported files or selection sets you have defined to narrow the data included in the selected view filter.

---

**Note:** Selection sets can contain objects that have no visible display, such as coordinates. In the **Selection Explorer**, all selection sets are available. In the **View Filter Manager**, however, only selection sets that contain at least one visible object are available.

---

**(object and layer tree)**

Click the  icon to expand a group, and check and uncheck boxes in the group to control the visibility of different types of objects and layers in the graphic view.

**Tab groups**

Click these tabs near the bottom of the pane to set additional display style options. They only appear when you select an unlocked view filter, and they are part of the view filter criteria.

For details, and to hide the tabs, see [Advanced View Filter Options](#) (on page 81).

**Context menu options**



<b>View all/ Hide all</b>	Within a group, use this option to check/uncheck all of the boxes, thereby displaying or hiding all of those object types or layers in the graphic view.
<b>View only this</b>	Within a group, use this option to check the selected object type or layer, and uncheck everything else in the group. <hr/> <b>Note:</b> If no data is visible after you select <b>View Only This</b> , make sure that the box for the layer the data is on is also checked. <hr/>
<b>Move up/ Move down</b>	On a group, use this option to move the group up or down in the tree so that your most frequently used groups are at the top. The order is saved with your project, and is the same for all view filters.
<b>Invert</b>	Within a group, use this option to check all of the boxes that are unchecked and uncheck all of the boxes that are checked.
<b>Layer options</b>	Within the <b>Layers</b> group, use this option to open the <b>Layer Options</b> (on page 71) command pane.

### Related topics

- ❑ [Create a View Filter](#) (on page 73)
- ❑ [Edit a View Filter](#) (on page 75)
- ❑ [Filter a View](#) (on page 76)
- ❑ [Troubleshoot a Layer or View Filter Problem](#) (see "Troubleshoot a Layer or View Filter Problem" on page 83)
- ❑ [Advanced View Filter Options](#) (on page 81)

### Advanced View Filter Options

Use these options to manage view filters, including controlling selectability and how new objects are handled. Most of the **View Filter Manager** options are also available. They are available in the **Advanced View Filter Settings** dialog.

## Options

### Filter name



### (selection set list)

If necessary, rename the selected view filter in this box.

Select from selection sets automatically generated by imported files or selection sets you have defined to narrow the data included in the selected view filter.

---

**Note:** Selection sets can contain objects that have no visible display, such as coordinates. In the **Selection Explorer**, all selection sets are available. In the **View Filter Manager**, however, only selection sets that contain at least one visible object are available.

---

### Include newly created layers

Check/uncheck this box to include/exclude object types and layers that are added to the project after the view filter was defined.

### Locked

Check/uncheck this box to make the view filter uneditable/editable.

### Show display styles

Check/uncheck this box to show/hide the display styles tabs in the **View Filter Manager**.

### Show CAD display options

Check/uncheck this box to show/hide the CAD data group (CAD block, line, text) in the **View Filter Manager**.

### Visible / Selectable (in the groups)

Check/uncheck the boxes to control what is visible and selectable in graphic views.

In addition to the standard data types and layers available in the **View Filter Manager**, there is a group for CAD blocks, lines, and text.

### Apply (button)

Click this to apply your changes and refresh the current graphic view.

### Point tab options

#### Show point labels

Check and uncheck this box to show and hide the label text of the point ID for points.

#### Show disconnected points

Check and uncheck this box to show and hide points that are not connected to baselines or vectors.

### Observations tab options

Click this tab for survey-related view filter options.

### GNSS tab options

Click this tab for survey-related view filter options.

### Context menu options

#### View all/ Hide all

Within a group, select this to check/uncheck all of the boxes, displaying or hiding all of those object types or layers in the graphic view.

#### View only this

Within a group, select this to check the selected object type or layer, and uncheck everything else in the group.

---

**Note:** If no data is visible after you select **View Only This**, make sure that the box for the layer the data is on is also checked.

---

#### Move up/ Move down

On a group, select this to move the group up or down in the tree so that your most frequently used groups are at the top.

The order is saved with your project, and is the same for all view filters.

#### Layer options


Within the **Layers** group, select this to open the **Layer Options** (on page 71) command pane.

### Related topics

- ❑ [Create a View Filter](#) (on page 73)
- ❑ [Edit a View Filter](#) (on page 75)
- ❑ [Filter a View](#) (on page 76)
- ❑ [Troubleshoot a Layer or View Filter Problem](#) (see "Troubleshoot a Layer or View Filter Problem" on page 83)
- ❑ [View Filter Options](#) (see "View Filter Manager Options" on page 79)

## Troubleshoot a Layer or View Filter Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
The graphic view redraws slowly when you make changes to view filters.	Your project contains a lot of data, and your computer's graphics memory is running at capacity.	<p>Click the  icon on the <b>View Filter Manager's</b> toolbar to open the <b>Advanced View Filter Settings</b> dialog. When you make changes to view filters in this dialog, the graphic view does not automatically redraw. You can click <b>Apply</b> at any time to have the view redraw.</p> <p>or</p> <p>Select <b>Project &gt; Project Settings</b>. Click <b>View</b> and then <b>View Filters</b> in the left pane, and select a default view filter other than <b>All</b> so that graphic views refresh more quickly.</p>


The points on a layer are not visible, even though you have the layer's box checked in the <b>View Filter Manager</b> to make it visible.	The <b>Point</b> box in the <b>Raw Data</b> group is not checked.	Make sure that the boxes for both the points and the layer that the points are on are checked, making them visible in the view.
Some of the selection sets you created do not appear in the <b>View Filter Manager's</b> selection sets list.	The missing selection sets that do not contain any visible objects.	None. Selection sets can contain objects that have no visible display, such as coordinates. In the <b>Selection Explorer</b> , all selection sets are available. In the <b>View Filter Manager</b> , however, only selection sets that contain at least one visible object are available.

### Related topics

- ❑ [Create a View Filter](#) (on page 73)
- ❑ [Edit a View Filter](#) (on page 75)
- ❑ [Filter a View](#) (on page 76)
- ❑ [View Filter Options](#) (see "View Filter Manager Options" on page 79)
- ❑ [Advanced View Filter Options](#) (on page 81)

## Troubleshoot a View or Selection Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
The graphic view redraws slowly when you make changes to view filters, or when you change the size of panes.	Your project contains a lot of data, and your computer's graphics memory is running at capacity.	Click the  icon on the <b>View Filter Manager's</b> toolbar to open the <b>Advanced Settings</b> dialog. When you make changes to view filters in this dialog, the graphic view does not automatically redraw. You can click <b>Apply</b> at any time to have the view redraw.  or  Select <b>Project &gt; Project Settings</b> . Click <b>View</b> and then <b>View Filters</b> in the left pane, and select a default view filter other than <b>All</b> so that graphic views refresh more quickly.
The 3D view is replaced by a red X and this message: "The system has run out of graphics memory. Close any unnecessary	The system has run out of graphics memory. (no screen saver interruption)	1. Close any unneeded programs that are running, especially ones that are graphics intensive. 2. Close all 3D views, including those with a red X. 3. Reopen the minimum number of 3D views you need.

windows and retry."		Long-term: Consider upgrading your graphics card.
The 3D view is replaced by a red X and this message: "The system has run out of graphics memory. Close any unnecessary windows and retry."	The screen saver came on and interrupted Microsoft® DirectX.	<ul style="list-style-type: none"> <li>▪ Close the 3D view and reopen it.</li> <li>▪ Close and reopen the program and the project.</li> <li>▪ Update to the latest version of DirectX.</li> </ul>
Some of the selection sets I created do not appear in the View Filter Manager's selection sets list.	The missing selection sets that do not contain any visible objects.	None. Selection sets can contain objects that have no visible display, such as coordinates. In the Selection Explorer, all selection sets are available. In the View Filter Manager, however, only selection sets that contain at least one visible object are available.
Your graphic views are pixilated or contain artifacts when you pan or rotate them.	You are not using the optimal advanced display setting.	Select <b>Tools &gt; Options</b> . In the <b>Options</b> dialog, click <b>Startup and Display</b> in the left pane, and then click <b>Advanced</b> . Check the <b>Override automatic detection</b> box, and select the appropriate option for your operating system.
Your mouse movements are delayed or track intermittently, even though you are using an advanced display setting (see above).	Your graphics card is integrated into the motherboard, or is not sufficient for advanced display settings.	<p>Select <b>Tools &gt; Options</b>. In the <b>Options</b> dialog, click <b>Startup and Display</b> in the left pane, and then click <b>Advanced</b>. Check the <b>Override automatic detection</b> box, and try each of these graphics display packages (in order, restarting the program between each):</p> <ol style="list-style-type: none"> <li>1. DirectX</li> <li>2. OpenGL</li> <li>3. GDI</li> </ol>
Your system suffers from generally poor graphics performance.	Two system settings are not set to optimize the graphics display.	Try the solution directly above first. If it does not improve your graphics performance, right-click on your Windows desktop, and select <b>Properties</b> from the context menu. In the <b>Display Properties</b> dialog, click the <b>Settings</b> tab, and select <b>Medium (16 bit)</b> in the <b>Color quality</b> list. Second, click the <b>Advanced</b> button in the same dialog. In the <b>Plug and Play</b> dialog, click the <b>Troubleshoot</b> tab, and set <b>Hardware acceleration</b> to <b>Full</b> .

# Calculate and Enter Values

This software incorporates interactive, graphical tools that you can use to calculate and enter values.

They include:

- [COGO controls](#) (see "Understanding COGO Controls" on page 86)
- [Running snap modes](#) (see "Set Running Snap Modes" on page 89)
- [Snap commands](#) (see "Use Snap Commands" on page 91)

Use these tools to calculate and enter angles, bearings, coordinates, distances, elevations, and offsets in your project. After you have imported survey, map, or engineering data, start at a known point or object, and use these functions to create other points and lines in your project.

COGO and snap tools are powerful because they enable you to enter data in consistent ways for a variety of commands.

## Understanding COGO Controls

COGO controls are the boxes in various commands that help you calculate angles, bearings, coordinates, distances, elevations, and offsets in your project. They enable you to enter data in a variety of ways, including:

- Typing values and point IDs in the box
- Picking points in graphic views
- Right-clicking in graphic views and selecting additional snap and COGO options from the context menu

COGO controls give you this flexibility so that you have many ways in which you can enter data within a single command, rather than forcing you to work through multiple commands.

Since COGO controls are used in many commands, once you understand how to use them, you will be able to apply your knowledge all to the commands that use them.

### **COGO cursors**

Depending on the type of control you are in, the appearance of the cursor changes in graphic views. The name of the control also appears on the status bar, and the status line's tooltip tells how use the control.

### Cursor types



Pick any point.



Pick any point or applicable object (arc, line, segment, point).



Pick any applicable object.



No anchor point has been defined.



No anchor point has been defined.



The pick function is unavailable, or the object you are trying to pick is invalid. Often, you can click a blank space in the **Project Explorer** or a command pane to refresh the cursor for the graphic views.

### Related topics

- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Set the Pick Aperture](#) (on page 88)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## COGO Expressions, Units, and Entry Formats

### Mathematical expressions

You can enter numbers in many of the COGO controls. When a COGO control supports numeric entry, you can use basic mathematic expressions by including the operators as shown. The value is calculated when you leave the control.

Operator	Example
<b>Addition</b>	10+10
<b>Subtraction</b>	10-2
<b>Multiplication</b>	10*8
<b>Division</b>	8/4
<b>Power</b>	10^2
<b>Simple expressions</b>	5*(6+8)
(for distance, bearing, and coordinate controls)	052>054 (for distance or bearing between points)
<b>Exponential notation</b>	1.01E+6
(for distance and coordinate controls)	
<b>Separated formats</b>	1,000,000 = 1000000
(for distance and coordinate controls)	1.000.000 = 1000000

### Units and data entry formats

The units, and entry and display formats, for data can be found and set in the [Units](#) (see "Unit Settings" on page 151) section of the *Project Settings* dialog.

The current distance units, which are also used for the gridlines, display on the status bar. Distances that you enter are in these units. You can, however, enter other units in COGO controls by including a character for the type of unit. For example, if your project units are set to *International foot*, you can enter 3m in a COGO control to specify 3 meters. The unit you enter will be converted to the project units.

### Entering point IDs

If you enter a point ID for a single value, such as an elevation, type quotation marks around the point ID to distinguish it as the elevation of the point, not just an elevation value. For example, to use the elevation for point 1001, type "1001" in the control. In coordinate controls where you are entering a pair of coordinates, typing the point ID without quotes will suffice. The coordinates of the point are used.

### Related topics

- ❑ [Change Project Units](#) (on page 152)
- ❑ [Unit Settings](#) (on page 151)

## Set the Pick Aperture

The pick aperture is the box appearing on certain cursors. It shows the area in which applicable objects can be picked.



**To set the size of the pick aperture:**

1. Select **Tools > Options**. The **Options** dialog displays.
2. Click **Startup and Display**.
3. Specify a value in the **Pick aperture** box.
4. Click **OK**.

**Related topics**

- ❑ [Startup and Display Options](#) (on page 139)

## Snaps Modes and Commands

Snaps use geometric calculations to help you specify and select coordinates and points more easily and accurately. Snaps can either use existing geometry or values that you enter during the command. There are two types of snap functions:

**Snaps****Running snap modes**

These modes are similar to the snap commands, except that (when enabled) they are always active. The running snap modes establish the order of precedence for the point snap modes. They can be temporarily overridden by using a specific snap command.

---

**Note:** Running snaps are available only for point (coordinate) snaps. There are no running snaps for angle, bearing, distance, offset, elevation, or station controls.

---

**Snap commands**

These commands enable you to enter angles, bearings, coordinates, distances, elevations, and offsets to calculate coordinates in your project. You can also select (or enter) applicable objects, such as breaklines, alignments, surfaces, and point IDs in snap commands.

**Related topics**

- ❑ [COGO Controls](#) (see "Understanding COGO Controls" on page 86)
- ❑ [Running Snap Mode Options](#) (on page 90)
- ❑ [Set Running Snap Modes](#) (on page 89)
- ❑ [Use Snap Commands](#) (on page 91)

### Set Running Snap Modes

Running snaps are frequently-used snaps that are constantly enabled ("running"), so that you do not have to initiate a specific snap command each time you need one. You can, however, specify which of the five running snap modes you want to be active at any time.

**Note:** If multiple snap modes are active, and you pick a point in a graphic view that satisfies multiple snaps, the snap closest to the center of the pick aperture prevails.

When a snap is active, the cursor's appearance indicates the type of snap. For instance, when you are in a snap mode that lets you pick an object, the cursor displays with a pick aperture (box). If you are in free snap, the cursor displays with cross hairs. See [COGO Controls](#) (see "Understanding COGO Controls" on page 86) for more information.

There is no visual feedback in graphic views when you snap to a point, but the coordinate of the point is entered into the control in which you are working.

### To set running snap modes:

1. Do one of the following:

- Click **Snap** on the status bar.
- Select **Project > Snap Mode**.

The *Snap mode* dialog displays.

2. Check (or uncheck) boxes next to any of the modes you want to enable (or disable).

**Note:** You need to have at least one snap mode enabled. Otherwise, you will not be able to pick anything in the graphic views.

3. Click **OK** to close the dialog.

**Note:** Running snap modes are superseded by snap commands.

### Related Topics

- [Running Snap Mode Options](#) (on page 90)

## Running Snap Mode Options

Use these options to set which frequently-used snaps are constantly enabled, so that you do not have to initiate a specific command each time you need a snap. They are available in the *Snap Mode* dialog.

## Options

<b>Point</b>	If a point object is inside the pick aperture, the coordinate of the point will be used.
<b>End point</b>	If the end point of a line or arc segment is inside the pick aperture, the coordinate of the end point of will be used.
<b>Insertion point</b>	If any part of a text or block object is inside the pick aperture, the insertion point of the object will be used.  This running snap mode is disabled by default.
<b>Surface vertex</b>	If a surface vertex is inside the pick aperture, the coordinate of the vertex will be used.
<b>Free</b>	The coordinate at the intersection of the cross hairs will be used. This mode will be used if none of the other modes are satisfied or selected.  If no other modes are active, the <b>Free</b> mode is active by default.

## Related topics

- ❑ [Set Running Snap Modes](#) (on page 89)

## Use Snap Commands

Snap commands use geometric calculations to help you specify coordinates more easily and accurately. Snaps can either calculate a snap point using existing geometry, or use parameters that you enter during a command. Snap commands, as opposed to running snaps, are single-instance snaps that you initiate each time you need one. They apply to the current command only.

There is no visual feedback in graphic views when you calculate (snap) to a point, but the coordinate of the point is entered into the control in which you are working.

### To use a snap command:

1. When you are in a [COGO control](#) (see "Understanding COGO Controls" on page 86), move your cursor into a graphic view and right-click. A context menu displays snap command options, depending on the control you are using.
2. Select one of the snap commands. The snap's command pane displays.
3. Specify parameters for the snap.
4. Click **OK** to return to the original COGO control.

**Note:** Snap commands supersede running snap modes.

## Stacked commands

To help you calculate a certain value, you can "stack" multiple snap commands. This means that you can access one snap command from within another until you have several snap commands stacked upon one another in the command pane. To practice, work through the tutorial.

### Related Topics

- ❑ [Set Running Snap Modes](#) (on page 89)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Enter an Angle

Use the Angle control to enter an angle by specifying a bearing in relation to the default bearing. The default bearing (and zero angle) is east. Positive angles are measured counter-clockwise from 0 to <360 degrees. "Angle" appears on the status bar when the command is active.

Angle controls are used in these snap commands:

- [Bearing + Angle Snap](#) (on page 96)

Angle controls give you access to these snap commands on the context menu:

- [Deflection Angle Snap](#) (on page 92)
- [Three Point Snap](#) (on page 93)

### To specify an angle:

- Pick an anchor point in a graphic view. With the cursor "rubber-banding" from the default bearing, pick another point to specify the angle.
- Type a value in the **Angle** box, using one of the standard entry formats. Check and set the entry format in *Project Settings* by selecting **Project > Project Settings > Units > Angular**.
- Right-click in a graphic view, select a snap command from the context menu, and specify the necessary parameters.

### Related topics

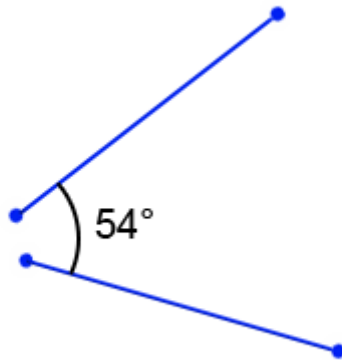
- ❑ [COGO Controls](#) (see "Understanding COGO Controls" on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Deflection Angle Snap

Use this command to calculate an angle between two bearings.

**Deflection Angle Snap** can be used in these controls:

- [Angle](#) (see "Enter an Angle" on page 92)



#### To use a Deflection Angle Snap:

1. While in an angle control, right-click in a graphic view, and select **Deflection Angle Snap** from the context menu. The **Deflection Angle Snap** command pane displays.
2. Pick two points or a line in the view, or type a value in the **Bearing 1** box.
3. Pick two more points or another line in the view, or type a value in the **Bearing 2** box. The angle between the bearings is recorded, and the command pane returns to the previous command.

#### Related topics

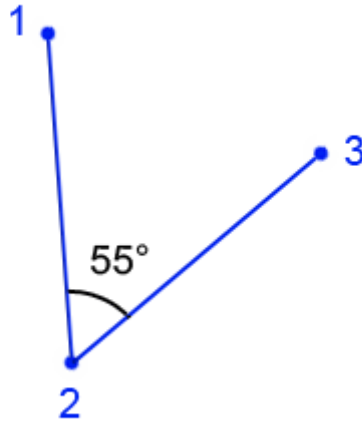
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### Three Point Snap

Use this command to calculate an angle based on three points picked in sequence.

**Three Point Snap** can be accessed from these control context menus:

- [Angle](#) (see "Enter an Angle" on page 92)



#### To use a Three Point Snap:

1. While in an angle control, right-click in a graphic view, and select **Three Point Snap** from the context menu. The **Three Point Snap** command pane displays.
2. Select an angle type (**Acute** or **Obtuse**) in the **Options** list.
3. Pick a point, or type a coordinate or Point ID in the **Start point** box.
4. Pick a second point, or type a coordinate or Point ID in the **Pivot point** box.
5. Pick a third point, or type a coordinate or Point ID in the **End point** box to specify the angle.
6. The angle between the three points is recorded, and the command pane returns to the previous command.

#### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Enter a Bearing

Use the Bearing control to specify an azimuthal bearing. The default bearing (and zero angle) is north. Positive bearings are measured clockwise. "Bearing" appears on the status bar when the command is active.

Bearing control is used in these snap commands:

- [Bearing Bearing Snap](#) (on page 99)
- [Bearing Distance Snap](#) (on page 100)
- [Bearing + Angle Snap](#) (on page 96)

Bearing control gives you access to these snap commands on the context menu:

- [Bearing + Angle Snap](#) (on page 96)
- [Point to Point Bearing Snap](#) (on page 97)

**To specify a bearing:**

- Pick an anchor point in a graphic view. With the cursor "rubber-banding" from the default bearing, pick another point to specify the bearing.
- Pick a line segment in a graphic view. The bearing of the segment is used. Each line segment has two bearings, so pick a point on the line segment close to the end to which you want to the bearing to travel. If you pick an arc segment, the bearing tangent to the segment at that point will be computed. If you pick a text object, the bearing at which the text object is located is used. This allows you to use the bearing control to align text objects with other text objects.
- Type a value in the **Bearing** box, using one of the standard entry formats. Check and set the entry format in **Project Settings** by selecting **Project > Project Settings > Units > Azimuth**.
- Type a point ID to point ID notation (e.g. 1>2) in the **Bearing** box, to recall the bearing between the points.
- Right-click in a graphic view, select a snap command from the context menu, and specify the necessary parameters.

**Horizontal Angle Modes**

You can also enter bearings using a prefix or suffix code for the angle mode. For example, for a 90 degree, 15 minute, and 2 second angle from the north azimuth, type: NA901502 or 901502NA.

The codes below are supported.

Code	Description
NA	North Azimuth
SA	South Azimuth
AR	Angle Right
AL	Angle Left
DR	Deflection Right

	Deflection Left
<b>DL</b>	
NE	Northeast
SE	Southeast
SW	Southwest
	Northwest
<b>NW</b>	

### Related topics

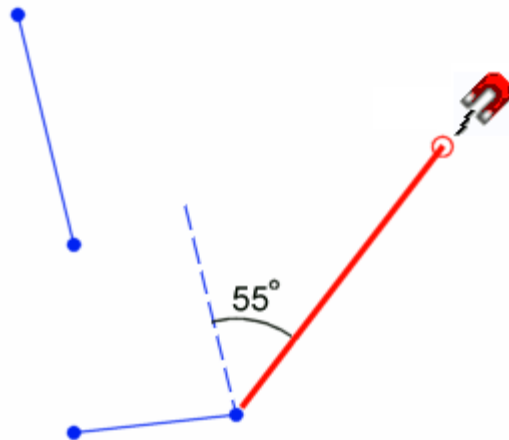
- ❑ [COGO Controls](#) (see "Understanding COGO Controls" on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Bearing + Angle Snap

Use this command to calculate a bearing by adding an angle to a given bearing.

**Bearing + Angle Snap** can be used in these controls:

- [Bearing](#) (see "Enter a Bearing" on page 94)



### To use a Bearing Plus Angle Snap:

1. While in a bearing control, right-click in a graphic view, and select **Bearing + Angle Snap** from the context menu. The **Bearing + Angle Snap** command pane displays.
2. Pick two points in the view, or type a value in the **Bearing** box.
3. Pick two more points, or type a value in the **Angle** box. The bearing is recorded, and the command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)



- ❑ [Snaps Modes and Commands](#) (on page 89)

## Point to Point Bearing Snap

Use this command to compute the bearing from one point to another point.

**Point to Point Bearing Snap** can be used in these controls:

- [Bearing](#) (see "Enter a Bearing" on page 94)

### To use a Point to Point Bearing Snap:

1. While in a bearing control, right-click in a graphic view, and select **Point to Point Snap** from the context menu. The **Point to Point Snap** command pane displays.
2. Pick a point in the view, or type a coordinate or Point ID in the **Reference point 1** box.
3. Pick a point, or type a coordinate or Point ID in the **Reference point 2** box. The bearing is recorded, and the command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Enter a Coordinate

Use the Coordinate control to specify X, Y, and Z, or latitude, longitude, and elevation coordinates of a point. If a Z or elevation coordinate is entered in the coordinate control, and there is also an elevation control for the same point, the value will be placed in the elevation control. "Coordinate" appears on the status bar when the command is active.

**To specify a coordinate:**

- Pick a point in a graphic view. If no running snap modes are enabled, the default operation does not specify a Z coordinate. If running snaps are enabled, the cursor will potentially snap to a location and use its Z value based on the snap mode.
- Type a point ID in the coordinate box using any of the standard entry formats. If there are multiple points with the same ID, then the one with the best quality is used. Check and set the entry format in the *Project Settings* by selecting **Project > Project Settings > Units > Coordinate**.

**Note:** If you enter a point ID for a single value, such as an elevation, type quotation marks around the point ID to distinguish it as the elevation of the point, not just an elevation value. For example, to use the elevation for point 1001, type "1001" in the control. In coordinate controls where you are entering a pair of coordinates, typing the point ID without quotes will suffice. The coordinates of the point are used.

- Type two or three numbers separated by a space or comma to specify a coordinate pair or triplet, in the format N, E, (Z). The separator is user-definable in the Project Settings. Typically, spaces or commas are used to specify coordinate pairs or triplets.

Examples: 27,42, (1)

27 42 1

27,42

27 42

In the plan view, these normally represent N, E, (Z). In the profile view these normally represent station and elevation. The control honors the ordering of the X and Y values as specified for the current view.

Although you will usually be entering grid based coordinates, you can also enter latitude and longitude coordinates in the format Latitude, Longitude, (Z). To do this, a coordinate system must be defined for the project. Do not use spaces to separate the coordinates because spaces are used to separate the angle components.

Examples: N40°35'18.12345", E120°23"12.32145", 1000

N40 35 18.12345, E120 23 12.32145, 1000

N40 35 18.12345, E120 23 12.32145

**To specify a relative coordinate:**

- Type an @ before a value to specify a relative distance from a previous point (when the cursor has an anchor point). The @ symbol must be used before each relative coordinate, but you can enter both relative and absolute coordinates in the same control. The relative distance separator is user-definable in the *Project Settings*, but typically, @ is used as shown.

Examples: @27,@42

@27 @42

In any view with a vertical exaggeration you can substitute a grade (G) for the elevation when setting a point relative to a previous point. This can be used in conjunction with a relative (@S) station value. Valid entries for grade are: (% or P) for percent of grade and (; or R) for ratio grade, e.g. 2 : 1.

Examples: @500,2%

@500 2P

In any view with a vertical exaggeration, you can substitute a Maximum (M) depth for the station and grade (G) for the elevation when setting a point relative to a previous point. The maximum depth will be the change in depth (elevation) from the previous point. Valid entries for grade are: (% or P) for per cent of grade and (; or R) for ratio grade, e.g. 2 : 1. This type of entry is typically used in defining templates used with roads.

Examples: 5,2%

5 2%

**Related topics**

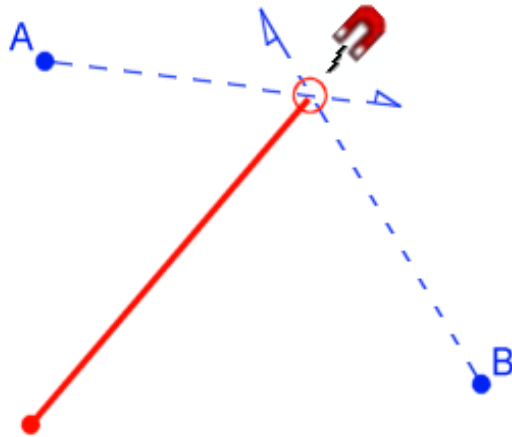
- ❑ [COGO Controls](#) (see "Understanding COGO Controls" on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

**Bearing Bearing Snap**

Use this command to calculate the intersection of two bearings defined by a first point and second point. No elevation (Z value) is set using this option. This snap is helpful in calculating the coordinates of a location that cannot be occupied, such as the center of a tree.

**Bearing Bearing Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use a Bearing Bearing Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Bearing Bearing Snap** from the context menu. The **Bearing-Bearing Snap** command pane displays.
2. Pick a point in the view, or type a coordinate or point ID in the **Reference point 1** box.
3. Pick a second point, or type a value in the **Bearing 1** box.
4. Pick a point, or type a coordinate or point ID in the **Reference point 2** box.
5. Pick a second point, or type a value in the **Bearing 2** box, to specify the bearing. The coordinate is calculated at the intersection of **Bearing 1** and **Bearing 2**, and the command pane returns to the previous command.

#### Related topics

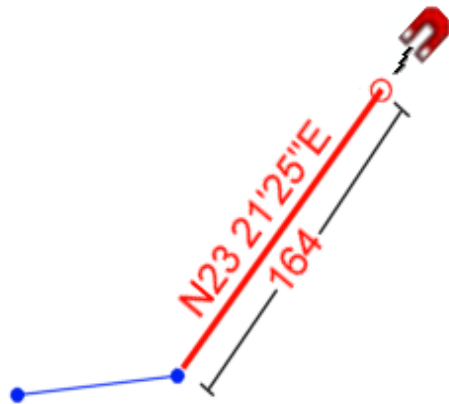
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### Bearing Distance Snap

Use this command to calculate a point based on a beginning point, a bearing, and a distance. The zero (0) bearing is north, and bearings are measured clockwise. No elevation (Z value) is set using this option.

**Bearing Distance Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



### To use a Bearing Distance Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Bearing Distance Snap** from the context menu. The **Bearing Distance Snap** command pane displays.
2. Pick a point in the view, or type a coordinate or point ID in the **Reference point** box.
3. Pick a second point, or type a value in the **Bearing** box.
4. Pick a third point, or type a value in the **Distance** box. The coordinate is recorded, and the command pane returns to the previous command.

### Related topics

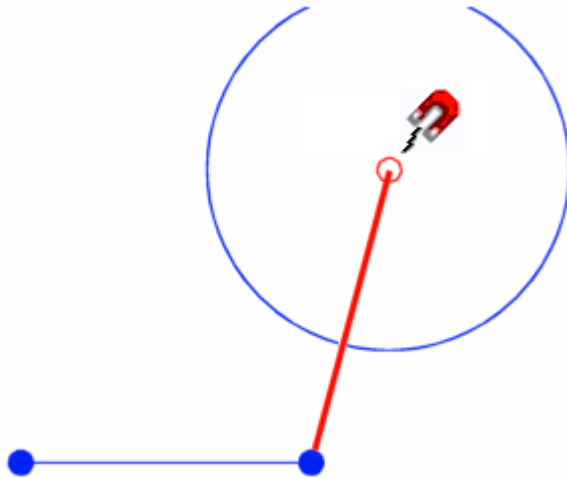
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Center of Arc Snap

Use this snap command to calculate the center point of an arc (or parabolic curve segment or spiral curve segment) when you select the arc. The elevation of the arc is used as the Z value.

**Center of Arc Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use a Center of Arc Snap:

1. Right-click in a graphic view, and select **Center of Arc Snap** from the context menu. The **Center of Arc Snap** command pane displays.
2. In the view, pick an arc. The coordinate is recorded, and the command pane returns to the previous command.

#### Related topics

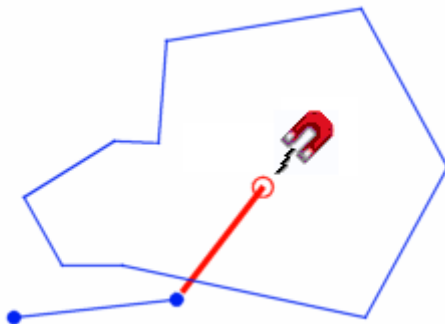
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### Center of Gravity Snap

Use this command to calculate a point based on the area within a closed line. If the line segment picked has an elevation, the Z value of the segment is used.

**Center of Gravity Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



**To use a Center of Gravity Snap:**

1. While in a coordinate control, right-click in a graphic view, and select **Center of Gravity Snap** from the context menu. The **Center of Gravity Snap** command pane displays.
2. Pick a segment of a closed line in the view. The coordinate at the center of the area is recorded, and the command pane returns to the previous command.

**Related topics**

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

**Distance Distance Snap**

Use this command to calculate a point based on radial distances from two reference points, selecting one of the two points where the resulting arcs intersect.

**Distance Distance Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)

**To use a Distance Distance Snap:**

1. While in a coordinate control, right-click in a graphics view, and select **Distance Distance Snap** from the context menu. The **Distance Distance** command pane displays.
2. Pick a point in the view, or type a coordinate or point ID in the **Center point 1** box.
3. Pick a second point, or type a value in the **Distance 1** box.
4. Pick a point, or type a coordinate or point ID in the **Center point 2** box.
5. Pick a second point, or type a value in the **Distance 2** box.
6. Select one of the intersecting points by picking a side in the view, or by selecting an option in the **Side** list. The coordinate is recorded, and the command pane returns to the previous command.

**Related topics**

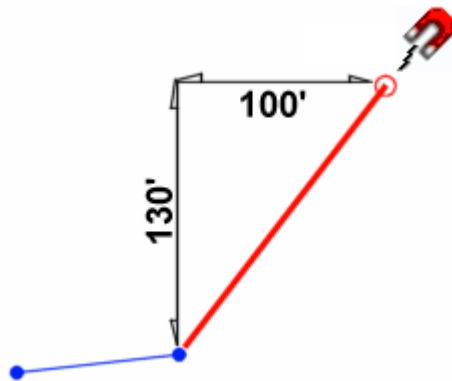
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

**Delta X Delta Y Snap**

Use this command to calculate a point using a relative X and Y distance from a reference point. The elevation of the origin point is used as the Z value.

**DxDy Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use a DxDy Snap:

1. While in a coordinate control, right-click in a graphic view, and select **DxDy Snap** from the context menu. The **DxDy Snap** command pane displays.
2. Pick a point in the view, or type a coordinate or point ID in the **Reference point** box.
3. Pick a second point, or type a distance from the reference point in the **Easting Distance** box.
4. Pick a third point, or type a distance from the reference point in the **Northing Distance** box. The coordinate is recorded, and the command pane returns to the previous command.

#### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

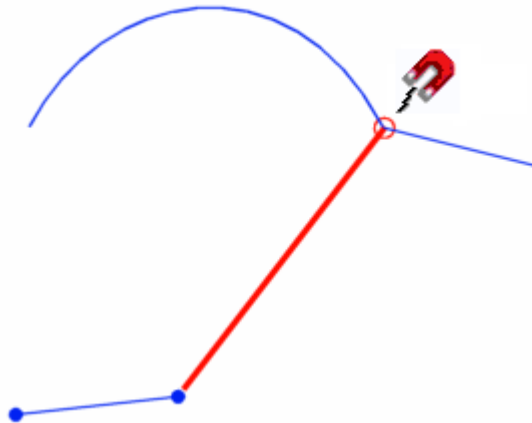
#### End Snap

Use this command to calculate the point at the end of a line segment closest to where you pick on the line segment. The elevation of the end point is used.

**End Snap** can be used in these controls:



- [Coordinate](#) (see "Enter a Coordinate" on page 97)



### To use an End Snap:

1. While in a coordinate control, right-click in a graphic view and select **End Snap** from the context menu. The **End Snap** command pane displays.
2. Pick a line segment near the end to which you want to snap. The coordinate at the end of the line segment is recorded, and the command pane returns to the previous command.

### Related topics

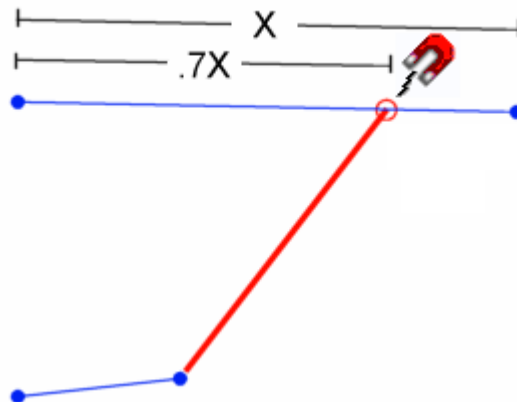
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Factor of Line Snap

Use this command to calculate a point at a factored distance along a line. You can enter any multiplication factor greater than zero (for example, 0.5 = 50% from the end of the line). If the line has a slope, the Z value is interpolated.

**Factor of Line Snap** can be used in these commands:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



### To use a Factor of Line Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Factor > Factor of Line Snap** from the context menu. The **Factor of Line Snap** command pane displays.
2. Pick a line near the end from which you want to factor the distance. The line is recorded in the **Line** box.
3. Type a factor in the **Multiplication Factor** box.
4. Click **OK**. The coordinate is recorded, and the command pane returns to the previous command.

### Related topics

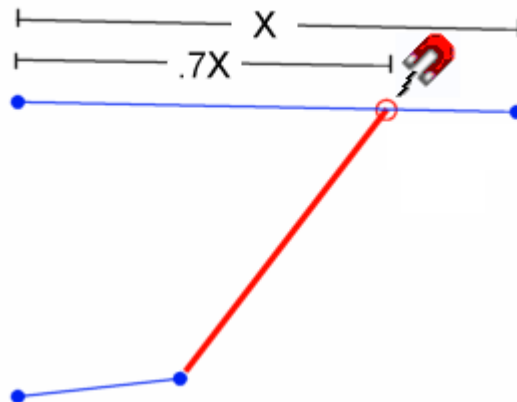
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Factor of Segment Snap

Use this command to calculate a point at a factored distance along a line segment. You can enter any multiplication factor greater than zero (for example, 0.5 = 50% from the end of the line). Zero (0) snaps to the end of the line closest to where you pick, and 1 snaps to the furthest end of the line. If the line has a slope, the Z value is interpolated.

**Factor of Segment Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use a Factor of Segment Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Factor > Factor of Segment Snap** from the context menu. The **Factor of Segment Snap** command pane displays.
2. Pick a line segment near the end from which you want to factor the distance. The segment is recorded in the **Segment** box.
3. Type a factor in the **Multiplication Factor** box.
4. Click **OK**. The coordinate is calculated, and the command pane returns to the previous command.

#### Related topics

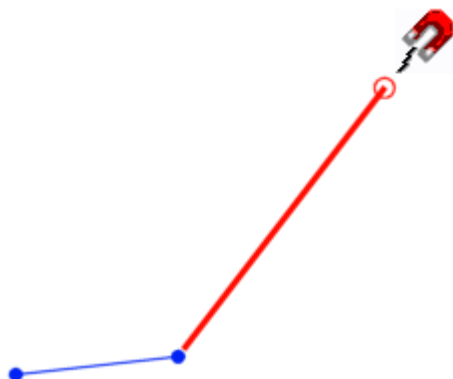
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### Free Snap

Use this command to use the coordinates of any point picked in a view. No elevation is set using this option.

**Free Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use a Free Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Free Snap** from the context menu. The **Free Snap** command pane displays.
2. Pick a point, or type a coordinate or point ID in the **Free Point** box. The command pane returns to the previous command.

**Note:** **Free Snap** is one of the default running snap modes, so unless you have it disabled in the **Snap Mode** dialog, you don't need to explicitly select **Free Snap** from the context menu. Use this option when you have multiple running snaps active and you don't want to snap to a point, end point, etc.

#### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Intersection of Lines Snap

Use this command to calculate the intersection (or projected intersection) of two lines. If there are several possible intersections, the intersection closest to where the lines were picked is used. No elevation is set using this option.

**Intersection of Lines Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)

#### To use an Intersection of Lines Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Intersection > Intersection of Lines Snap** from the context menu. The **Intersection of Lines Snap** command pane displays.
2. Pick a line in the view. The line is recorded in the **Line 1** box.

- Pick another line. The line is recorded in the *Line 2* box. The coordinate is recorded, and the command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Intersection of Offset Segments Snap

Use this command to calculate the intersection of offsets from two selected line or arc segments. From the end of the segment closest to where it was picked, looking to the other end, positive offsets are to the right and negative offsets are to the left of the segment.

**Intersection of Offset Segments Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)

### To use an Intersection of Offset Segments Snap:

- While in a coordinate control, right-click in a graphic view, and select **Intersection > Intersection of Offset Segments Snap** from the context menu. The *Intersection of Offset Segments Snap* command pane displays.
- Pick a line or arc segment in the view. The segment is recorded in the *Line segment 1* box.
- Pick a point offset from the line, or type a value in the *Offset 1* box. For the offsets, you can enter a negative or positive offset distance from the line.
- Pick a second line or arc segment. The segment is recorded in the *Line segment 2* box.
- Pick a point offset from the second line, or type a value in the *Offset 2* box. You can enter a negative or positive offset distance from the line. The coordinate is recorded, and the command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Intersection of Offset Lines Snap

Use this command to calculate the intersection of offsets from two lines or arcs. The offsets are projected from the entire lines or arcs, rather than just line or arc segments. From the end of the line closest to where it was picked, looking to the other end, positive offsets are to the right and negative offsets are to the left of the line.

You can also calculate the intersection of offsets from two horizontal alignments (HALs).

**Intersection of Offset Lines Snap** is used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)

#### To use an Intersection of Offset Lines Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Intersection > Intersection of Offset Lines Snap** from the context menu. The **Intersection of Offset Segments Snap** command pane displays.
2. Pick a line in the view. The line is recorded in the **Line 1** box.
3. Pick a point offset from the line, or type a value in the **Offset 1** box. For the offsets, you can enter a negative or positive offset distance from the line.
4. Pick a second line. The line is recorded in the **Line 2** box.
5. Pick a point offset from the second line, or type a value in the **Offset 2** box. The coordinate is recorded, and the command pane returns to the previous command.

#### Related topics

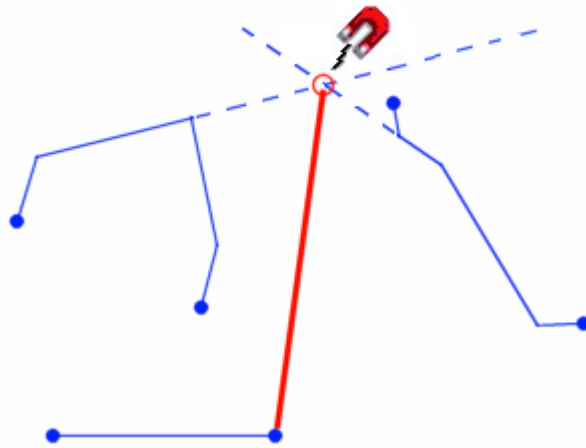
- [Understanding COGO Controls](#) (on page 86)
- [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- [Snaps Modes and Commands](#) (on page 89)

### Intersection of Segments Snap

Use this command to calculate the projected intersection of two line or arc segments. If there are several possible intersections, the intersection closest to where the lines were picked is used.

**Intersection of Segments Snap** can be used in these commands:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use an Intersection of Segments Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Intersection > Intersection of Segments Snap** from the context menu. The **Intersection of Segments Snap** command pane displays.
2. Pick a line segment in the view. The segment is recorded in the **Line segment 1** box.
3. In the view, pick another line segment. The segment is recorded in the **Line segment 2** box. The coordinate is recorded, and the command pane returns to the previous command.

#### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### Middle of Point to Point Snap

Use this command to calculate the midpoint between two points.

**Middle of Point to Point Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)

#### To use a Middle of Point to Point Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Middle of Point to Point Snap** from the context menu. The **Middle of Point to Point Snap** command pane displays.
2. Pick a point in the view, or type a coordinate or point ID in the **From point** box.

- Pick a point, or type a coordinate or point ID in the *To point* box. The midpoint between the two selected points is recorded, and the command pane returns to the previous command.

### Related topics

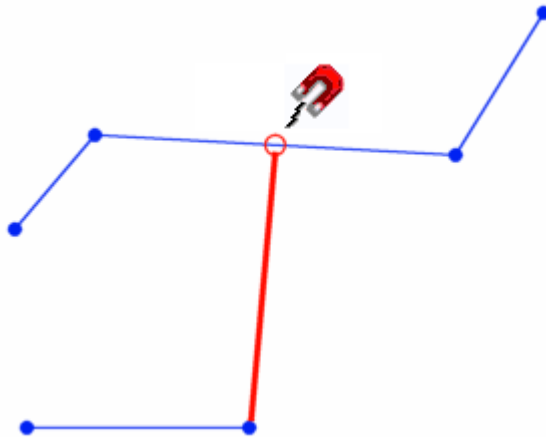
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Middle of Segment Snap

Use this command to calculate the middle point of a line or arc segment. If the line segment has a slope, the Z value is interpolated from the segment.

**Middle of Segment Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



### To use a Middle of Segment Snap:

- While in a coordinate control, right-click in a graphic view, and select **Middle of Segment Snap** from the context menu. The **Middle of Segment Snap** command pane displays.
- Pick a line segment in the view. The coordinate is recorded, and the command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

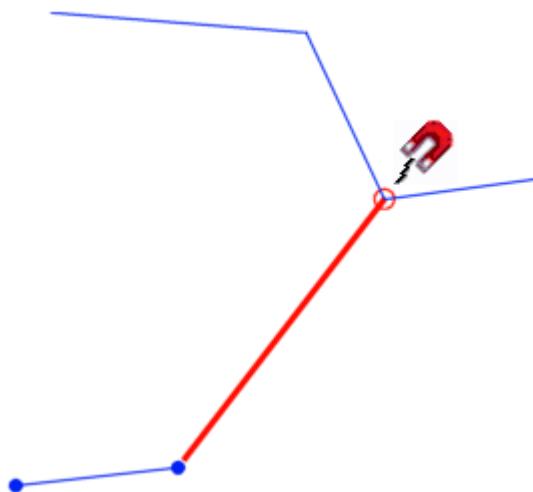


## Nearest to Line Snap

Use this command to calculate the closest point on a line within the pick aperture to the point you pick. If the point is on a line with a slope, the elevation value is interpolated.

**Nearest to Line Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



### To use a Nearest to Line Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Nearest to Line Snap** from the context menu. The **Nearest to Line Snap** command pane displays.
2. Pick a line in the view. The coordinates on the line nearest to your pick location are recorded, and the command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Offset Line Snap

Use this snap command to calculate a point based on an offset from a station along the entire length of a line or an arc. Station zero is the end of the line closest to where it was picked. From station zero, looking down the line, positive offsets are to the right and negative offsets are to the left of the line. If the line has a sloping elevation, the Z value is interpolated from the station used.

**Note:** You can use this option to select a specific position along an arc by making the offset distance zero.

**Offset Line Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)

#### To use an Offset Line Snap:

1. While in a coordinate control, right-click in a graphics view, and select **Offset > Offset Line Snap** from the context menu. The **Offset Line Snap** command pane displays.
2. Pick a line in the view. The line is recorded in the **Line** box.
3. Pick a point, or type a value in the **Station** box.
4. Pick a point, or type a value in the **Offset** box. You can enter a negative or positive offset distance from the line. The coordinate is recorded, and the command pane returns to the previous command.

#### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Offset Segment Snap

Use this command to calculate a point based on an offset from a station along a line or arc segment. Station zero is the end of the line closest to where it was picked. From station zero, looking down the line, positive offsets are to the right and negative offsets are to the left of the segment.

**Offset Segment Snap** can be used in these controls:

[Coordinate](#) (see "Enter a Coordinate" on page 97)

#### To use an Offset Segment Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Offset > Offset Segment Snap** from the context menu. The **Offset Segment Snap** command pane displays.
2. Pick a line or arc segment in the view. The segment is recorded in the **Line segment** box.
3. Pick a point, or type a value in the **Station** box. You can enter a negative or positive offset distance from the segment.
4. Type a value in the **Offset** box. The coordinate is calculated, and the command pane returns to the previous command.

**Tip:** You can use this option to select a specific position along an arc by making the offset distance zero (0).

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Perpendicular to Segment Snap

Use this command to calculate a perpendicular intersection from a point to a line segment or arc segment. If the line segment has an elevation, the Z value will be interpolated from the segment.

**Perpendicular to Segment Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)

### To use a Perpendicular to a Segment Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Perpendicular > Perpendicular to Segment Snap** from the context menu. The **Perpendicular to Segment Snap** command pane displays.
2. Pick a line segment in the view. The line is recorded in the **Line segment** box.
3. Pick a point or type a coordinate in the **Reference point** box. The coordinate is recorded, and the command pane returns to the previous command.

### Related topics

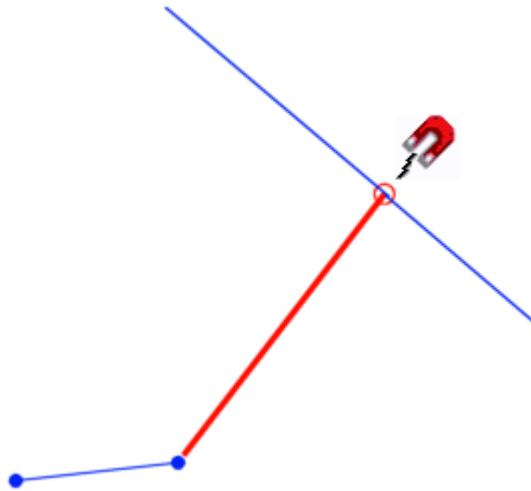
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Perpendicular to Line Snap

Use this command to calculate a perpendicular intersection from a point to a line or arc. The entire length of the line or arc is used in the calculation. You can also calculate a perpendicular intersection of a point to a horizontal alignment (HAL).

**Perpendicular to Line Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use a Perpendicular to Line Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Perpendicular > Perpendicular to Line Snap** from the context menu. The **Perpendicular to Line Snap** command pane displays.
2. Pick a line or a HAL in the view. The line or HAL is recorded in the **Line** box.
3. Pick a point, or type a coordinate in the **Reference point** box. The coordinate is recorded, and the command pane returns to the previous command.

#### Related topics

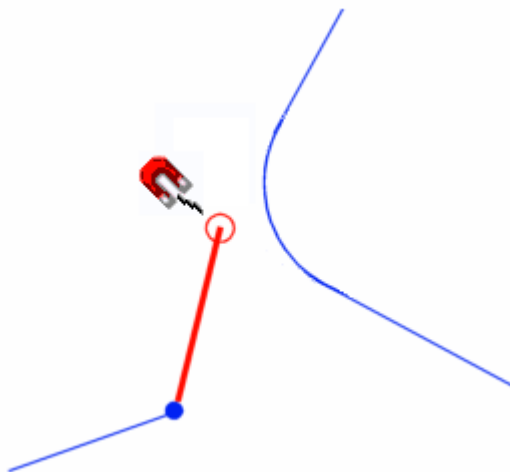
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### Point of Intersection Snap

Use this command to calculate the point of intersection (PI) of a selected arc, spiral, or parabola. No elevation is set using this option.

**Point of Intersection Snap** can be used in these commands:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use a Point of Intersection Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Point of Intersection Snap** from the context menu. The **Point of Intersection Snap** command pane displays.
2. Pick a spiral, arc, or parabola in the view. The intersection coordinates are recorded, and the command pane returns to the previous command.

#### Related topics

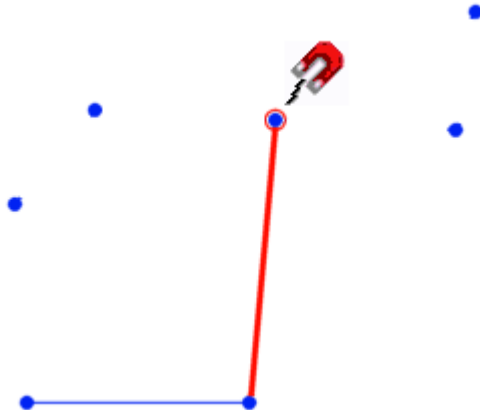
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### Point Snap

Use this command to specify the coordinates of a point ID. The elevation of the point is used as the Z value.

**Point Snap** can be used in these controls:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



### To use a Point Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Point Snap** from the context menu. The **Point Snap** command pane displays.
2. Pick a point in the view, or type a point name in the **Point ID** box. The coordinates are recorded, and the command pane returns to the previous command.

### Related topics

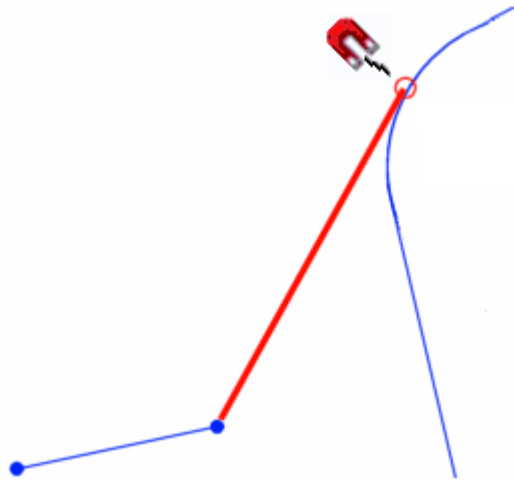
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Tangent Snap

Use this command to calculate a point on an arc tangent to a previous point. No elevation is set using this option.

**Tangent Snap** can be used in these commands:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use a Tangent Snap:

1. While in a coordinate control, right-click in a graphic view, and select **Tangent Snap** from the context menu. The *Tangent Snap* command pane displays.
2. Pick a point, or type a coordinate or point ID in the *Reference Point* box.
3. Pick an arc. The coordinates on the arc tangent to the reference point are recorded, and the command pane returns to the previous command.

#### Related topics

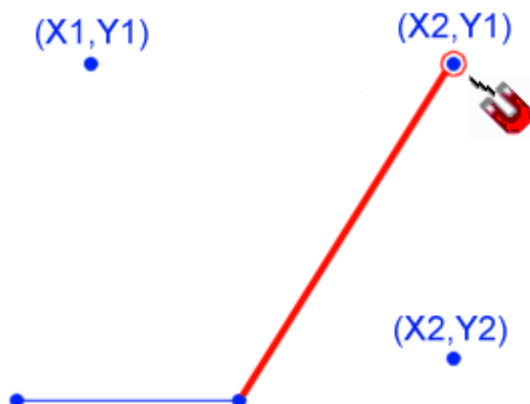
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### XY Snap

Use this command to calculate a point based on the easting (Y-coordinate) of the first point you pick, and then to the northing (X-coordinate) of the second point you pick. If the second point has an elevation, it will be used for the Z value.

**XY Snap** can be used in these commands:

- [Coordinate](#) (see "Enter a Coordinate" on page 97)



#### To use an XY Snap:

1. While in a coordinate control, right-click in a graphic view, and select **XY Snap** from the context menu. The **XY Snap** command pane displays.
2. Pick a point, or type a coordinate or point ID in the **Easting of point** box.
3. Pick a point, or type a coordinate or point ID in the **Northing of point** box. The coordinate is recorded, and the command pane returns to the previous command.

#### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Enter a Distance

Use the Distance control to specify a distance. Typically, this is a horizontal distance. When calculating 3D points, the horizontal distance can also be a slope distance (SD) and you will be alerted as such. "Distance" appears in the status bar when this control is active.

Distance control is used in these commands:

- [Bearing Distance Snap](#) (on page 100)
- [Distance Distance Snap](#) (on page 103)
- [DxDy Snap](#) (see "Delta X Delta Y Snap" on page 103)
- [Distance + Distance Snap](#) (see "Distance + Distance Snap (distance)" on page 121)
- [Factor of Distance Snap](#) (see "Factor of Distance Snap (distance)" on page 122)

Distance control gives you access to these commands on the context menu:



- [Distance + Distance Snap](#) (see "Distance + Distance Snap (distance)" on page 121)
- [Factor of Distance Snap](#) (see "Factor of Distance Snap (distance)" on page 122)
- [Point to Point Snap](#) (see "Point to Point Snap (distance)" on page 123)
- [Radius of Arc Snap](#) (see "Radius of Arc Snap (distance)" on page 124)

#### To specify a distance:

- Pick a point in a graphic view. With the cursor "rubber-banding", pick another point to specify the distance.
- Pick a line or line segment; the length of the segment will be used as the distance.
- Type a value in the **Distance** box to specify the horizontal distance or slope distance (SD), using any of the standard entry formats. Check and set the entry format in the **Project Settings** by selecting **Project > Project Settings > Units > Distance**.

**Note:** When specifying a distance and slope from a point in the profile view, the distance is always horizontal.

- Type a point ID to point ID (e.g. 1>2), to recall the distance between two points.
- Type a distance using mathematical operators or an expression to calculate the distance. The valid operators are + - \* / and () for expressions.
- Right-click in a graphic view, select a snap command from the context menu, and specify the necessary parameters.

**Note:** In commands where the direction is not determined, the cursor will "rubber band" as a circle. The center point is the anchor point, and the circle is drawn through the second point.

#### Related topics

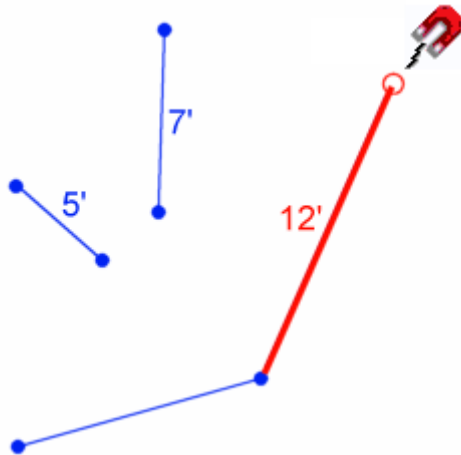
- [COGO Controls](#) (see "Understanding COGO Controls" on page 86)
- [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- [Snaps Modes and Commands](#) (on page 89)

### Distance + Distance Snap (distance)

Use this command to calculate a distance by adding two distances. Either distance can be negative.

**Distance + Distance Snap** can be used in these controls:

- [Distance](#) (see "Enter a Distance" on page 120)
- [Offset](#) (see "Enter an Offset" on page 128)



#### To use a Distance + Distance Snap:

1. While in a distance or offset control, right-click in a graphic view, and select **Distance + Distance Snap** from the context menu. The **Distance + Distance Snap** command pane displays.
2. Pick two points or a line segment, or type a value in the **Distance 1** box.
3. Pick two more points or a line segment, or type a value in the **Distance 2** box. The distance is recorded, and the command pane returns to the previous command.

#### Related topics

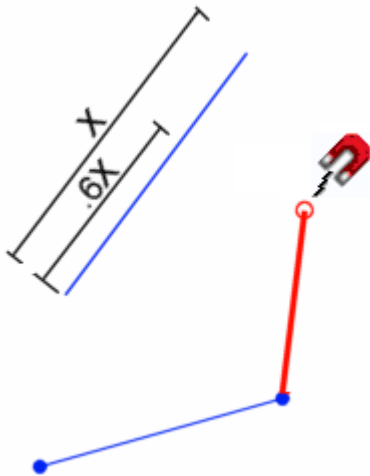
- [Understanding COGO Controls](#) (on page 86)
- [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- [Snaps Modes and Commands](#) (on page 89)

#### Factor of Distance Snap (distance)

Use this command to calculate a distance by applying a multiplication factor to a distance.

**Factor of Distance Snap** can be used in these controls:

- [Distance](#) (see "Enter a Distance" on page 120)
- [Offset](#) (see "Enter an Offset" on page 128)



#### To use a Distance Factor Snap:

1. While in a distance or offset control, right-click in a graphic view, and select **Factor of Distance Snap** from the context menu. The **Factor of Distance Snap** command pane displays.
2. Pick two points, or a line segment in the view, or type a value in the **Distance** box.
3. Type a factor value greater than zero (>0) in the **Multiplication Factor** box. For example, type 2 for 200% of the distance.
4. Click **OK**. The distance is recorded, and the command pane returns to the previous command.

#### Related topics

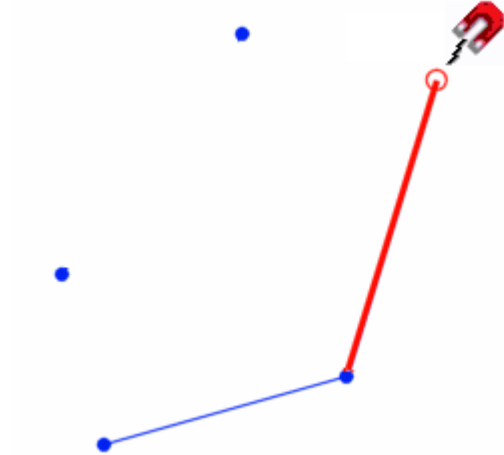
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### Point to Point Snap (distance)

Use this command to calculate the distance between two points.

**Point to Point Snap** can be used in these controls:

- [Bearing](#) (see "Enter a Bearing" on page 94)
- [Distance](#) (see "Enter a Distance" on page 120)
- [Offset](#) (see "Enter an Offset" on page 128)



#### To use a Point to Point Snap:

1. While in a distance control, right-click in a graphic view, and select **Point to Point Snap** from the context menu. The **Point to Point Snap** command pane displays.
2. Pick a point in the view, or type a coordinate or point ID in the **From point** box.
3. Pick a point, or type a coordinate or point ID in the **To point** box. The distance is recorded, and the command pane returns to the previous command.

#### Related topics

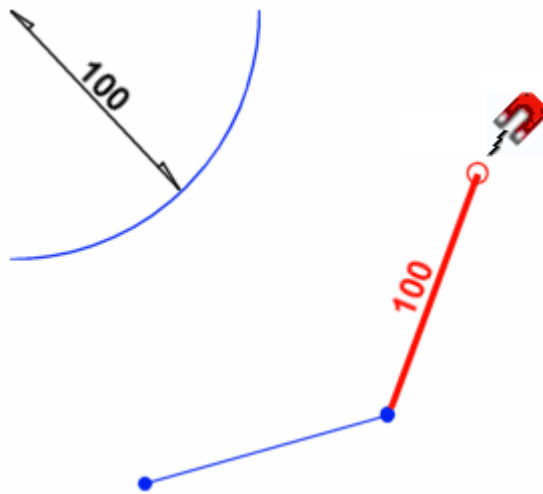
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

#### Radius of Arc Snap (distance)

Use this command to calculate a distance based on the radius of an arc.

**Radius of Arc Snap** can be used in these controls:

- [Distance](#) (see "Enter a Distance" on page 120)
- [Offset](#) (see "Enter an Offset" on page 128)



#### To use a Radius of Arc Snap:

1. While in a distance or offset control, right-click in a graphic view, and select **Radius of Arc Snap** from the context menu. The **Radius of Arc Snap** command pane displays.
2. Pick an arc in the view. The arc's radius is recorded, and the command pane returns to the previous command.

#### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Enter an Elevation

Use the Elevation control to specify an elevation. You can snap to any 3d object to specify the elevation. "Elevation" appears in the status bar when the control is active.

Elevation control gives you access to these commands on the context menu:

- [Elevation Undefined Snap](#) (see "Undefined Snap" on page 126)
- [From Surface Snap](#) (on page 126)

#### To specify an elevation:

- Pick an object in a graphic view to use its elevation. If you pick a line, the elevation will be interpolated along the slope of the line, based on the position you pick.
- Type a value or a point ID in the *Elevation* box.
- Right-click in a graphic view, select a snap command from the context menu, and specify the necessary parameters.

#### Related topics

- [COGO Controls](#) (see "Understanding COGO Controls" on page 86)
- [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- [Snaps Modes and Commands](#) (on page 89)

### Undefined Snap

Use this command to specify that the elevation is undefined. Use it when no elevation is required.

**Undefined Snap** can be used in these controls:

- [Elevation](#) (see "Enter an Elevation" on page 125)

#### To use an Undefined Snap:

1. While in an elevation control, right-click in a graphic view, and select **Undefined Snap** from the context menu. The *Elevation* box is filled with a ? to show that the elevation is valid as undefined.
2. Click **OK**, if necessary.

#### Related topics

- [Understanding COGO Controls](#) (on page 86)
- [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- [Snaps Modes and Commands](#) (on page 89)

### From Surface Snap

Use this command to calculate the elevation of a point on a 3D surface. The elevation of the X,Y position on the surface will be used as the Z value.

**From Surface Snap** can be used in these controls:

- [Elevation](#) (see "Enter an Elevation" on page 125)

#### To use a From Surface Snap:

1. While in an elevation control, right-click in a graphics view, and select **Point to Point Snap** from the context menu. The **Point to Point Snap** command pane displays.
2. Pick a surface in the view, or select one from the **Surface** list.
1. Pick a point on the selected surface, or type a coordinate in the **Point** box. The elevation at that point on the surface is recorded, and the command pane returns to the previous command, if necessary.

#### Related topics

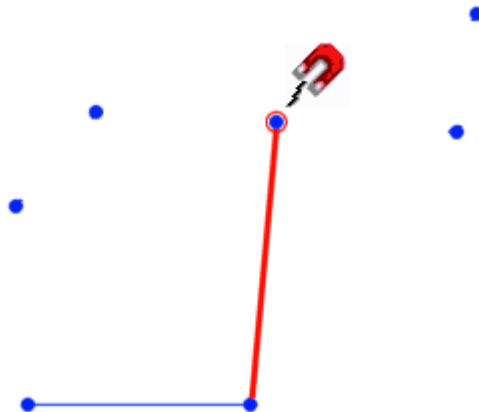
- [Understanding COGO Controls](#) (on page 86)
- [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- [Snaps Modes and Commands](#) (on page 89)

### At Point Snap

Use this command to specify an elevation using the elevation of a named point .

**At Point Snap** can be used in these controls:

- [Elevation](#) (see "Enter an Elevation" on page 125)



#### To use an At Point Snap:

1. While in an elevation control, right-click in a graphic view, and select **At Point Snap** from the context menu. The **At Point Snap** command pane displays.
2. Pick a named point in the view, or type a point ID in the **Point** box. The elevation of the point is recorded, and the command pane returns to the previous command.

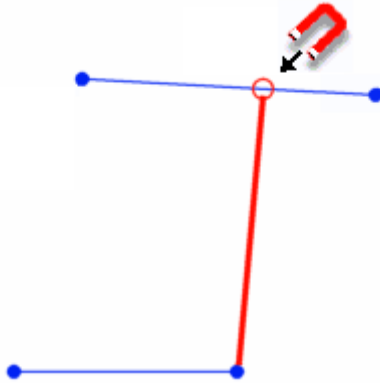
#### Related topics

- [Understanding COGO Controls](#) (on page 86)
- [COGO Expressions, Units, and Entry Formats](#) (on page 87)

- ❑ [Snaps Modes and Commands](#) (on page 89)

## Along Line Snap

Use this command to specify an elevation using the elevation at a location along a 3D line.



**Along Line Snap** can be used in this control:

- [Elevation](#) (see "Enter an Elevation" on page 125)

### To use an Along Line Snap:

1. While in an elevation control, right-click in a graphic view, and select **Along Line Snap** from the context menu. The **Along Line Snap** command pane displays.
2. Pick a 3D line in the view. A "rubber-band" line appears between your cursor and the line.
3. Using the rubber-band line, pick a point along the line, or type a distance from the beginning point of the line (in station format) in the **Distance Along** box. The elevation at that location is recorded, and the command pane returns to the previous command, if necessary.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Enter an Offset

Use the Offset control to specify an offset from a given line. "Offset" appears on the status bar when the control is active.

Offset control is used in these commands:



- [Offset Line Snap](#) (on page 113)
- [Offset Segment Snap](#) (on page 114)

Offset control gives you access to these commands on the context menu:

- [Distance + Distance Snap](#) (see "Distance + Distance Snap (distance)" on page 121)
- [Distance Snap](#) (on page 130)
- [Factor of Distance Snap](#) (see "Factor of Distance Snap (distance)" on page 122)
- [Point ID Snap](#) (see "Offset at Point Snap" on page 132)
- [Point to Point Snap](#) (see "Point to Point Snap (distance)" on page 123)
- [Radius of Arc Snap](#) (see "Radius of Arc Snap (distance)" on page 124)

#### To specify an offset:

- Pick a point in a graphic view to define the coordinate through which the offset will pass. The control uses the perpendicular distance from the line to the specified position to calculate the offset distance.
- Type a distance value in the **Offset** box. From the end of the line closest to where it was picked, looking to the other end, positive offsets are to the right and negative offsets are to the left of the line.
- Right-click in a graphic view, select a snap command from the context menu, and specify the necessary parameters. The **At Point Snap** command gives you the ability to use any of the point snap modes to specify the offset distance from a specific point to the line.

#### Related topics

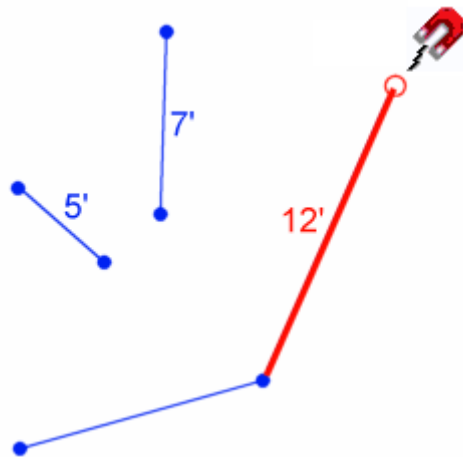
- [COGO Controls](#) (see "Understanding COGO Controls" on page 86)
- [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- [Snaps Modes and Commands](#) (on page 89)

### Distance + Distance Snap (offset)

Use this command to calculate an offset by adding two distances. Either distance can be negative.

**Distance + Distance Snap** can be used in these controls:

- [Distance](#) (see "Enter a Distance" on page 120)
- [Offset](#) (see "Enter an Offset" on page 128)



#### To use a Distance Plus Distance Snap:

1. While in a distance or offset control, right-click in a graphic view, and select **Distance + Distance Snap** from the context menu. The **Distance + Distance Snap** command pane displays.
2. Pick two points or a line segment, or type a value in the **Distance 1** box.

#### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
  - ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
  - ❑ [Snaps Modes and Commands](#) (on page 89)
1. Pick two more points or a line segment, or type a value in the **Distance 2** box. The total distance is recorded, and the command pane returns to the previous command.

### Distance Snap

Use this command to calculate an offset based on the distance between two points, or based on the length of a line segment..

**Distance Snap** can be used in these controls:

- [Offset](#) (see "Enter an Offset" on page 128)

#### To use a Distance Snap:

1. While in an offset control, right-click in a graphic view, and select **Distance Snap** from the context menu. The **Distance Snap** command pane displays.

- Pick two points or a line segment in the view, or type a value in the **Distance** box. The distance or length of the line segment is recorded.
- Click **OK**. The command pane returns to the previous command.

### Related topics

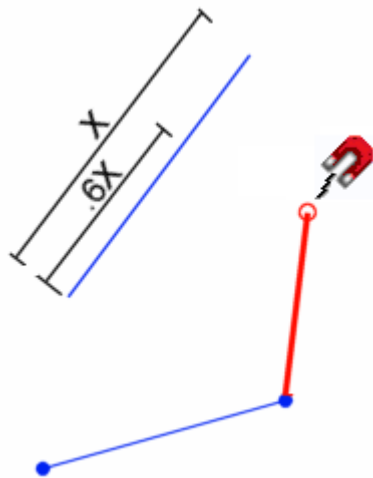
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Factor of Distance Snap (offset)

Use this command to calculate an offset by applying a multiplication factor to a distance.

**Factor of Distance Snap** can be used in these controls:

- [Distance](#) (see "Enter a Distance" on page 120)
- [Offset](#) (see "Enter an Offset" on page 128)



### To use a Factor of Distance Snap:

- While in a distance or offset control, right-click in a graphic view, and select **Factor of Distance Snap** from the context menu. The **Factor of Distance Snap** command pane displays.
- Pick two points or a line segment in the view, or type a value in the **Distance** box.
- Type a factor value greater than zero ( $>0$ ) in the **Multiplication Factor** box. For example, type 2 for 200% of the distance.
- Click **OK**. The offset distance is recorded, and the command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Offset at Point Snap

Use this command to specify an offset from a line by picking a point at a distance away from the line.

**Offset at Point Snap** is used in these controls:

- [Offset](#) (see "Enter an Offset" on page 128)

#### To use an Offset at Point Snap:

1. While in an offset control, right-click in a graphic view, and select **Offset at Point Snap** from the context menu. The **Offset at Point** command pane displays.
2. Pick a point in the view, or type a point ID or coordinate in the **Point** box using one of the standard formats. The command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

### Point to Point Snap (offset)

Use this command to compute the offset between one point and another point.

**Point to Point Snap** can be used in these controls:

- [Offset](#) (see "Enter an Offset" on page 128)

#### To use a Point to Point Snap:

1. While in an offset control, right-click in a graphic view, and select **Point to Point Snap** from the context menu. The **Point to Point Snap** command pane displays.
2. Pick a point in the view, or type a coordinate or Point ID in the **From point** box.
3. Pick a point, or type a coordinate in the **To point 2** box. The offset distance is recorded, and the command pane returns to the previous command.

### Related topics

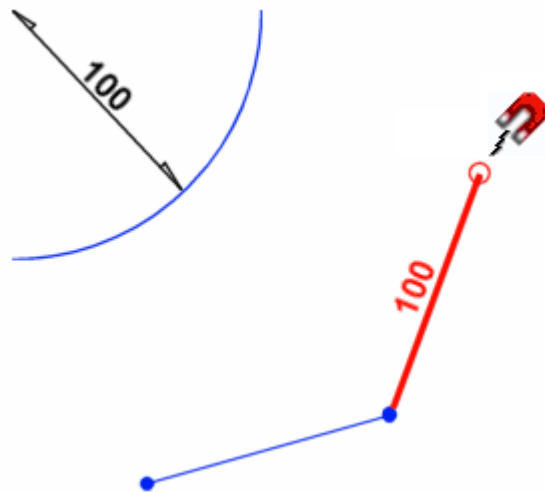
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Radius of Arc Snap (offset)

Use this command to calculate an offset based on the radius of an arc.

**Radius of Arc Snap** can be used in these controls:

- [Distance](#) (see "Enter a Distance" on page 120)
- [Offset](#) (see "Enter an Offset" on page 128)



### To use a Radius of Arc Snap:

1. While in a control, right-click in a graphic view, and select **Radius of Arc Snap** from the context menu. The **Radius of Arc Snap** command pane displays.
2. Pick an arc in the view. The arc's radius is recorded, and the command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Enter a Station

Use the Station control to specify a station on a selected alignment. The station is calculated from the specified point to a perpendicular point on the alignment. "Station" appears on the status bar when the control is active.

Station control is used in these commands:

- [Offset Line Snap](#) (on page 113)
- [Offset Segment Snap](#) (on page 114)

Station control gives you access to this command on the context menu:

- [Station at Point Snap](#) (on page 134)
- [Station From Line Snap](#) (on page 135)

### To specify a station:

- Pick a point along an alignment in a graphic view.

**Note:** When the station control is active the cursor will "rubber-band" from a horizontal alignment (HAL).

- Type a value in the **Station** box, using any of the standard entry formats. Check and set the entry format in the **Project Settings** by selecting **Project > Project Settings > Units > Station**.

If you have defined an alignment and station equations, you can enter the station and segment. If there is more than one roadway that uses the alignment, only the first one found is used. An alignment with two station equations has three segments: one before the first equation, one between the two equations and one after the second equation. To define the station and segment enter "(station):(segment)" e.g., 14000 : 3. If no segment is specified, the first segment containing the specified station will be used. You can also specify an absolute distance along an alignment, regardless of station equations or beginning station, by typing zero (0) for the segment number, e.g., 14000 : 0.

- Right-click in a graphic view, select a snap command from the context menu, and specify the necessary parameters.

### Related topics

- ❑ [COGO Controls](#) (see "Understanding COGO Controls" on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Station at Point Snap

Use this command to calculate a station based on a point. You must pick the point in a profile view.

**Station at Point Snap** is used in these controls:

- [Station](#) (see "Enter a Station" on page 133)

### To use a Station at Point Snap:

1. While in a station control, right-click in a graphic view, and select **Station at Point Snap** from the context menu. The **Station at Point Snap** command pane displays.
2. Pick a point in a profile view, or type a coordinate or point ID in the **Point** box. The coordinate or point ID is recorded, and the command pane returns to the previous command.

### Related topics

- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Station from Line Snap

Use this command to calculate a station value from a point perpendicular to an alignment or line.

**Station from Line Snap** is used in these controls:

- [Station](#) (see "Enter a Station" on page 133)

### To use a Station from Line Snap:

1. While in a station control, right-click in a graphic view, and select **Station from Line Snap** from the context menu. The **Station from Line Snap** command pane displays.
2. Pick a line in a graphic view.
3. Pick a point along the line, or type a value in the **Station** box, to specify a station. The station is recorded, and the command pane returns to the previous command.

### Related topics


- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Snaps Modes and Commands](#) (on page 89)

## Measure Values Between Points

Calculate and report values between points in your project.

- In the plan view, the command measures bearing and distance.
- In the profile view, it measures the delta station, slope, and slope distance.
- In the cross-section and surface slicer views, it measures delta offset, slope, and slope distance.

### To measure values between two points:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Tools > Measure**.

The **Measure** command pane displays.
2. Pick the first point in a graphic view or type a point ID or coordinate (in the format X,Y) in the **From** box.

**Note:** You can also right-click in the view to access [COGO controls](#) (see "Understanding COGO Controls" on page 86) and [snaps](#) (see "Snaps Modes and Commands" on page 89) when picking points.

3. Pick another point or type a point ID or coordinate in the *To* box. The measured values appear in the *Results* group.
4. To measure other values, continue picking *From* and *To* points.

**Note:** You can change to a different graphic view between measurements.

5. Click **Close**.

### Related topics

- ❑ [Measure Options](#) (on page 136)
- ❑ [Calculate the Inverse Between Points](#) (on page 230)
- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Measure Options

Use these options to calculate and report the bearing, distance, slope, slope distance, delta offset, and delta station between any two points, depending on which graphic view you use. They are available in the *Measure* command pane.

### Options

<b>From/To</b>	Pick points in graphic views, or type point IDs or coordinates (in the format X,Y) in the boxes, and click <b>Measure</b> or press <b>[Enter]</b> .
<b>Results</b>	This shows the values between the selected points:  <b>Slope</b> - In the cross-section view, the slope is relative to the centerline. In the surface slicer view, the slope is relative to the first point of the line.  <b>Offset</b> - In the cross-section view, the offset is relative to the centerline. In the surface slicer view, the offset is relative to the first point of the line.
<b>Measure</b>	When you type in <i>To</i> and <i>From</i> points, this acts as the <b>[Enter]</b> key, calculating the <i>Results</i> .

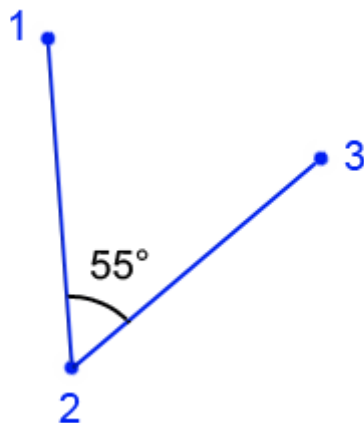
### Related topics

- ❑ [Measure Values Between Points](#) (on page 135)



## Measure Angles

Measure the angle between three selected locations and/or named points (points with IDs).



### To measure an angle:

1. Select **Survey > Measure Angle**. The *Measure Angle* command pane displays.
2. Verify the correct measurement type is selected in the *Options* box.
3. Click in the *Start point* box and select the point or location in *Plan* view. The point or coordinate you selected is displayed in the *Start point* box and the cursor moves to the *Pivot point* box.
4. Select the pivot point or location in *Plan* view. The point or coordinate you selected is displayed in the *Pivot point* box and the cursor moves to the *End point* box.
5. Select the end point or location in *Plan* view. The point or coordinate you selected is displayed in the *End point* box and the measure angle is displayed in the *Result* box.

You can continue to select different points or locations for the end point to display additional angles in the *Result* box.

**Note:** As an alternative to selecting points and/or locations in the *Plan View*, you can type point IDs or coordinates directly in the point boxes.

You can click the **Reset** icon located at the top of the pane to clear all measurements and results.

You can click the **Clear** button located beneath the *Result* box to remove all angles displayed in the *Result* box.

### Related topics

- [Measure Angle Options](#) (on page 138)

## Measure Angle Options

Use these options to measure the angle between three selected locations and/or named points (points with IDs). They are available in the **Measure Angle** command pane.

### Options

<b>Start point</b>	Click in the box and then select the start point in <b>Plan View</b> , or type the point ID or coordinate in the box.
<b>Pivot point</b>	<p>If you selected the start point by selecting it in <b>Plan View</b>, the cursor automatically moves to this box. If you typed the point ID or coordinate in the <b>Start point</b> box, click in this box.</p> <p>Select the pivot point in <b>Plan View</b>, or type the point ID or coordinate in the box.</p>
<b>End point</b>	<p>If you selected the pivot point by selecting it in <b>Plan View</b>, the cursor automatically moves to this box. If you typed the point ID or coordinate in the <b>Pivot point</b> box, click in this box.</p> <p>Select the end point in <b>Plan View</b>, or type the point ID or coordinate in the box.</p> <p>The measured angle is displayed in the <b>Result</b> box.</p> <p>You can continue to select different points or locations for the end point to display additional angles in the <b>Result</b> box.</p>
<b>Options</b>	Select the appropriate measurement method.
<b>Result</b>	Displays the measured angle.

### Related topics

- [Measure Angles](#) (on page 137)

## CHAPTER 4

# Set Up Projects

## Choose Application Options

Use the *Options* dialog to set program-wide options, such as startup preferences, default file locations, Internet download parameters, and display properties.

### Related topics

- ❑ [Choose Local Site Settings](#) (on page 159)
- ❑ [Choose Project Settings](#) (on page 147)

## Startup and Display Options

Set startup and display options to control whether a project or start page automatically open when you start the program, and to specify program-wide preferences for the graphical user interface.

### To access these settings:

1. Select **Tools > Options**. The *Options* dialog displays.
2. Click **Startup and Display** in the left pane.

These options are available:

## Startup options

### Starting state

**No project** - Select this to have no projects and no start page open when the software starts.


**Last project** - Select this to open only the most recent project when the software starts.

**Open project command** - Select this to initiate the **Open Project** command, which enables you to select any project to open, when the software starts.

**Display start page** - Select this to display only the start page when the software starts.

The start page is a view that lists features of the software and links to tours, tutorials, release notes, and other documentation to help you get started. By default, the start page appears when you launch the software, and closes when you start or open a project.

### Start page

Type a path or click the  button to navigate to an alternate HTML page to use as the start page, which will display when you have **Display start page** selected in the **Starting State** box (see above).

### Recently-used file list

Enter the number of recent project files to list at the bottom of the **File** menu, allowing you quick access to these projects.

## Graphics window options

### Display data tips

Check this box to display more detailed information when you hover over objects in a graphic view.

---

**Note:** This may conflict with the data list that appears when you graphically select an area with multiple objects.

---

### Background color

Click an option to set the color of the background used in the graphics view.

### Pick aperture

Enter a size (in pixels per side) for the box displayed on various cursors. This is the box within which an object can be picked.

Examples of cursors with pick apertures:



## Application display options

- Window display mode**
- Tabbed views (SDI)** - Select this to display each created data view (for example, a plan view or point spreadsheet) as a tabbed pane, enabling you to access each by clicking a tab, and allowing views to be split. In addition, data listings can be "floated", meaning that they can be displayed as undocked, movable windows.
  - Multiple window views (MDI)** - Select this to display each created data view in a separate window, allowing windows to be tiled or cascaded.

### Related Topics

- ❑ [Choose Application Options](#) (on page 139)
- ❑ [File Location Options](#) (on page 142)
- ❑ Internet Download Options

### Advanced Display Options

Use these options to control what graphics display package the program uses. In certain situations, choosing to use a different display package can improve graphics performance when you are exploring rich data sets in the **Plan View** (on page 25) and **3D View** (on page 28).

**Note:** Generally, you only need to override your graphics display package if you are accessing (via Remote Desktop or Microsoft® NetMeeting®) this program on your computer from a remote computer that uses a different graphics package.

### To access these settings:

1. Select **Tools > Options**. The *Options* dialog displays.
2. Click **Startup and Display** in the left pane.
3. Click **Advanced** in the *Graphics Window Options* group.

These options are available:

## Advanced display options

### Graphics display package

**Override automatic detection** - Check this box if you want to manually select a graphics display package other than the one detected on your computer. Then, select from the options below.

**DirectX** - Select this if you are using Microsoft DirectX® 9.0c (Runtime version 4.09.0000.0904) distributed on June 2007 or later on Windows® XP Service Pack 2 or Windows Vista. The driver should run properly with any Direct3D 9.0 compliant graphics hardware with the latest vendor-supplied drivers.

---

**Note for users running Vista:** Although Vista comes pre-installed with DirectX 9, the driver uses the d3dx9\_<version\_number>.dll which does not come preinstalled with the operating system. Please make sure that this .dll is installed before trying to use the DirectX 9 driver.

---

**OpenGL** - Select this if you are using OpenGL v1.1 or later on any Windows platform. The driver should run properly with any OpenGL 1.1 compliant graphics hardware.

**GDI** - Select this if you are using Windows GDI on any Windows platform. The driver should run correctly on any GDI compatible device.

### Related Topics

- ❑ [Startup and Display Options](#) (on page 139)
- ❑ [Choose Application Options](#) (on page 139)

## File Location Options

This command displays the **Options** dialog, which shows file management preferences.

### To access these settings:

1. Select **Tools > Options**. The **Options** dialog displays.
2. Click **File Locations** in the left pane.

These options are available:

## Project Management

### Project management folder

Select or browse for the folder to use as a default for saved project files.

### Use project subfolders

Enable to have the program create project folders and subfolders, and organize your files in them.

If you disable **Use project subfolders**, the root folder remains the default file location for projects, imported, and exported data until you change the path.

### Export folder

Specify the folder to use as a default for exported files.

### Download and import folder

Specify the folder to use as an archive for files that are downloaded and imported from field devices or the Internet.

---

**Note:** When the Office Synchronizer application is used to transfer data between the computer and a field device, the Office Sync root folder is used.

---

### Copy imported files to import folder

Enable to have imported files duplicated in the import folder you specify.

This option is automatically enabled when **Use project subfolders** is enabled, but can also be set if you are not using the project subfolders option.

This option can be enabled from the **Import** command pane as well.

---

**Tip:** Disable **Copy Imported files to import folder** when you are simply reviewing data or importing large files that you do not want to save in your **Import** folder.

---

## Templates

### Template folder

Specify the folder to use as the default file location for all projects that are saved as template projects.

Projects stored in this folder appear in the list of templates when you create a new project.

## Data

### Synchronizer root folder

Specify the data synchronization area (also known as the **System root sync folder**). This folder is used by Office Synchronizer to store data transferred between the computer and field devices

## Related Topics

- ❑ [Choose Application Options](#) (on page 139)
- ❑ [Internet Download Options](#)
- ❑ [Results of Default Folder Locations](#) (on page 145)


- ❑ [Startup and Display Options](#) (on page 139)
- ❑ Set Default Folder Locations

### Change the Template Folder

The template folder contains all templates shipped with the office software and any templates you have created.

**Note:** If you change the template folder, the existing templates remain in the original folder location. To move the templates, use Windows® Explorer.

#### To change the template folder:

1. Select **Tools > Options**. The *Options* dialog displays.
2. Select **File Locations**.
3. In the *Template group*, type a path for the location to which you want to save templates, or click the  icon to browse folders.
4. In the *Browse for Folder* dialog, select the folder in which to store templates and then click **OK**.

**Note:** Select **Make New Folder** to create a new template folder.

5. Click **OK**.

When you create a new template, it will automatically be saved to the folder you selected.

### Related topics

- ❑ [File Location Options](#) (on page 142)
- ❑ Set Default Folder Locations

### Set Default File Locations

Projects can contain a variety of files, including imported raw data files, a project file with edited data, and exported data files. One way to manage projects more easily and logically is to specify where each file is saved by default. In the *Options* dialog, **Use project subfolders** is enabled by default. This option creates project folders and subfolders, and organizes your project files in them.

1. When you install this software, a new folder named **Trimble Business Center** is created. The default path for the folder depends on your operating system:
  - In Windows® XP or earlier: **C:\Documents and Settings\*(username)*\My Documents\Trimble Business Center\.**
  - In Windows Vista™: **C:\Users\*(user name)*\Documents\Trimble Business Center\.**



2. When you create a new project, and import or export data, a new folder called *Unnamed* is created in the *Trimble Business Center* folder. If you create a project, but do not import or export data, the folder is not created.
3. If you save the current project, the *Unnamed* folder is renamed to the project name, and the project file is created at the same level.

**Warning:** If you do not save the current project, the *Unnamed* folder and subfolders will be discarded when you close the project.

4. As you create and save additional projects, they and their subfolders are also saved at the same level, so you can easily find and open them.

### To disable Use project sub-folders

The *Use project subfolders* option is a project setting. You can change it from project to project, but once you create a new project, the mode can't be changed for that project. The project will use the same mode, regardless of the options setting. Therefore, if you do not want to use project subfolders, you must disable the option before creating a project.

1. Select **Tools > Options**. The *Options* dialog appears.
2. Click **File Locations** in the left pane.
3. Uncheck **Use project subfolders** in the *Project Management* group box. The *Export folder* box, *Download and import folder* box, and *Copy Imported files to import folder* option become editable.
4. Click **OK**.

**Note:** When you enable *Use project subfolders*, the *Copy Imported files to import folder* option is enabled also, but you can also set it independently if you are not using project subfolders.

**Tip:** You can manually create additional subfolders based on how your data is organized. For instance, if you have multiple field crews collecting data on multiple days, you may want to create subfolders based on those crews or days.

### Related Topics

- [File Location Options](#) (on page 142)
- [Results of Default Folder Locations](#) (on page 145)

### Results of Default Folder Locations

Results of file operations when *Use project subfolders* is enabled:

**Operation****Result****New Project**

A new project file is created, and a new subfolder called **Unnamed** is created when data files are copied. If the project is not saved, this subfolder is deleted.

**Save Project**

If the project is saved for the first time, the **Unnamed** subfolder will be renamed to the name of the project file. If the subfolder is empty, it is deleted.

**Save Project As**

The project file is saved with the new name. If the project has associated data, the project subfolder is copied and saved with the new name.

**Download file**

Downloaded data is saved in the project's import subfolder.

**Export file**

Exported data is saved in the project's export subfolder.

**Import file**

Imported data is saved in the project's import subfolder.

**Related Topics**

- ❑ [File Location Options](#) (on page 142)
- ❑ Set Default Folder Locations

**SCS File Location Options****HEAVY CONSTRUCTION** 


Use this option to specify the location of the report template named SCS900 Reporting Utility.xls or SCS Report Utility.xls, which is installed with the SCS900's Report Utility. This template is used when you view the results of a work order. The default path is **C:\Program Files\Trimble\**.

**To access this setting:**

1. Select **Tools > Options**. The *Options* dialog displays.
2. Click **SCS** and then **File Locations** in the left pane.

## Options

### Work order report template

Type the path to the SCS Report Utility file in the box, or click  to browse for the path.

## Related Topics


- ❑ [Choose Application Options](#) (on page 139)
- ❑ [Results of Default Folder Locations](#) (on page 145)
- ❑ [Startup and Display Options](#) (on page 139)
- ❑ Set Default Folder Locations

## Choose Project Settings

Use the **Project Settings** dialog to set various parameters for your projects, including settings for coordinate systems, units, computations, and views. For a description of any setting, click the name of the setting. The description appears in the information box at the bottom of the dialog. For explanations of terms, see the Glossary.

**Tip:** Save time and effort, and make your projects more consistent, by choosing project settings and then saving the project as a template from which to begin other projects. You can also save baseline processing and network adjustment settings as "styles", which are similar to templates. Share templates and styles with co-workers to ensure consistency across your company.

### To choose project settings:

1. Do one of the following:
  - Select **Project > Project Settings**.
  - Click the  icon on the toolbar.

The **Project Settings** dialog appears.

2. Click a section to view the settings. When you click in a property, a descriptions displays in the info box at the bottom of the dialog.
3. Edit the settings as needed, and click **OK**.

**Note:** Settings with gray text are read-only; settings with black text can be edited.

## Related topics


- ❑ [Choose Application Options](#) (on page 139)
- ❑ [Create a Project Template](#) (on page 163)
- ❑ [Local Site Settings](#) (see "Choose Local Site Settings" on page 159)

## General Information Settings

Use general information settings to:

- Review the project's file properties
- Add a reference number and description of the project
- Add your company's contact information

**Add the names of the field and office staff associated with the project  
To access the settings:**

1. Do one of the following:
  - Select **Project > Project Settings**.
  - Click the  icon on the toolbar.

The *Project Settings* dialog displays.
2. Click **General Information** in the left pane.

### Related topics

- ❑ [Choose Project Settings](#) (on page 147)

## Coordinate System Settings


Use coordinate system settings to:

- Review the current coordinate system and datum transformation

**Note:** To select or create a different coordinate system, click **Change** at the bottom of the dialog.

- Change the geoid model and specify its quality
- Review the local site location and coordinates
- Check the network adjustment transformation parameters
- Review projection, vertical datum, and site calibration details

**See the shift grid name and filename To access the settings:**

1. Do one of the following:
  - Select **Project > Project Settings**.
  - Click the  icon on the toolbar.

The *Project Settings* dialog displays.
2. Click **Coordinate System** in the left pane.

### Related topics

- ❑ [Change the Coordinate System](#) (on page 149)
- ❑ [Choose Project Settings](#) (on page 147)

## Change the Coordinate System



Choose the appropriate coordinate system and geoid model for your project by selecting one that you have used recently. Up to ten of the most recently used systems are stored.

### To select an existing coordinate system:

1. Do one of the following:
  - Select **Project > Change Coordinate System**.
  - Select **Project > Project Settings > Coordinate System**, and click **Change**.

The *Select Coordinate System* dialog displays.

**Note:** The *Select Coordinate System* dialog is part of the *Coordinate System Manager*, a separate application. When only the dialog is open, pressing **[F1]** does not open online help. To view help for the *Coordinate System Manager*, click **Tools > Coordinate System Manager** to open the application. Then press **[F1]** for help.

2. Select **Recently Used System**.
3. Click  or , or press **[PageUp]** or **[PageDown]**, to view the available coordinate systems.
4. When the coordinate system that you want is displayed, click **Finish**. The coordinate system project settings are updated, and your project is recomputed using the new coordinate system.

**Caution:** To avoid problems or unexpected results in your project, do not change the coordinate system after you import data.

### Related topics

- [Coordinate System Manager](#)
- [Coordinate System Settings](#) (on page 148)
- [Define a New Coordinate System](#) (on page 149)

### Define a New Coordinate System

Define parameters for a new coordinate system if the one you need doesn't appear in the list of recently used systems.

### To define a new coordinate system:

1. Do one of the following:
  - Select **Project > Change Coordinate System**.
  - Select **Project > Project Settings > Coordinate System**, and click **Change**.

The *Select Coordinate System* dialog displays.

2. Select **New System**, and click **Next**.
3. To define a default project based on Transverse Mercator parameters:
  - Click **Default Projection**, and click **Next**.
  - Enter the Transverse Mercator parameters that are requested, and click **Next**.
4. To define a projection based on a coordinate system group and zone:
  - Click **Coordinate System and Zone**, and click **Next**.
  - Select the coordinate system group from the list on the left, and select the zone from the list on the right and click **Next**.
5. Select the geoid model you want to use and click **Finish**. Your project is recomputed using the new coordinate system.

### Related topics

- ❑ [Coordinate System Manager](#)
- ❑ [Coordinate System Settings](#) (on page 148)
- ❑ [Change the Coordinate System](#) (on page 149)
- ❑ [Define a Projection](#) (on page 176)

### Restore the Original Coordinate System File

If you have previously installed Trimble® Geomatics Office™ (TGO), you may have an existing `current.csd` file which stores recently-used or custom coordinate systems. During the installation of the software, any existing `current.csd` file is renamed `current.csd.date.xx-xx-xx`.

#### To restore the `Current.csd` file:

1. Open Windows® Explorer and browse to one of these locations, depending on your operating system:
    - In Windows XP or earlier: `C:\Documents and Settings\All Users\Application Data\Trimble\GeoData` unless you have previously installed TGO. If you have installed TGO and then this software, the path is `C:\Program Files\Common Files\Trimble\GeoData`.
- Note:** Due to network security, you may not be able to edit this path unless you have administrative rights.
- In Windows Vista™: `C:\ProgramData\Trimble\GeoData\` or `C:\Program Files\Common Files\Trimble\GeoData\`.
  2. Rename `current.csd` to `current-TBCC.csd`.
  3. Rename the `current.csd.date.xx-xx-xx` to `current.csd`.

## Related topics


- [Change the Coordinate System](#) (on page 149)

## Unit Settings

Use unit settings to review and change project units and unit display formats for:

- Coordinates
- Distances
- Angles and vertical angles
- Azimuths
- Pressure
- Temperature
- GPS time (specifically, not GNSS time)
- Stationing

### Area and volumes To access the settings:

1. Do one of the following:
  - Select **Project > Project Settings**.
  - Click the  icon on the toolbar.

The **Project Settings** dialog displays.

2. Click **Units** in the left pane.

### Entering units

When you edit unit format settings, you are changing how the units display in views, spreadsheets, and commands. You can, however, enter units in any of the valid display formats that you see in the format settings. These are converted to the display format for the unit.

### Converting units

If your project units are set to one type, such as **International foot**, you can still enter other types of units by including a character for the type. For example, you can enter 3m to specify 3 meters. The unit you enter is converted to the project units.

**Caution:** If you change the units of your project, it will be recomputed when you click **OK**. It is recommended that you exit **Project Settings** immediately after changing units, before changing other project settings.

### Related topics

- ❑ [Change Project Units](#) (on page 152)
- ❑ [Choose Project Settings](#) (on page 147)

### Change Project Units

Edit unit settings to control what units are used, and how they display in views, spreadsheets, and commands. Regardless of which unit display format you choose, you can enter units in any of the available formats that you see.

**Note:** If your project units are set to one type, such as *International foot*, you can still enter other types of units by including a character for the type. For example, you can enter 3m to specify 3 meters. The unit you enter is converted to the project units.

**Note:** You can also enter feet and inches by typing the two numbers with a space in between. For example, type 4\_8 for 4 feet, 8 inches.

### To change the project units:

1. Select **Project > Project Settings**. The *Project Settings* dialog displays.
2. Click **Units**, and then click the type of units you want to change in the left pane.

**Note:** To change project units from feet to meters, click **Distance** and then select **Meters** in the *Display* box.

3. Select the box for which you want to change the units or format.
4. Make the desired changes.
5. To save and apply the changes, click **OK**.

**Caution:** If you change the units of your project, it will be recomputed when you click **OK**. It is recommended that you exit *Project Settings* immediately after changing units, before changing other project settings.

### Related topics


- ❑ [COGO Expressions, Units, and Entry Formats](#) (on page 87)
- ❑ [Unit Settings](#) (on page 151)

### View Settings

Settings are available for the various types of graphic, spreadsheet, and other views in which your project data is displayed. Use view settings to:



- Specify whether to include 2D data in the 3D view
- Choose the input method for horizontal and vertical alignment segments
- Control graphic view display characteristics, such as plot scales and vertical exaggeration


Control annotation text, and gridline intervals, colors and line styles in views **Note:** You can also click the  icon on the toolbar to turn gridlines on and off.

- Show and hide individual columns for data on spreadsheet views
- Set the default view filter used when you open new graphic views

**Note:** If you delete a view filter that you have set as the default, the **Default View Filter** will revert to the **All** view filter, which cannot be deleted.

**Tip:** If your project has a lot of data, you may want to select a default view filter other than **All** so that graphic views refresh more quickly.

#### To access the settings:

1. Do one of the following:
  - Select **Project > Project Settings**.
  - Click the  icon on the toolbar.

The **Project Settings** dialog displays.
2. Click **View** in the left pane.
3. Click any of the view types to display and edit individual settings.

#### Related topics

- [Choose Project Settings](#) (on page 147)

#### Change the Gridline Display

Display gridlines in the plan view to understand a project's scale and location.

#### To display or hide gridlines:

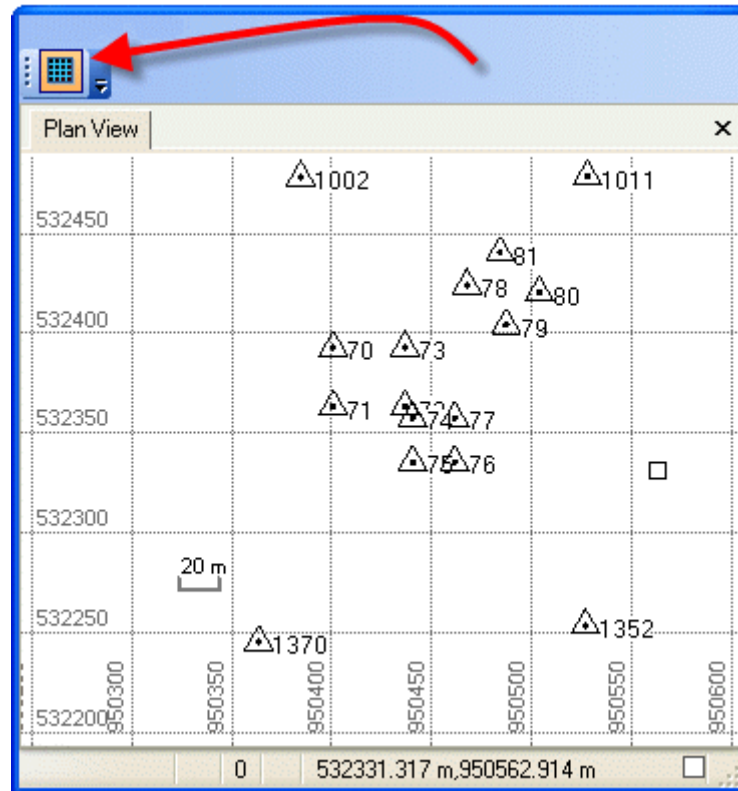
- Click the  icon on the toolbar.

#### To change gridline properties:

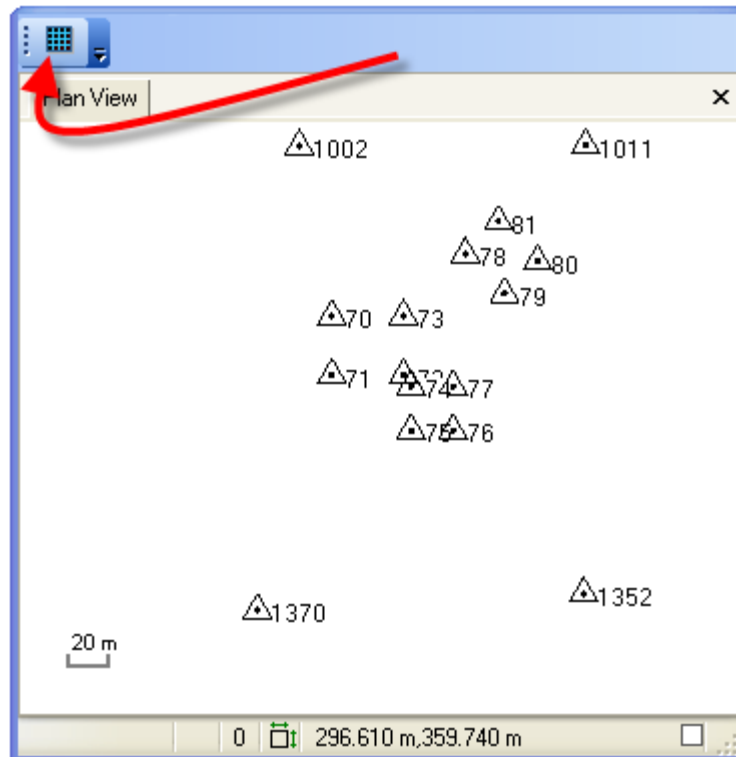
1. Select **Project > Project Settings**. The **Project Setting** dialog displays.
2. Click the **View** folder.
3. Expand **Plan View**, and click **Grid Line Definition**.
4. Click in the boxes and edit the properties as needed.

5. Click **OK**.

Example with grid lines **On**.



Example with gridlines **Off** (same data, same scale).




### Related topics

- [View Settings](#) (on page 152)


### Toggle Line Marking

Display or hide markers and labels for horizontal and vertical values along linestrings in 2D views to make viewing, understanding, and editing them easier. Markers are symbols that distinguish between horizontal segment end points, vertical control points and the overall line's end points. Labels are annotations that indicate the elevation of vertical control points.

#### To show or hide line markings:

- Click the  icon on the toolbar.
- Select **View > Toggle Line Marking**.

#### To set which line markings are shown:

1. Do one of the following:
  - Select **Project > Project Settings**.
  - Click the  icon on the toolbar.

The *Project Settings* dialog displays.

2. Click **View** in the left pane, and click **Display Options**.
3. Edit individual settings in the **Marking** section.

### Related topics

- ❑ [Change the Gridline Display](#) (on page 153)
- ❑ [View Settings](#) (on page 152)

## Computational Settings


Use computational settings to:

- Review and set horizontal and vertical tolerances for computed survey data, including points of varying quality, and meaned GPS vectors.

**Note:** If computation results on data fall outside these tolerances, the data is flagged in the **Project Explorer** and graphic views, and a message appears in the **Flags** pane.

- Select a level of confidence for displayed error values.
- Specify the horizontal and vertical tolerances to enforce when creating surface breaklines from 3D alignments.
- Set the maximum edge length and angle defaults for creating surface triangles.

### To access the settings:

1. Do one of the following:
  - Select **Project > Project Settings**.
  - Click the  icon on the toolbar.

The **Project Settings** dialog displays.

2. Click **Computational Settings** in the left pane.

### Related topics

- ❑ [Choose Project Settings](#) (on page 147)
- ❑ [Project Computations](#) (on page 157)

## Project Computations

The **Compute Project** command enables you to compute your project after you make changes to the data. The computation applies the changes made to all affected observations and determines the coordinates for points in the chronological sequence a surveyor would typically expect. If conflicting data is available for the computation of a coordinate, the software gives preference to higher quality data over lower quality data.

**Note:** When you import data into a project, change project settings, or change the coordinate system, the project is automatically computed for you.

### Related topics

- Compute Project Command

### Run a Project Computation Report

Generate a computation report to see a summary of the errors and warnings that occurred during the last computation of your project data.

#### To run a Project Computation Report:

- Select **Reports > Project Computation Report**. The *Project Computation Report* displays in your default Web browser.

#### To customize the report:

1. Select **Reports > Report Options**. The *Report Options* command pane displays.
2. Select **Project Computation Report** in the list.
3. Expand sections and specify output settings in the **Settings** group as needed.
4. Click **Apply** if you want to customize additional reports, or **OK** to close the command pane.


### Related topics

- [Project Computations](#) (on page 157)
- [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Feature Code Processing Settings

Use feature processing settings to configure how feature codes are processed.

#### To specify feature processing settings:

1. Do one of the following:
  - Select **Project > Project Settings**.
  - Click the  icon on the toolbar.

The *Project Settings* dialog displays.

2. Select **Feature Code Processing** in the left pane.
3. Click each section and set the options as shown in the following table.
4. Click **OK**.

### Options

#### General

**Process feature codes on import** **Prompt** - Select this option if you want to be prompted on whether or not to process feature codes during data import. If you select to process feature codes during import and a feature definition (.fxl) file has not been specified in the project settings, you will be prompted to specify it at the time of import.

**Yes** - Select this option if you want feature codes to be automatically processed during data import without prompting you. If a feature definition (.fxl) file has not been specified in the project settings, you will be prompted to specify it at the time of import.

**No** - Select this option if you never want feature codes to be processed during data import.

**Decimal precision** Specify the number of decimals to display with a numeric attribute (real number) when no feature definition (.fxl) file is specified.

#### Processing

**Feature definition file** Specify the feature definition (.fxl) file you want to use to process feature codes in the project. This is required if you want to process feature codes.

### Related topics

- [Choose Project Settings](#) (on page 147)
- [Understanding Feature Data](#) (on page 359)
- [Workflow for Feature Data](#) (on page 361)
- [Enter, Edit, and Delete Feature Code Strings](#) (on page 363)
- [Process Feature Codes](#) (on page 367)

### Abbreviation Settings

Use abbreviation settings to control notations for horizontal and vertical alignment classifications in the *Alignment Geometry Report*.

#### To access the settings:

1. Do one of the following:
  - Select **Project > Project Settings**.

- Click the  icon on the toolbar.

The **Project Settings** dialog displays.

2. Click **Abbreviations** in the left pane.

### Related topics

- [Choose Project Settings](#) (on page 147)

## Choose Local Site Settings

Enter local site settings to create a ground coordinate system to accommodate elevation differences between your site and the ellipsoid. The scale factor can be calculated for you.

You should define local site settings for the project location at the start of the project because this information is used in calculations. The application uses:

- The project latitude to calculate the earth's radius
- The project latitude and longitude to define the grid scale factor

### **The project height as the default elevation and to calculate the elevation factor To specify local site settings:**

1. Select **Project > Local Site Settings**. The **Local Site Settings** command pane displays.
2. Select a coordinate system to display in the **Coordinate type** list. Any changes take effect immediately.

If you select **Grid**:

1. Click in the **Northing** box.
2. Pick a point in a graphic view, right-click for options, or type coordinates in the **Northing** and **Easting** boxes to specify the location for the local site.
3. Pick a point in a graphic view, right-click for options, or type a value in the **Elevation** box.

If you select **Local** or **Global**:

1. Click in the **Latitude** box.
2. Pick a point in a graphic view, right-click for options, or type coordinates in the **Latitude** and **Longitude** boxes to specify the location for the local site.
3. Pick a point in a graphic view, right-click for options, or type a value in the **Height** box.

3. Type a value in the *Ground scale factor* box, or check **Compute ground scale factor from project location** to have the value computed.
4. To close the dialog, click **OK**.

**To view the local site settings:**

1. Select **Project > Project Settings**. The *Project Setting* dialog displays.
2. Select **Coordinate System** and then **Local Site**. The read-only *Local Site* settings display.

**Related topics**

- ❑ [Coordinate System Settings](#) (on page 148)
- ❑ [Calibrate a Site](#) (on page 184)

**Local Site Setting Options**

Use these options to define the current project's local site. They are available in the *Local Site Settings* command pane.



## Options

### Project location

#### Coordinate type

**Grid** - Select this to enter planar northing, easting, elevation, and height values in a grid system.

**Local** - Select this to enter ellipsoidal latitude, longitude, height, and elevation values on a local datum.

**Global** - Select this to enter global ellipsoidal longitude, height, and elevation values on a global datum.

### Ground coordinates

#### Ground scale factor

Type a factor by which to convert grid distances to ground distances.

#### Compute ground scale factor from project location

Check this to have the ground scale factor calculated for you. The value will be a product of the grid factor (determined from the horizontal project location) and the height scale factor (determined from the vertical project location).

---

**Note:** When *Compute ground scale factor* is checked, the *Ground scale factor* box is unavailable.

---

### Coordinate Display

#### False Northing offset

Type a value for the distance to offset from the northing and easting values.

#### False Easting offset/

For example, if you enter:

- False northing offset:  
-6,540,000
- Then a northing of 6,542,111 becomes:  
2,111


### Related topics

- ❑ [Calibrate a Site](#) (on page 184)
- ❑ [Choose Local Site Settings](#) (on page 159)

## Create a New Project

You can create a new project using the default template, or you can select a new template from which to create a project.

**To create a new project using the default template:**

- Click the  icon. This is the quickest way to create a new project.

**To create a new project by choosing a template:**

1. Do one of the following:
  - Select **File > New Project**.
  - Press **[Ctrl] + N**.
2. In the *New Project* dialog, select a template.
3. Click **OK**.

**To create a new project upon starting the application:**

1. Select **Tools > Options**. The *Options* dialog displays.
2. Select **General > Startup and Display**.
3. Select **Open new default project** in the *Starting state* list.
4. Click **OK**. Each time you start this software, a new project will be started using the default template.

**Note:** Creating a new project closes any project that is currently open.

**Related topics**

- ❑ [Choose Application Options](#) (on page 139)
- ❑ [Choose Project Settings](#) (on page 147)
- ❑ [Change the Default Template](#) (on page 164)
- ❑ [Startup and Display Options](#) (on page 139)

## Use a Project Template

Several default templates are provided so you can start projects with consistent distance units. It is even more efficient, however, to create your own templates. In your own templates, you can save additional project settings, including:

- Company, user, and file information, such as field and office operators, contact numbers, and addresses
- Coordinate system information, such as a datum transformation and geoid model
- Units settings, such as coordinate formatting
- View settings, such as a plot scale and grid line definition
- View filters and selection sets
- Computational settings, such as horizontal and vertical tolerances

When you start projects using your own templates, any settings you specified are included, saving you time and effort, and making your projects more consistent. You can even share templates with co-workers to ensure consistency across your company.

### Related topics

- ❑ [Create a Project Template](#) (on page 163)


## Create a Project Template

When you save a project as a template, all project settings and data are saved in the new template file, and the new name is added to the template list.

**Warning:** Saving data that requires computation in project templates is **not** recommended; the data may not recompute properly.

**Note:** It is a good idea to create a template for each set of coordinate system/project units you commonly use.

### To create a project template using an existing project:

1. Open, or create, a project that you want to save as a template.
2. Review and modify project settings as needed.
3. Create layers and view filters that you want to be available in future projects.
4. Remove any unnecessary data from the project.
5. Select **File > Save Project as Template**. The *Save Project as Template* dialog displays.
6. In the **Name** field, type a name for the template.
7. If you want this template to be used when you create a project by clicking the  icon, check the **Save project as default template** box.
8. Click **OK**. The template is saved in the template folder. When select **File > New Project**, the template appears among the other templates.

**To delete a project template:**

1. Select **File > Save Project as Template**. The *Save Project as Template dialog* displays.
2. Click in the left column for template you want to delete.
3. Click **Delete**. A confirmation message displays.


**Note:** You cannot delete the template that is set as the default. If you want to delete the default template, set another as the default.

4. Click **Yes**.
5. Click **OK**.

**Related topics**

- [Change the Default Template](#) (on page 164)
- [Change the Template Folder](#) (on page 144)
- [Define a New Coordinate System](#) (on page 149)
- [Create a View Filter](#) (on page 73)
- [Save a Project](#) (on page 165)
- Save Project As Command

**Change the Default Template**

When you create a new project by clicking the  icon on the toolbar, the default template is used.

**To change the default template:**

1. Select **File > Save Project as Template**.
2. Select the template you want to use as a default.
3. Click **Set as default**.
4. Click **OK**.


**Related topics**

- [Create a Project Template](#) (on page 163)
- [Change Template Folder](#) (see "Change the Template Folder" on page 144)

**Open an Existing Project**

Use these commands to quickly open a recent project, or browse to another previous project.

1. Do one of the following:
  - Select **File > Open Project**.

- Click the  icon on the toolbar.
- Press **[Ctrl] + O**.

The **Open File** dialog displays, showing a list of available projects in the folder last opened.

2. Select a project from the list, or browse to locate a project in a different folder.
3. Click **Open**.

#### To open a recent project:

- Select **File**, and then select a project from the list of recent projects (at the bottom of the **File** menu). The project opens.

**Note:** Opening a project closes any project that is currently open.


#### Related topics

- ❑ [Create a New Project](#) (on page 161)
- ❑ [Startup and Display Options](#) (on page 139)

## Save a Project

You can save a project, rename a project/save a project to a different path or filename.

#### To save a project:

1. Do one of the following:
  - Select **File > Save**.
  - Press **[Ctrl] + S**.
  - Click the  icon on the toolbar.
2. Select the folder where you want to save the file, and assign a file name.
3. Click **Save**.

#### To rename an existing project:

1. Select **File > Save As**. The **Save As** dialog displays.
2. Select the folder in which you want to save the file in the **Save in** box.
3. Type a name in the **File name** box.
4. Click **Save**.

**Note:** You cannot overwrite an existing project by naming a project the same name as an existing project.

### Related topics

- ❑ [Archive a Project](#) (on page 166)
- ❑ [Create a Project Template](#) (on page 163)
- ❑ Close All Windows Command
- ❑ Close Project Command

## Archive a Project

Use this command to save a project (.vce) file and its associated subfolder in a compressed (.zip) file of the same name. This enables you to quickly compile all parts of a project into a smaller file that is suitable for sending to a colleague or archiving.

**Note:** File archiving only works if you have *Use project subfolders* checked in the *File Locations* section of the *Options* dialog.

### To archive a project:

1. Make sure the project you want to archive is closed.
2. Select **File > Archive Project**. The *Archive a Project* dialog displays.
3. Select the file you want to archive in the *Save in* box.
4. Click **Save**. The .zip file appears, next to the project file and subfolder.

### Related topics

- ❑ [Save a Project](#) (on page 165)
- ❑ Set Default Folder Locations

## Print a View or Report


You can print a graphic view to any Windows-supported printing device. You can also print a report using your default Web browser's print command.

### To select a printer:


1. Select **File > Page Setup**.
2. In the *Page Setup* dialog, set the paper size, orientation, and margins.
3. Click **Printer**. The second *Page Setup* dialog appears.
4. In the *Name* list, select the printer you require.
5. Click **OK** in both dialogs.

**Note:** You can also select a printer within the *Print* command.

**To preview a print job:**


1. Click the view you want to print to make it active.
2. Do one of the following:
  - Select **File > Print Preview**.
  - Click the  icon.

The *Print preview* dialog opens.

3. In the *Print preview* dialog, you can:
  - View the plan view of your project
  - Zoom to view less or more detail
  - Select the page layout
  - Click the  icon to print, or **Close**.

**Note:** You cannot cancel a print job if you use this option.

**To print:**

1. Do one of the following:
  - Select **File > Print**
  - Press **Ctrl + P**
  - Click the  icon.
2. Select the printer, page range and number of copies to print.
3. Click **OK**.

**Related topics**

- Page Setup Command
- Print Preview Command

## Troubleshoot a Project Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
You cannot reopen a .vce project file.	<p>The project file is locked, possibly due to a crash or an improper program shutdown.</p> <p>Improper shutdown includes when:</p> <ul style="list-style-type: none"> <li>▪ A project is open and the power to the computer is interrupted.</li> <li>▪ A project is open and the process is ended from Windows® Task Manager.</li> </ul>	<p>Delete the lock (project name.lk) from the file project folder.</p> <p>The project will lose all changes made since the last save.</p>

## Troubleshoot a Program Freeze

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
Program freezes when selecting many objects.	<p>The <b>Properties</b> pane is open.</p> <p>When you select many objects with the <b>Properties</b> pane open, it looks for the properties common to all of the objects, slowing the program down and making it look frozen.</p>	<p>Close the program. If you have trouble reopening the project, check the directory where the .vce file is stored. If there is a lock (*.lk) file with the same name as the project, delete it and reopen the project. Close the <b>Properties</b> pane before reselecting the objects.</p>
Program doesn't respond in the expected way; nothing seems to work.	<p>The mouse may be set to a mode other than <b>Select</b>.</p>	<p>Check your mouse mode on the <b>Mouse</b> toolbar. If needed, reset it to <b>Select</b>.</p>



The program freezes.	Toolbars are corrupted.	<p>Consider contacting Technical Support.</p> <p>Otherwise, remove the application data folder located at <b>C:\Documents and Settings\<user name="">\Application Data\Trimble\Trimble Business Center\<version&gt;< b=""></version&gt;<></user></b></p> <hr/> <p><b>Note:</b> If you do not see the <b>Application Data</b> folder at the path listed above, it may be hidden. To show hidden folders, in Windows® Explorer, select <b>Tools &gt; Folder Options</b>. Click the <b>View</b> tab and select <b>Show hidden files and folders</b> in <b>Advanced Settings</b>. Then click <b>OK</b>.</p> <hr/>
The program appears to freeze when you float a pane or try to open a dialog.	If you are running the program on a secondary monitor, and you float a pane or use a command that launches a dialog, the pane or dialog might appear out of either monitor's visible range. It will be located off of the primary monitor, in the space opposite the secondary monitor, causing Trimble Business Center to appear 'frozen'.	To reach the dialog or pane, right-click the application's name on the Windows Taskbar and select <b>Move</b> . Then, press the appropriate arrow key to move the dialog into your primary's monitor's visible range.
Program appears to freeze when you are trying to e-mail SCS files.	If you are running the program on a secondary monitor, and you attempt to e-mail SCS files using the <b>Compress/E-mail SCS Files</b> command, your e-mail program may open a dialog confirming the operation out of either monitor's visible range. The dialog will be located off of the primary monitor, in the space opposite the secondary monitor, causing the program to appear 'frozen'.	To reach the dialog and confirm the e-mail operation, right-click the e-mail application's name on the Windows Taskbar and select <b>Move</b> . Then, press the appropriate arrow key to move the dialog into your primary's monitor's visible range.

## CHAPTER 5

# Set Up Geodetic Reference Data

## Understanding Geodetic Reference Data

This software uses a coordinate system to transform measurements on a curved surface (the earth) to a flat surface (a map or plan). For example, a coordinate system is used to calculate grid coordinates for a point measured using GPS (GPS measurements are made on the WGS-84 ellipsoid).

A coordinate system can consist of the following elements:

- A datum transformation (between the WGS-84 ellipsoid and the local ellipsoid)
- A projection
- A geoid model
- A GPS site calibration (consisting of an horizontal adjustment and a vertical adjustment)

You must select a coordinate system for every project. If you do not have a system, or do not know which system to select, use the default projection.

**Note:** Make sure that the points in a project are within a reasonable distance from the projection origin according to the properties of the projection used.

### Coordinate system database

The coordinate system database is stored as a file called *Current.csd*. It contains all of the coordinate system information. The office software supplies an extensive set of published coordinate systems from around the world. To define or edit coordinate systems, zones, sites, datum transformations, ellipsoids, and geoid models, use the [Coordinate System Manager](#) (on page 173).

### Related topics

- ❑ [Change the Coordinate System](#) (on page 149)
- ❑ [Coordinate System Manager](#)
- ❑ [Define a New Coordinate System](#) (on page 149)
- ❑ [Restore the Original Coordinate System File](#) (on page 150)

## Define the Coordinate System

### Change the Coordinate System


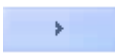
Choose the appropriate coordinate system and geoid model for your project by selecting one that you have used recently. Up to ten of the most recently used systems are stored.

#### To select an existing coordinate system:

1. Do one of the following:
  - Select **Project > Change Coordinate System**.
  - Select **Project > Project Settings > Coordinate System**, and click **Change**.

The *Select Coordinate System* dialog displays.

**Note:** The *Select Coordinate System* dialog is part of the *Coordinate System Manager*, a separate application. When only the dialog is open, pressing **[F1]** does not open online help. To view help for the *Coordinate System Manager*, click **Tools > Coordinate System Manager** to open the application. Then press **[F1]** for help.

2. Select **Recently Used System**.
3. Click  or , or press **[PageUp]** or **[PageDown]**, to view the available coordinate systems.
4. When the coordinate system that you want is displayed, click **Finish**. The coordinate system project settings are updated, and your project is recomputed using the new coordinate system.

**Caution:** To avoid problems or unexpected results in your project, do not change the coordinate system after you import data.

#### Related topics

- ❑ [Coordinate System Manager](#)
- ❑ [Coordinate System Settings](#) (on page 148)
- ❑ [Define a New Coordinate System](#) (on page 149)

### Define a New Coordinate System

Define parameters for a new coordinate system if the one you need doesn't appear in the list of recently used systems.

#### To define a new coordinate system:

1. Do one of the following:

- Select **Project > Change Coordinate System**.
- Select **Project > Project Settings > Coordinate System**, and click **Change**.

The **Select Coordinate System** dialog displays.

2. Select **New System**, and click **Next**.
3. To define a default project based on Transverse Mercator parameters:
  - Click **Default Projection**, and click **Next**.
  - Enter the Transverse Mercator parameters that are requested, and click **Next**.
4. To define a projection based on a coordinate system group and zone:
  - Click **Coordinate System and Zone**, and click **Next**.
  - Select the coordinate system group from the list on the left, and select the zone from the list on the right and click **Next**.
5. Select the geoid model you want to use and click **Finish**. Your project is recomputed using the new coordinate system.

### Related topics

- ❑ [Coordinate System Manager](#)
- ❑ [Coordinate System Settings](#) (on page 148)
- ❑ [Change the Coordinate System](#) (on page 149)
- ❑ [Define a Projection](#) (on page 176)

## Restore the Original Coordinate System File

If you have previously installed Trimble® Geomatics Office™ (TGO), you may have an existing `current.csd` file which stores recently-used or custom coordinate systems. During the installation of the software, any existing `current.csd` file is renamed `current.csd.date.xx-xx-xx`.

### To restore the `Current.csd` file:

1. Open Windows® Explorer and browse to one of these locations, depending on your operating system:
  - In Windows XP or earlier: `C:\Documents and Settings\All Users\Application Data\Trimble\GeoData` unless you have previously installed TGO. If you have installed TGO and then this software, the path is `C:\Program Files\Common Files\Trimble\GeoData`.

**Note:** Due to network security, you may not be able to edit this path unless you have administrative rights.

- In Windows Vista™: `C:\ProgramData\Trimble\GeoData\` or `C:\Program Files\Common Files\Trimble\GeoData\`.

2. Rename `current.csd` to `current-TBCC.csd`.
3. Rename the `current.csd.date.xx-xx-xx` to `current.csd`.

### Related topics

- [Change the Coordinate System](#) (on page 149)

## Troubleshoot a Coordinate System Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
You cannot create or edit coordinate systems, or save sites.	You are running as a Limited User (non-administrator). Limited users do not have “write” permissions for the <i>current.csd</i> file, which means that you cannot create or edit coordinate systems, or save sites.	<p>You must be granted “write” permissions for the <i>current.csd</i> file by an administrator. The location of that file depends on your operating system:</p> <ul style="list-style-type: none"> <li>▪ <b>In Windows® XP or earlier:</b> C:\Documents and Settings\All Users\Application Data\Trimble\GeoData or C:\Program Files\Common Files\Trimble\GeoData.</li> <li>▪ <b>In Windows Vista™:</b> C:\ProgramData\Trimble\GeoData or C:\Program Files\Common Files\Trimble\GeoData.</li> </ul> <hr/> <p><b>Note:</b> This may be a non-issue if other Trimble software has been previously installed, and access rights have been resolved.</p> <p><b>Note:</b> If you do not see the <i>Application Data</i> folder at the path listed above, it may be hidden. To show hidden folders, in Windows® Explorer, select <b>Tools &gt; Folder Options</b>. Click the <b>View</b> tab and select <b>Show hidden files and folders</b> in <i>Advanced Settings</i>. Then click <b>OK</b>.</p>

## Coordinate System Manager

The *Coordinate System Manager* is a standalone utility that gives you access to your coordinate system database (*Current.csd*). Use the manager to create coordinate systems, or to determine which coordinate systems, geoid models, and sites are available for use in your project.

**To open the Coordinate System Manager:**

- Select **Tools > Coordinate System Manager**.

**Note:** The *Coordinate System Manager* has its own help system. Open the utility and select **Help > Help Topics**, or press **[F1]** within the software.

**Related topics**

- ❑ [Change the Coordinate System](#) (on page 149)
- ❑ [Coordinate Systems](#) (see "Understanding Geodetic Reference Data" on page 170)
- ❑ [Define a New Coordinate System](#) (on page 149)
- ❑ [Restore the Original Coordinate System File](#) (on page 150)

**Scale-Only Projection**

The software allows you to import survey data using a scale-only projection. Because the software cannot calculate global or local coordinates from grid positions in a scale-only projection, certain functionality is affected. Note the following when working with scale-only projection data:

- You can specify a scale factor for the scale-only projection when you import the data, and after import in the *Coordinate System* section of the *Project Settings* dialog.
- The *Create Point* and *Add Coordinate* commands allow to enter a grid point or coordinate in scale only.
- The *Local Site Settings* and *Site Calibration* commands are disabled.
- When using the *Inverse* command, only grid and ground distances are available.
- The *Point List* report includes only grid data.

When working with coordinate system data and scale-only projection data in the same project, note the following:

- When you import scale-only projection data into a project that already contains coordinate system data, the scale-only projection data is converted to the existing coordinate system.
- When you import coordinate system data into a project that contains scale-only projection data, the scale-only projection data is converted to the existing coordinate system.



**Use a Datum****Create a Datum Grid File**

If you have chosen a coordinate system for your project that uses a datum, and the datum hasn't been defined yet, you must create it before uploading to a field device.

To do this, find a coordinate system with the appropriate datum to use. If you know of a coordinate system that uses the datum you need, you can select it to combine its .dgf files into a single .cdg file suitable for uploading.

**Note:** When you download or import data into a project, the datum used by the imported data must correspond with the datum defined for the project's coordinate system.

#### To create a datum grid:

1. Select **Project > Project Settings**. The *Project Settings* dialog displays.
2. Click **Coordinate System** and then **Datum Transformation** in the left pane.
3. Review the parameters. You can only create a datum grid file if the **Latitude Grid File** and **Longitude Grid File** boxes are shown, and .dgf files are listed in them. If there are, click **OK**, and proceed to step 9.
4. If there aren't .dgf files shown, click **Change** at the bottom of the dialog. The *Select Coordinate System* dialog displays.
5. Select **Recently Used System**.
6. Click  or , or press **[PgUp]** or **[PgDn]**, to view the available coordinate systems.
7. When you find a coordinate system that uses the datum you need, click **Finish**. The *Project Settings* dialog updates.
8. Confirm that .dgf files are shown in the **Latitude Grid File** and **Longitude Grid File** boxes. These files will be combined into a datum grid file (.cdg) file that you can save and upload to a field device.
9. Click **OK** to close the *Project Settings* dialog.
10. Do one of the following:
  - Select **Project > Datum Gridding**.
  - In the *Device* pane, with a device connected, click **Tasks** and select **Upload datum grid (.dgf) file**.

The *Datum Gridding* command pane displays. The .dgf files appear in the *Project Datum Grid* group.
11. Click **Create** at the bottom of the command pane. The *Save As* dialog displays.
12. Accept the default file name, and click **Save**. The datum grid file appears in the *Datum Grid Files (.cdg)* list. Now it can be uploaded to a field device.

#### Related topics

- [Upload a Datum Grid](#) (see "Upload a Datum Grid File" on page 227)

## Datum Grid Options

Use these options to select a datum file to upload to a field device. They are available in the *Datum Gridding* command pane.

**Note:** Click the lower group header to switch between the current project's datum and a stored datum grid file.

### Options

<b>Folder</b>	Select, or navigate to, the folder in which the datum grid files (.cdg) are installed.
<b>Datum grid files (.cdg)</b>	This displays the names of the available datum grid files.
<b>Description</b>	This shows the regions covered by the datums.
<b>Size</b>	This displays the file sizes. Files larger than 1 MB may take a while to upload.
<b>Project/Selected datum grid</b>	<p>If you <b>have</b> a datum grid file selected in the list, this displays details on it.</p> <p>If you <b>do not have</b> a datum grid file selected, this displays details on the datum used in your current project's coordinate system.</p>
<b>Upload</b>	Click this to transfer the datum grid file to the connected field device.
<b>Create</b>	Click this to save a new datum grid file.
<b>Use project datum</b> (via the context menu)	<p>Check this before uploading if you want to upload a temporary copy of the current project's datum, instead of selecting one of the permanently stored datum grid files in the list.</p> <p><b>Upload to Device</b> is also available from the context menu.</p>

### Related topics

- ❑ [Create a Datum Grid File](#) (on page 174)
- ❑ [Upload a Datum Grid](#) (see "Upload a Datum Grid File" on page 227)

## Define a Projection

Use a false origin to define a projection when you import raw GNSS data for which you did not previously specify the projection.

### To define a projection:

1. Import and check-in your raw GNSS data. If there is not associated projection, the *Projection Definition* dialog displays.
2. If needed, type grid coordinates in the **Northing** and **Easting** boxes to base the origin on the best known grid coordinates.



3. Click **OK**. The new coordinates become the projection's origin.

### Related topics

- ❑ Check-In Raw GNSS Data
- ❑ [Local Site Setting Options](#) (on page 160)

## Use a Geoid

### Geoid Options

Use these options to select a geoid file to upload to a field device. They are available in the *Geoid Sub-Gridding* command pane.

#### Options



Click this to display the **Select Coordinate System** dialog, where you can choose a recently-used coordinate system, or define a new one to use in your project.

#### Folder

Select, or navigate to, the folder in which the geoid grid files (.ggf) are installed.

#### File (.ggf files)

This displays the names of the available geoid grid files. If any geoid sub-grid files have been defined, they display beneath the name of the complete geoid grid file.

#### Description

This shows the regions covered by the geoids.

#### Size

This displays the file sizes. Files larger than 1 MB may take a while to upload.

#### Upload

Click this to transfer the geoid grid file to the connected field device.

---

**Note:** This is only available if you have a field device connected and accessed the command through the **Tasks** button.

---

#### Create sub-grid

Click this to define a sub-area of the geoid file to save upload time and field device memory.

### Related topics

- ❑ [Define a Geoid Subgrid](#) (see "Define a Geoid Sub-Grid" on page 177)
- ❑ [Upload a Geoid File to a Field Device](#) (see "Upload a Geoid File" on page 228)

### Define a Geoid Sub-Grid


If the geoid file you want to use for data collection is too large, define a subsection of the area to use before uploading to a field device. This saves upload time and field device memory.

**Note:** When you download or import data into a project, the geoid sub-grid used by the imported data must correspond with the geoid defined for the project's coordinate system.



### To create a geoid sub-grid:

1. Do one of the following:
  - Select **Project > Geoid Sub-Gridding**.
  - In the **Device** pane, with a device connected, click **Tasks** and select **Upload geoid (.ggf) file**.

The **Geoid Sub-Gridding** command pane displays.

2. If needed, select the folder containing the installed geoid files in the **Folder (.ggf)** list, or click the  icon and navigate to the folder.

**Note:** The default location for .ggf files depends on your operating system:  
 In Windows® XP or earlier: **C:\Documents and Settings\All Users\Application Data\Trimble\GeoData** unless you have previously installed Trimble® Geomatics Office™ (TGO). If you have installed TGO and then this software, the path is **C:\Program Files\Common Files\Trimble\GeoData**.  
 In Windows Vista™: **C:\ProgramData\Trimble\GeoData\** or **C:\Program Files\Common Files\Trimble\GeoData\**.

3. Select a geoid file in the **Geoid File (.ggf)** list.
4. Right-click and select **Create sub-grid** from the context menu. The **Create Geoid Grid File** dialog displays. The whole globe is visible, and the geoid is centered. In the **Geoid File Properties** group, the size of the whole geoid appears in the **Size** box.
5. In the **Suffix to Append** box, type text to attach to the end of the original geoid file name to define the sub-grid. The complete file name appears as you type the suffix. For example, if you will be collecting data in Baja, Mexico, add Baja as the suffix to the geoid Mexico97.ggf to create the file Mexico97Baja.ggf.
6. If needed, uncheck borders or rivers in the **Globe Properties** group to simplify the display.
7. Click the  icon to zoom in to the extents of the geoid. For explanations of the other viewing tools, see **Geoid Sub-grid Options**.
8. Click the  icon, and window around the portion of the geoid that you need to create a sub-geoid. The size of the geoid sub-grid appears in the **Size** box.

**Note:** You cannot adjust the sub-grid window that you draw, but you can redraw it multiple times.

9. Click **Save**. The **Save Geoid Grid File As** dialog displays.

10. Click **Save** again. The geoid sub-grid file appears in the geoid list just below the original geoid file.

### Related topics

- ❑ [Geoid Subgrid Options](#) (see "Geoid Sub-Grid Options" on page 179)
- ❑ [Upload a Geoid File to a Field Device](#) (see "Upload a Geoid File" on page 228)

### Geoid Sub-Grid Options

Before uploading to a field device, use these options to define a sub-area of a larger geoid file to save upload time and field device memory. They are available in the **Create Geoid Grid File** dialog.

## Options

### Geoid file properties

#### Geoid

This displays the name of the geoid on which the sub-grid will be based.

#### File name

This displays the geoid's file name.

#### Suffix to append

Type text to attach to the end of the original geoid file name to identify the sub-grid.

#### Size (KB)

Shows the original geoid size until you draw the sub-grid. Then it shows the geoid sub-grid's size.

### Sub-grid properties

#### First latitude

These display the coordinates of the first corner of the window you pick when drawing a sub-grid.

#### First longitude

#### Second latitude

These display the coordinates of the opposite corner of the window you pick when drawing a sub-grid.

#### Second longitude

### Globe properties

#### Cursor latitude

These display the coordinates of the current cursor location. If the cursor is not over the globe, they display the point at which the cursor left the globe.

#### Cursor longitude

#### US state borders

Uncheck this to hide state boundaries in the United States.

#### National borders

Uncheck this to hide boundaries of nations.

#### Major rivers

Uncheck this to hide major waterways.



Click this to activate the selection tool. Then click and drag to draw a window within the current geoid to define the opposite corners of a geoid sub-grid.

Any portion of the sub-grid that you draw outside of the geoid boundary is ignored.

---

**Note:** To erase a sub-grid, click anywhere off of the globe.

**Tip:** After you window to create a sub-grid, you can click within the sub-grid to zoom in.

---



Click this to activate the pan tool. Then click and drag on the globe to move to different locations.



Click this to zoom in to the extents of the geoid. Then click it again to zoom in by an increment (x2), or pick a point on the globe to center it and zoom in on that point.



Click this to activate the zoom out tool. Then click it again to zoom out by an increment (x2), or pick a point on the globe to center it and zoom out from that point.



Click this to zoom to the extents of the globe, and center the current geoid.

### Related topics

- [Define a Geoid Subgrid](#) (see "Define a Geoid Sub-Grid" on page 177)

## Calibrate a Site

### Understanding Site Calibration

The site calibration process establishes the relationship between WGS-84 data collected by GNSS receivers and local control positions (expressed as a local map grid with elevations above sea level). This relationship is defined by a series of mathematical transformations. Site calibration enables you to pair up GNSS and local control points to be used in the calibration. (GNSS coordinates must be derived from GNSS points and observations, and grid points must be derived from grid points and terrestrial observations). This software then computes and applies the mathematical transformations using least squares.

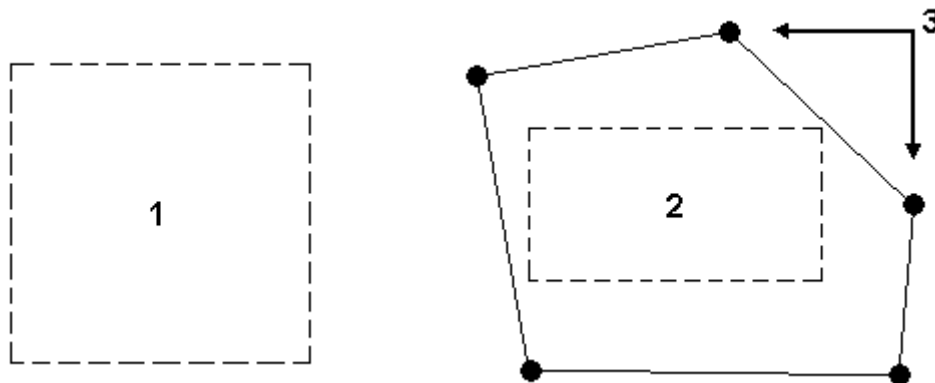
The mathematical transformations that are applied in order to convert WGS-84 positions to grid coordinates are:

1. A datum transformation to convert the WGS-84 latitude, longitude, and ellipsoidal height coordinates to latitude, longitude, and ellipsoidal height coordinates relative to the ellipsoid of the local map grid.
2. A map projection to convert the local ellipsoid latitude and longitude coordinates into local map grid northing and easting coordinates (the height value is not altered in this process).
3. A geoid model to WGS-84 height to get approximate elevation above sea level.
4. A horizontal adjustment of the transformed grid coordinates to best fit local control data. This adjustment allows for any local variations in the projection system that cannot be accommodated in the overall datum transformation.
5. A height adjustment to convert the heights above the local ellipsoid or elevations derived from the geoid to local control elevations above sea level.

The horizontal and vertical adjustment are stored as part of the coordinate system definition for the project. All GNSS points in the database are updated using the calibration parameters, resulting in more accurate local grid coordinate values.

You can save the new coordinate system definition (which includes the calibration parameters) as a site for use in future projects in the same area.

If you save a calibration as a site with the intention of using the site in another project, make sure that the project area is fully enclosed by the points used in the calibration. For example, in the following diagram, it is valid to use the site definition for project B, but not for project C.



1. Project C area

2. Project B area

3. Points used in calibrating Project A

If you use the **Here** key in Trimble® Survey Controller™ to start a Real Time Kinematic (RTK) base, and transfer the Survey Controller (.dc) file to this software, the base position, and therefore all rover points from the base, are of unknown quality (for all components, horizontal, height, and elevation).

**Note:** After performing a site calibration, if you apply local site settings, the calibration you have defined will become invalid and be removed. An error message warns you that continuing will remove the site calibration.

## Using Geoid Models

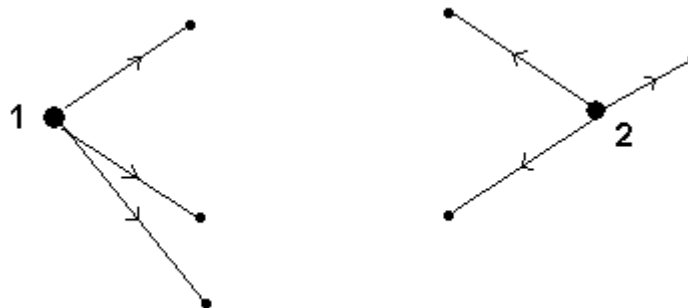
If the coordinate system for the project defines the use of a geoid model for point elevation determination, then elevations are determined directly off the WGS-84 heights by interpolation on the geoid model grid. However, it is still possible to apply a height adjustment on top of the elevations produced by a geoid model to allow for small local variations that a large scale geoid model cannot take into account. Elevations determined from a vertical calibration are given a survey quality.

## Rules for ensuring a useful calibration

- WGS-84 coordinates must be relatively correct.

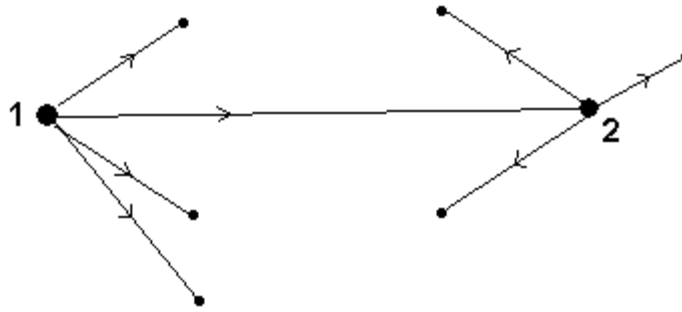
It is possible to generate autonomous GNSS points. However, you should not use more than one autonomous WGS-84 point in a calibration, for example make sure that only a single base station has been set up using the **Here** key in Trimble Survey Controller. Other base stations should be set up on positions measured by a GNSS vector in terms of the autonomous base station. This defines the relationship between them and allows you to perform a calibration with points used from either base station correctly. This is described below:

The following figure shows no relationship between the base stations:



Points from Base 1 and Base 2 should not be used for calibration.

In the following figure, a relationship is defined between the base stations by occupying the second base station using the first base station:



Points from Base 1 and Base 2 can be used for a calibration.

- The best possible WGS-84 coordinates should be used for the initial base station in a survey.

The precision of GNSS vectors (real-time or postprocessed) is affected by the accuracy of the base coordinate. An error of up to one part per million (1 ppm) can be introduced for each ten meters of error in the base coordinates. For example, if your primary WGS-84 reference point has an error of one hundred meters and your baseline is two kilometers long, you may have an unnecessary extra error of two centimeters in your GNSS vectors.

### Related topics

- [Calibrate a Site](#) (on page 184)

## Calibrate a Site

Calibrate sites to minimize residuals between WGS-84 RTK data you collect and local control coordinates. For a calibration, associate GNSS points with grid points at the same positions. These point pairs are used to compute and apply mathematic transformations (using least squares) to find the transformation that gives the adjustment parameters that best fit the control grid coordinates when applied to GNSS positions.

Calibrate a site if you:

- Did not calibrate in the field
- Need a report of quality control records
- Want to transfer a calibration to Trimble® Survey Controller™

Need to add extra points to a calibration in Trimble Survey Controller Horizontal calibrations consist of three parameters:




- Translation (move)
- Rotation (turn)
- Scale (shrink or stretch)

Vertical calibrations consist of two parameters:


- Lift (raise or lower)
- Tilt (change the northing and easting incline of the geoid or local plane)

Site calibration creates a set of local site settings. When a site calibration is complete, the site settings are used in the computation of all other imported GNSS data.


#### To calibrate a site:

1. Import or add your grid points using the **Add Point** command.
2. Import your GNSS data.
3. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Survey > GNSS Site Calibration**.

The **Site Calibration Calculation** command pane displays.

4. Click the **Calibration Settings** tab.
5. To compute a horizontal shift consisting of translations in the north/south and east/west directions, a rotation around a defined origin and a scale factor, leave the **Horizontal Calibration** box checked.
6. Check the **Set scale factor to 1** box if you want to maintain the scale of your horizontal distances.
7. To compute a vertical shift at a defined origin, leave the **Vertical Calibration** box checked.
8. Select **Vertical Shift + Incline Plane** in the list if you need to include inclinations in the north and east directions in the vertical calibration.
9. Click the  icon to display the **Project Settings** dialog, where you can define a geoid model, if needed.
10. Click the **Point List** tab to select the calibration point pairs that will be used in the calibration computations.
11. Click in the **GNSS Point** box.
12. Pick a GNSS point in a graphic view, right-click for options, or type a point ID in the box.

**Note:** The point you select must have global coordinates. Points with local coordinates cannot be selected as GNSS points. Point coordinates must be computed from global coordinates.

**Note:** Click  to expand a group if you want to view coordinate information on the selected point.

13. Click in the **Grid Point** box.
14. Pick a grid point (at the same location as the GNSS point) in a graphic view, right-click for options, or type a point ID in the box.
15. Select an option in the **Type** list to specify how the pair of points is to be calibrated. The calibration type you select must be valid for the points. For example, if the grid point does not have an elevation, you cannot set the type to **Horizontal and vertical**.
16. Repeat steps 11 - 15 to add additional pairs of points, if needed.

**Note:** There is no limit to the number of pairs that may be defined to compute a calibration. Adding a more pairs will not always improve the calibration results, but it will provide additional checks on the validity of the computed parameters. At minimum, you need three pairs of points for a horizontal calibration, and four pairs of points for a vertical calibration with an incline shift.

17. Click **Compute** to compute the GNSS calibration parameters. The **Results** tab appears, summarizing the transformation and listing the horizontal and vertical residuals for each pair of points. The narrow image indicates the vertical magnitude and direction of the shift, and the square image indicates the horizontal.
18. Click **Save As Site** if you want to make the calibrated site available to use as a coordinate system. The **Save Coordinate System as Site** command pane displays. Type a name for the site and click **OK**.
19. Click **Assign**. The calibration is recomputed to update all of the GNSS points, and the coordinate system details are updated with the calibration parameters.

**Note:** Sites can be recalibrated at any time.

#### To remove a site calibration:

- Select **Survey > Remove Site Calibration**. The calibration is removed, and the project is recomputed.

#### Related topics

- ❑ [Choose Local Site Settings](#) (on page 159)
- ❑ [Site Calibration Options](#) (on page 187)

## Site Calibration Options

Use these options to establish a relationship between the WGS-84 RTK data you collected and local control coordinates. They are available in the **Site Calibration Calculation** command pane.

### Options



Click this to display the **GNSS Calibration Report**, showing details of the calibration computation, all the computed parameters, and a listing of the computed control point coordinates compared with their known positions and individual residual values.



Click this to display the **Site Calibration** section of the **Project Settings**.



Click this to display the **Local Site Settings** command pane, where you can define the local site's coordinate type, scale factor, and offsets.



Click this to display the **Add Point** command pane, where you can create a new office-entered point.

## Calibration settings

### Horizontal calibration

Check this to compute a horizontal shift consisting of translations in the north/south and east/west directions, a rotation around a defined origin and a scale factor.

The transformation parameters are computed using least squares methods to find the transformation that gives the adjustment parameters that, when applied to the GNSS positions, best fit the control grid coordinates.

The horizontal adjustment reduces any residual error between the control coordinates and the grid coordinates calculated from the GPS positions.

### Set scale factor to 1

Check this to prevent your horizontal distances from being scaled.

---

**Tip:** It is wise to first compute a horizontal adjustment without the scale factor set to one to check the computed scale factor. If the computed scale factor is not close to one, it could indicate a problem in the selected calibration point pairs.

---

### Vertical calibration


Check this to perform an inclined plane adjustment consisting of a vertical shift at a defined origin and inclinations in the north and east directions.

The parameters for this adjustment are computed using least squares methods to find an adjustment plane that best fits the elevations derived from the GNSS heights with the control point elevations. This requires three three-dimensional calibration point pairs. With a single three-dimensional calibration point pair, only the vertical shift parameter can be computed. If there are two three-dimensional calibration point pairs available the system defines a correction plane that exactly fits these pairs.

If the project uses a geoid model, then the vertical adjustment is computed and applied on top of the geoid model corrections.

### Geoid model

This shows the name of the geoid model that is part of the coordinate system definition.


To [change the geoid model](#) (see "Change the Coordinate System" on page 149), click the  icon to display the **Project Settings** dialog.

**Vertical shift** - Select this to simply compute a vertical shift at a defined origin.

**Incline plane** - Select this to include inclinations in the north and east directions in the vertical shift.

### Compute

Click this to compute the GNSS calibration parameters after you have changed the calibration settings. Results are summarized in the **Computation Summary** on the **Calibration Settings** tab.

<b>Save as site</b>	<p>Click this to display the <b>Save as Site</b> dialog, in which you can save the current calibration and coordinate system details in the coordinate system database as a site definition.</p> <p>You can use this site definition as the coordinate system definition for future projects.</p>
<b>Computation summary</b>	<p>This shows a summary of the last calibration computation.</p> <p>Use it to confirm that the computed calibration is valid without having to look at the detailed computation report.</p>
<b>Point list</b>	
<b>GNSS point</b>	<p>Click in the box. Then pick a GNSS point in a graphic view, right-click for options, or type a point ID that you want to calibrate with the grid point in the next box.</p> <hr/> <p><b>Note:</b> The point you select must have global coordinates. Points with local coordinates cannot be selected as GNSS points. Point coordinates must be computed from global coordinates.</p> <hr/>
<b>Grid point</b>	<p>Click in the box. Then pick a grid point in a graphic view, right-click for options, or type a point ID that you want to calibrate with the GNSS point in the previous box.</p> <hr/> <p><b>Note:</b> Click  if you want to view coordinate information on the selected point.</p> <hr/>
<b>Type</b>	<p><b>Horizontal and vertical</b> - Select this when the point pair is suitable for determining both horizontal and vertical adjustments.</p> <p><b>Horizontal</b> - Select this when the point pair is suitable only for determining a horizontal adjustment (the elevation value for the grid point is not reliable).</p> <p><b>Vertical</b> - Select this when the point pair is suitable only for determining a vertical adjustment (the northing and easting values for the grid point are not reliable).</p> <p><b>Ignored</b> - Select this when the point pair is not to be used in the computation of any transformation parameters.</p> <p>This option is useful if you have a problem in your calibration computation and are trying to locate a suspect calibration point pair. You can use this to remove a calibration point pair temporarily from the computation to see if the results are improved.</p>

**Results**

**(summary)**

**Horizontal scale factor** - This displays the computed scale factor for the horizontal adjustment. If you check the **Set scale factor to 1** box, this displays “1”.

Use this to confirm that the computed scale factor is close to 1. If it is not, there may be a problem with one or more of the calibration point pairs.

**Horizontal rotation**

**Maximum slope of inclined plane** - This shows the maximum inclination for the computed height adjustment, based on the computed slope north and slope east values.

**Vertical shift at origin****Residuals**

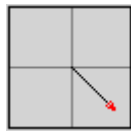
**Maximum horizontal residual** - This shows the highest horizontal difference between paired points.

**Maximum vertical residual** - This shows the highest vertical difference between paired points.

The residuals for each pair of points are represented by the images shown below.



These images indicate the relative magnitude of the vertical shift. Investigate and resolve the reported point pairs with the longest arrows first.



These images indicate the relative magnitude and direction of the horizontal shift.

Investigate and resolve the reported point pairs with the longest arrows and directions that do not match the other residuals first.

**Assign**

Click this to recompute so that all GNSS points are updated using the calibration parameters.

The coordinate system details are also updated.

**Related topics**

- ❑ [Calibrate a Site](#) (on page 184)

**Save a Calibrated Site as a Coordinate System**

After you perform a site calibration, you can name and save the site so it is available to use as a coordinate system.

**To save a site as a coordinate system:**

- Click **Save As Site** at the bottom of the **Site Calibration** command pane.

The **Save Coordinate System as Site** command pane displays.

**Related topics**

- ❑ [Calibrate a Site](#) (on page 184)
- ❑ [Change the Coordinate System](#) (on page 149)

- ❑ [Define a New Coordinate System](#) (on page 149)

## Separate Global and Grid/Local Coordinate Points for Site Calibration



If you import GNSS data and optical data into your project and one or more coordinates in the GNSS data have the same point ID as one or more coordinates in the optical data, points will be created under the **Points** node that includes both global and grid/local coordinates. In this case, a site calibration cannot be performed using these points. You must first rename the coordinate point IDs in the imported optical data file. Then, you must disable any remaining grid and/or local coordinates listed beneath points defined by global coordinates.

### To separate global and grid/local coordinate points for site calibration:

1. In the **Project Explorer**, change the point ID for each coordinate included in the optical data that has the same ID as a point included in the GNSS data:
  - a. Beneath the optical data file node, right-click the coordinate whose point ID you want to change, and select **Properties** from the context menu. The **Properties** pane displays.
  - b. In the **Properties** pane, enter a new point ID for the observation in the **Point ID** field and press **[Enter]**.

For example, if the original point ID is "a", you might enter a new point ID of "a\_grid".



The new point is created for the coordinate beneath the **Points** node in the **Project Explorer**.

- c. Repeat these steps for each coordinate point ID you want to rename.
2. Select **Survey > Site Calibration**.

If there are no grid and/or local coordinates listed beneath points defined by global coordinates, the **Site Calibration** pane displays. You are now ready to [calibrate the site](#) (see "Calibrate a Site" on page 184).

If there are grid and/or local coordinates listed beneath points defined by global coordinates, a message displays identifying the points. You should disable the grid and/or local coordinates for these points before performing the site calibration. Proceed with the next step in this procedure.

3. In the **Project Explorer**, disable any grid and/or local coordinates listed beneath points defined by global coordinates:

- a. Beneath the the **Points** node, click the  icon to expand the point you want to view.
  - b. Right-click the grid or local coordinate you want to disable, and select **Properties** from the context menu. The **Properties** pane displays.
  - c. In the **Properties** pane, select "Disabled" in the **Status** list.
  - d. Repeat these steps for each coordinate you want to disable.
4. When you are done, do either of the following to compute the project:
- Select **Project > Compute Project**.
  - Click the  icon on the status bar.

You are now ready to [calibrate the site](#) (see "Calibrate a Site" on page 184).

### Related topics

- ❑ [Understanding Site Calibration](#) (on page 181)
- ❑ [Calibrate a Site](#) (on page 184)

## Run a Site Calibration Report

After you calibrate a site, generate a **Site Calibration Report** to see details on the local site settings, horizontal and vertical calibration parameters, and residual differences between GNSS and grid points in your project.

### To run a Site Calibration Report:

- Select **Reports > Site Calibration Report**.  
The **Site Calibration Report** displays in your default Web browser.

### To modify the report:

- Select **Reports > Report Options**. Select **Site Calibration Report** in the command pane, and click **OK**.

In the **Settings** group at the bottom of the command pane, you can specify the header and footer data to display.

### Related topics

- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)




## CHAPTER 6

# Import Data


## Import Data

After you browse for a folder within the **Import** command, the files in that folder are analyzed to match a file type with an importer. The analyzer looks for identifying information within the file, the format of the file, and the file extension to assign an importer to the file. If an ASCII file is marked as "unknown", you may need to create a custom importer for that file.

### To import data:

1. Open an existing or new project.
2. Do one of the following:
  - Select **File > Import**.
  - Click the  icon on the toolbar.

The **Import** dialog displays.

3. Select a folder in the **Import Folder** list, or click the  icon to browse for a folder. The default is the folder that you last imported from. The files contained in the selected folder appear in the **Select File** area. The file names and file types are listed. File type is the name of the importer that is used to read the file.
4. Select the file(s) that you want to import in the **Select File** group.


**Note:** The order in which you import data can affect the computation results.


**Note:** If you need to set a custom importer, right-click on a file to set the file type.

5. Click **Import**. The data displays in graphic views and in the **Project Explorer**.

**Tip:** You can also double-click, or [drag and drop](#) (see "Drag and Drop to Import" on page 194), files to import them.

6. To view an [Import Summary](#) (see "Run an Import Report" on page 215) report, select **Reports > Import Summary**.

**Note:** To view the file, click the  icon. This will open the file in Notepad or another text editor program.

**Note:** To display only those files that the converter recognizes, click the  icon on the pane's toolbar. The icon functions as a toggle switch.

**Note:** To change the file type, right-click the file, and select an option from the **Set File Type** list. The options that are offered are based on internal scanning of the file. The file extension is also used as a clue.

**Note:** To import more than one file, use **[Ctrl] + click** or **[Shift] + click**.

**Note:** To change the file type, right-click, select **Set File Type**, and select one of the options in the drop-down list. The options that are offered are based on internal scanning of the file. The file extension is also used as a clue.

### Related topics

- ❑ [Drag and Drop to Import](#) (on page 194)
- ❑ [Import Data](#) (on page 193)
- ❑ Import Data Formats
- ❑ [Import Data in a Custom Format](#) (on page 209)
- ❑ [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)

## Drag and Drop to Import

You can import data by dragging files from your desktop, Windows® Explorer, or the *Import* command pane's file list, into a data view.

### To drag and drop a file into a project:

1. Open a project.
2. Do one of the following:
  - Locate the files that you want to import on your desktop or in Windows® Explorer.
  - Open the *Import* command pane and browse to the folder that contains the file(s) to import.
3. Click and drag the file(s) onto a data view.

**Caution:** The software allows you to set import properties when importing certain types of files. These properties are displayed in the *Import* command pane when you select **File > Import** and select the file type. If you choose to import one of these types of files using the drag-and-drop method rather than the *Import* command pane, you will not be able to see the default import properties or change them.

### Related topics

- ❑ [Import Data](#) (on page 193)

## Importable Data Formats

HEAVY CONSTRUCTION 


The *Import* command and data synchronization functions for sites, designs and work orders enable you to bring the following types of data into your project. See file-specific topics for details.

File format	Data types
.cal	Calibration files
.crd, .mos	<a href="#">GENIO</a> (see "Import GENIO Files" on page 198) alignment files
.csv, .pts	ASCII point files
.dc	<a href="#">Trimble data collector</a> (see "Import Data Collector Files (.dc)" on page 196) files
.dgn	<a href="#">MicroStation</a> (see "Import MicroStation Files (.dgn)" on page 204) design files
.dxf, .dwg	CAD files
.ini	site settings files
.reb, .*	<a href="#">REB</a> (see "Import REB Files (.reb)" on page 206) roadway and/or surface files
.ttm	<a href="#">Trimble terrain model</a> (see "Import Trimble Surface Files (.ttm)" on page 207) surface files
.txt	SCS work order report and record files
.xml	<a href="#">LandXML</a> (see "Import LandXML Files (.xml)" on page 200) files
.yxz	<a href="#">Wirth YXZ</a> (see "Import Wirth YXZ Files (.yxz)" on page 207) files

### Related topics

- ❑ [Import Data](#) (on page 193)
- ❑ [Import Data in a Custom Format](#) (on page 209)

## Import ASCII Files

ASCII files give you the flexibility to import data from a variety of sources, or even to create a file using a text editor. When you import an ASCII file, the *Import Format Editor* may display, prompting you to create a custom importer to accommodate the file. You can also access the editor any time by clicking the  icon on the *Import* command pane toolbar.

**Note:** You can import Nikon NEH files (.asc) from Field Surveyor.

**Tip:** ASCII point files (.pts) can be imported as a surface.

### Related topics

- [Import Data](#) (on page 193)
- [Import Data in a Custom Format](#) (on page 209)
- [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)

## Import Data Collector Files (.dc)

Import data collector files from a variety of field devices, including:

- Trimble® Survey Controller™ (up to version 10t; for files that have been converted)
- Trimble® Digital Fieldbook™ (v2,v3, and v5)

### Related topics

- [Import Data](#) (on page 193)
- [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)

## Import CAD Files (.dxf/.dwg)

CAD .dwg and .dxf files are drawing files used in architecture, civil engineering, design, and mapping industries. At the bottom of the import pane, you can specify settings that affect the imported data.

**Importable properties and objects:**

- 3D Face (as a CAD 3D Face)
- 3D Polyline (as a CAD 3D Poly Line)
- Arc (as a CAD Arc)
- Attribute (as a CAD Attribute Definition)
- Attribute Reference (as a CAD Attribute Reference)
- Block Reference (as a CAD Block Reference)
- Circle (as a CAD Circle)
- Coordinate (as a Coordinate)
- Ellipse (as a CAD Ellipse)
- Hatch (as a CAD Hatch)
- Leader (as a CAD Leader)
- Lightweight polyline (as a CAD Lightweight Polyline)
- Line (as a CAD Line)
- M Line (as a CAD Multi-line)
- M Text (as a CAD Multi-line Text)
- Point (as a CAD Point)
- Point (as a Point)
- Polyline (as a CAD PolyLine)
- Ray (as a CAD Ray)
- Shape (as a CAD Shape)
- Solid (as a CAD Solid)
- Spline (as a CAD Spline)
- Text (as a CAD Text)
- Trace (as a CAD Trace)
- Xline (as a CAD XLine)

**Non-importable properties and objects:**

- 3D solid
- Dim 3 Point Angular (importer)
- Dim Aligned (importer)
- Dim Angular (importer)

- Dim Diametric (importer)
- Dimension (importer)
- Dim Ordinate (importer)
- Dim Radial (importer)
- Dim Rotated (importer)
- Polyface mesh
- Polygon mesh
- Raster image
- Region
- Tolerance

#### Related topics

- [Import Data](#) (on page 193)
- [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)

## Import GENIO Files

Import GENIO data to create alignments. There are three types of GENIO strings that contain varying amounts of data and can be used in different ways.

## File Type

### GENIO 3D string

- X, Y, and Z coordinates
- 2D lines

These can be used as reference data for manually creating horizontal alignments, but they do not import as alignments, and cannot be converted into alignments.

### GENIO 6D string

- X, Y, and Z coordinates
- 2D lines
- Station data
- Instantaneous tangencies and radii

These can be converted into alignments with horizontal and vertical components after import.

### GENIO 12D string

- X, Y, and Z coordinates
- 3D lines
- Station data
- Tangencies and radii with additional parameters

These import as alignments with horizontal and vertical components.

## Related topics

- ❑ [Create an Alignment from a GENIO String](#) (on page 297)
- ❑ [Import Data](#) (on page 193)
- ❑ [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)
- ❑ [Workflow for Importing Alignments](#) (see "Workflow for Using Imported Alignments" on page 287)

## Import GNSS Job Files (.job)

Import GNSS files from the following field devices:

- TDS Survey Pro™ (Survey Pro Jobs)

**Note:** You can also import .raw (raw data files) from TDS Survey Pro.

- Spectra Precision® Field Surveyor
- Trimble® Survey Manager™
- Trimble® Digital Fieldbook™ (v2,v3, and v5)
- Trimble® Survey Controller™ (versions 11 and later)

### To force a kinematic occupation to process as static:

1. When importing a GNSS data file, select the file in the *Import* command pane *Files* list.

2. Select **Yes** in the **Force Static** box in the **Settings** group. The selected occupation is converted to static data, as indicated in the **Project Explorer**.

### Related topics

- ❑ Force Continuous Command
- ❑ Force Static Command
- ❑ [Import Data](#) (on page 193)
- ❑ [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)
- ❑ Run a Job File Report

## Import LandXML Files (.xml)

LandXML is an open XML file format. The format was principally created by and for the survey, civil engineering, and transportation industries. LandXML format supports points, surfaces, and alignments when imported. This software supports version 1.0 of the LandXML standard.

When you import a LandXML file, the data is previewed for conflicts. If there are no conflicts, the file is imported and the data appears in the graphic views. If conflicts are found, the **LandXML Import Conflicts** (see "Resolve LandXML Conflicts" on page 202) dialog displays, enabling you to resolve the conflicts. The import process continues afterwards. If there are errors after the import process, a warning flag appears on the status bar.

When you import LandXML files, the points, alignments, and surfaces are handled in these ways:



## LandXML Import Results

### Points

LandXML files can contain any number of points (referred to as COGO points in the LandXML format). When points are imported:

- They become points, with associated coordinates, and can be edited. These points can be referenced by a surface.
- They are placed on a layer named after the "points" section of the file. (If a layer does not exist, it is created.)
- Their point IDs are based on the "name" fields in the file.
- Their feature codes are based on the "desc" fields in the file.

### Alignments

LandXML files can contain any number of alignments. The required component of an alignment object is a geometric definition of the horizontal alignment.

Required attributes are name, alignment length, and beginning station value.

Optional attributes are station equations, profile, and cross-section components.

When alignments are imported:

- They are placed on new layers if the "alignment" sections of the file have unique names.
- Their names are based on the "name" fields in the file.
- Their geometric definitions are used to create alignments, with the geometry defining the horizontal component of the alignment. Horizontal alignment geometry can consist of lines, arcs, and spirals.
- If the "name" attribute exists in the geometric definition, it is used to name the horizontal alignment.
- The starting station value from the alignment object is used to station the horizontal alignment.

The alignment may also contain one or more vertical alignments (profiles). If a vertical alignment exists:

- It is used to create a representation of the vertical alignment in the profile view.
- The "name" attribute is used to name the vertical alignment.

If the alignment contains stored cross-sections:

- The stored cross-sections appear in both the plan view and the cross-section view.
- A surface is automatically created using the cross-section station values, offsets, and elevations upon import. The surface is named using the file name.

**Surfaces**

LandXML files can contain any number of surfaces.

A surface can be defined in the file in either (or both) of two ways:

- Source Data - Includes the points, point lists, boundaries, breaklines, and contours used to create the surface. This data imports as objects that are separate from the surface, meaning that you can add or remove them from the surface, or edit their properties.
- Definition - Includes 3D points and triangles defining the surface. This data imports as an integral part of the surface. This method can support holes and islands in surfaces.

**Related topics**

- ❑ [Resolve LandXML Conflicts](#) (on page 202)
- ❑ Results of Importing LandXML Files

**Resolve LandXML Conflicts**

During the LandXML import process, if any objects in the file are ambiguous, the **LandXML Import Conflict** dialog displays. LandXML objects that do not conform to the import format will be reported as conflicts or discarded if:

- The file is corrupted.
- A surface is only defined by a watershed.
- A surface contains one or many point files.
- A surface has both Source data and Definition data description.
- An alignment profile contains a gap.
- An alignment profile contains a spiral that is not a clothoid.
- The components of an alignment profile are not in the proper order.

**To resolve conflicts in LandXML objects:**

1. Select a conflict in the **List of conflicts**. Options for resolving the conflict appear in the **Selected conflict** area. The description and options in this depend on the selected conflict.
2. Click an option for resolving the conflict.
3. Click **Resolve this conflict**. A check mark appears in the **Status** column next to the conflict, indicating that the conflict is resolved. The next conflict in the list is automatically selected.
4. Repeat the steps above until all of the conflicts are resolved.

5. Click **Import** to finish the import process. When the file finishes importing, the LandXML objects appear in the project explorer as points, surfaces, or alignments. Discarded objects and the reasons for the discard are listed as **Errors** in the **Import Summary**.

**Notes:** You can change the resolution for solved conflicts any time before importing.

### Related topics

- ❑ [Import LandXML Files \(.xml\)](#) (on page 200)
- ❑ [LandXML Conflict Resolution Options](#) (on page 203)

## LandXML Conflict Resolution Options

Use these options to resolve ambiguities in the import of LandXML files. They are available in the **LandXML Import Conflicts** dialog. There are two general types of LandXML conflicts.

## Columns

<b>Status</b>	<ul style="list-style-type: none"> <li>✓ Resolved conflict</li> <li>? Unresolved conflict</li> </ul>
<b>#</b>	Shows the conflict number in the list
<b>Description</b>	Displays the type of object and the reason for the conflict

## Resolution Options

<b>Conflict type</b>	<p><b>Triangle-based definition</b> - Select this when you only want to import the triangles defining the surface. This data becomes the surface and cannot be edited.</p> <p>Select this only when you do not need to edit the data, and want a smaller file size and faster speed. This option also handles holes and islands in the data.</p> <p><b>Point/Breakline-based source data</b> - Select this to import the points, point lists, boundaries, breaklines, and contours used to create the surface. This data defines the surface, but remains separate from it, enabling you to edit the data to modify the surface.</p> <p>If you are unsure, select this option to retain the ability to edit the data.</p>
<b>Conflict type</b>	<p><b>Use a new name</b> - Select if you want to save the original imported object, and import an identical one with a new name. Type a new name in the box.</p> <p><b>Overwrite existing surface</b> - Select if you want to discard the existing surface of the same name, and replace it with this one.</p> <hr/> <p><b>Caution:</b> If you overwrite the existing surface, all related observed objects are deleted, even they are not from a LandXML import.</p>

## Related topics

- [Resolve LandXML Conflicts](#) (on page 202)

## Import MicroStation Files (.dgn)

MicroStation .dgn files are design files used in civil engineering, design, mapping, and architecture industries. At the bottom of the import pane, you can add a prefix to the layer (level) names, and specify whether to create a selection set comprised of the imported data.

**Importable properties and objects:**

- Arc (as an open CAD Ellipse or a CAD Arc (when minor and major axes are equal))
- B-spline (as a CAD Spline if header denotes a spline (includes non-uniform and rational B-spline))
- Cell (as individual CAD Lines and other CAD objects)
- Color (maps to default colors and bylayer for bylevel)
- Complex chain (as separate CAD Lines, 3D Polylines, Arcs, and Splines)
- Complex shape (as separate CAD Lines, 3D Polylines, Arcs, and Splines)
- Curve (as a CAD Spline)
- Ellipse (as a CAD Ellipse or a CAD Circle (when minor and major axes are equal))
- Level (as a layer)
- Line (as a CAD Line)
- Line string (as a CAD 3D Polyline)
- Line style (maps to one of the eight pre-defined line styles and to bylayer; all others map to solid)
- Multiline (as a CAD Multi-line)
- Point string (as CAD Points (if detached) or a CAD 3D Polyline (if contiguous))
- Shape (as a CAD 3D Polyline)
- Shared cell and cell reference (as a CAD Block and CAD Block Reference)
- Tag (as CAD Text)
- Text (as CAD Text)
- Text node (as separate CAD Text objects)

**Non-importable properties and objects:**

- 3D surface
- 3D solid
- B-spline (if header denotes a surface)
- Boundary
- Cone
- Conic
- Dimension
- External reference files

- Meshes
- Raster data
- Raster header

### Related topics

- ❑ [Import Data](#) (on page 193)
- ❑ [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)

## Import REB Files (.reb)

Import REB files to use as road model data, including alignments and surfaces, and/or surface data. REB data is separated into different files for these types of data: horizontal alignment, vertical alignment, cross-sections, points/coordinates, surface triangles, and breaklines.

You can select one or more **REB** files to import at the same time. Since different types of REB data are stored in separate files, if any data is dependent upon another type of data, both files must be imported at the same time. For example, you cannot add a vertical alignment file without concurrently adding the horizontal alignment file it is dependent upon.

Although REB files often use the .reb extension, they also use numeric extensions, such as: \*.021, \*.040, \*.066, \*.66, \*.D21, \*.D30, \*.D40, \*.D45, \*.D49, \*.D58, and \*.D66. REB extensions indicate the type of data the files contain:

- **.D21 and .021** - vertical alignment data
- **.D30** - point/coordinate data (GAEB)
- **.D40 and .040** - horizontal alignment data
- **.D45** - point/coordinate data
- **.D49** - breakline data (GAEB)
- **.D58** - surface triangle data (GAEB)
- **.D66 and .66** - cross-section data

**Note:** Cross-sections from REB data are converted into breaklines when they are imported.

**Note:** You can also import Wirth YXZ cross-section data in conjunction with an REB alignment file to create a road model.

**Note:** REB is a data specification most-commonly used in Germany.

### Related topics

- ❑ [Import Data](#) (on page 193)
- ❑ [Import Wirth YXZ Files \(.yxz\)](#) (on page 207)
- ❑ [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)

## Import Trimble Surface Files (.ttm)

Import triangulated terrain models to visualize surfaces and compute volumes between surfaces.

**Note:** You can also import ASCII point files (.pts) as surfaces using a custom format created in the [Import Format Editor](#) (see "Import Data in a Custom Format" on page 209).

### Related topics

- ❑ [Import Data](#) (on page 193)
- ❑ [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)
- ❑ [Workflow for Using Imported Surfaces](#) (on page 321)

## Import Wirth YXZ Files (.yxz)

Import cross-section data in a Wirth YXZ (.yxz) file to create a surface.

### Related topics


- ❑ [Import Data](#) (on page 193)
- ❑ [Run an Import Summary Report](#) (see "Run an Import Report" on page 215)

## Import DiNi Digital Level Files (.dat)




DiNi digital level .dat (M5) files contain level data recorded in the field using a Trimble DiNi Digital level.

### To import DiNi Digital level .dat files:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **File > Import**.

The *Import* command pane displays.

2. Do one of the following:
  - Select a folder in the *Import Folder* list.
  - Click the  icon to browse for a folder.

The default is the folder that you last imported from. The files contained in the selected folder appear in the *Select File(s)* area. The file names and file types are listed.

3. Select the DiNi digital level .dat file(s) you want to import.
1. In the **Settings** section at the bottom of the **Import** command pane, change the **Automatically Numbered Points** properties if necessary.

These properties help the software identify which points were auto-numbered by the DiNi level and which point IDs were entered by the user. Auto-numbered points typically are not points of interest and should not be created as points in the project.

- **Starting Point** - Specifies the first point ID number of the range of point ID numbers in the file you want to specify as automatically numbered points that should not be created in the project.
  - **Maximum Points** - Specifies the maximum number of points in the file you want to be included in the range.
  - **Increment** - Specifies the increment to be used when identifying point numbers in the range.
  - **Ending Point** - Specifies the calculated ending point for the point range based on the other specified properties.
2. When you are done, click **Import**. The **Level Editor** dialog displays. In this dialog, you can see which level points were entered by the user (points of interest) and will be created in the project, and which points were automatically numbered by the DiNi and will not be created in the project. If necessary, you can make changes by checking or unchecking any point to include or not include it in the project. For instructions, see [View and Edit Level Data](#) (on page 255).

**Note:** If you import a text file with "Unknown" or "Mapping" coordinate quality into a project that already contains level point data, duplicate points will be created for points in the text file ([lightweight points](#) (see "Understanding Point Types" on page 269)) and points already in the project ([normal points](#) (see "Understanding Point Types" on page 269)) that have the same ID (that is, points will not merge as expected). To avoid this problem, import the text file first to create the lightweight points in the project, then import the level point data. The lightweight points from the text file will merge with the normal points from the other point data to create normal non-duplicated points. For more information, see [Understanding Point Types](#) (on page 269).

### Related topics

- ❑ [Import Data](#) (on page 193)
- ❑ [View and Edit Level Data](#) (on page 255)

## Use Terramodel (.pro) Data

To use data from Terramodel (.pro) files in this software, the data must be exported from Terramodel in formats that can be imported:



- Export CAD linework and point blocks in CAD (.dxf) format

Export surfaces and alignments in LandXML (.xml) format **Note:** Roading data cannot be exported to a LandXML file.

Alternately, you can launch [SCS Data Manager](#) (on page 376) from within this software and use it to add a Terramodel (.pro) file to an SCS design outside of this program.


## Import Data in a Custom Format

Use the *Import Format Editor* to define a custom format for importing an ASCII file with a specifically defined format. The converters created with this editor are used within the Import command to import ASCII files with a non-standard format.

You can create a converter to import any of the following:

- Delimited files containing ASCII data that is separated by a specific character (e.g. .csv files)
- Fixed-width files containing ASCII data that is in pre-defined columns
- Files with ASCII data that is defined by a string of text
- Files with ASCII data that can be defined by a [regular expression](#) (see "Reference: Regular Expressions" on page 214)

### To import a custom format:

1. Do one of the following:
  - Select **File > Import Format Editor**.
  - In the *Import* dialog, click the  icon.


The *Import Format Editor* opens and displays the [Select Definition](#) (see "Definition Options" on page 210) dialog.

**Note:** If you try to import an ASCII format that is not recognized by any importers, the *Import Format Editor* may display automatically.

2. Select a custom format in the definition list.
3. Click **Next** and select options in the [Description and Search Type](#) (see "Definition Options" on page 210) dialog.
4. Click **Next** and select options in the [General Properties](#) (see "General Properties Options" on page 212) dialog.
5. Click **Next** and select options in the [Fields](#) (see "Fields Options" on page 213) dialog.
6. Click **Finish** to create the importer file.

**Tip:** For each format type, you can select to show or not show the *Import Format Editor* automatically, every time you import an ASCII non-standard file. Set this option in the [Select general properties](#) (see "General Properties Options" on page 212) dialog of the editor. When checked, the **Test** section displays the actual contents of the file and a sample of how it will be parsed using the selected format. For details, see [Fields Options](#) (on page 213).

#### To test a custom format importer:

1. Select a custom format in the definition list.
2. Click **Test** in any of the four *Import Format Editor* dialogs. The dialog expands.
3. Click **Read File** and select the number of lines you want the importer to read. If you select **View File**, it will open in a text editor.
4. Click the  icon and navigate to the type of file you want to import, and click **Open**. The importer will read the file and highlight any values that it is unable to convert.

**Note:** The file must have the same file extension as the importer you chose.

5. Select a different importer or edit the file to accommodate the reported errors.

#### Related topics

- ❑ [Definition Options](#) (on page 210)
- ❑ [Description and Search Type Options](#) (on page 211)
- ❑ [General Properties Options](#) (on page 212)
- ❑ [Fields Options](#) (on page 213)
- ❑ Import Data Formats
- ❑ [Reference: Regular Expressions](#) (on page 214)

#### Definition Options

Use these options to create new format definitions. These buttons appear to the right of the list of definitions on the first dialog of the *Import Format Editor*.

## Options

- New** Click this to enter a new definition name in the list. A unique name is required; a descriptive name is recommended. Click any other definition row to finish.
- Copy** After you click on description (listing on left), click **Copy** to enter a definition name. A unique name is required; a descriptive name is recommended. Click **OK** to return.
- Rename** Select the name of one of the custom formats you have created, and click this to edit the name.
- Delete** After you click on a description (listing on left), click this to remove the definition from the list.

**Note:** To remove a description as an import option, you can click the **Enable** checkbox until no green check appears. If you would do not want to display these disabled descriptions, enable **Only show enabled definitions** at the bottom left of the dialog box.

## Related topics

- ❑ [Description and Search Type Options](#) (on page 211)
- ❑ [General Properties Options](#) (on page 212)
- ❑ [Fields Options](#) (on page 213)
- ❑ [Import Data in a Custom Format](#) (on page 209)
- ❑ [Reference: Regular Expressions](#) (on page 214)

## Description and Search Type Options

Use these options to define the type of custom importer you want to create, and add a description. They are available in the second dialog of the **Import Format Editor**.

### Options

- Description** Enter a descriptive string to describe this importer (optional).
- Type** Select the option that describes the file you want to import:
- Delimited** - this file contains data that is separated by a specific character.
  - Fixed Width** - this file contains data that is in pre-defined columns.
  - Search for Text** - this file contains data marked on either side by a text string.
  - Regular Expression** - this file contains data that can be identified with a [regular expression](#) (see "Reference: Regular Expressions" on page 214).

### Related topics

- ❑ [Definition Options](#) (on page 210)
- ❑ [General Properties Options](#) (on page 212)
- ❑ [Fields Options](#) (on page 213)
- ❑ [Import Data in a Custom Format](#) (on page 209)

- ❑ [Reference: Regular Expressions](#) (on page 214)

## General Properties Options

Use these options to define how you want the file delimited and saved, and the data stored. They are available in the third dialog of the *Import Format Editor*.

### Options

<b>Delimiter</b>	Select the character that separates the fields from drop-down list. If you select <other>, you must specify the required character. This can be one of the following: _ ) ( * & ^ % \$ # @ ! ~ `
<b>Store points as</b>	<b>Surface</b> - creates a surface model from the points; no individual points are stored in the project. <b>Points</b> - creates individual points in the project.
<b>Default file extension (recommended)</b>	Enter the default extension for the import format. The import analyzer uses this extension to help it decide what conversion options to offer the user in the context menu. This field is optional. If left blank, a default extension of ".txt" is assumed.
<b>Show editor on import</b>	Check to automatically open the <i>Import Format Editor</i> when importing a non- standard ASCII file.
<b>Text qualifier</b>	Enter a special character to identify the beginning and ending of the string, for example " .
<b>Skip number of header lines</b>	Enter the number of lines to skip at the beginning of the file before reading the data to import.
<b>Start undefined ID numbering</b>	Enter a number to use as the starting ID when auto-generating IDs for unidentified points during import. If no number is entered, this software will not assign a number to unidentified points during import.
<b>Undefined elevation</b>	Enter a character or value to indicate that an elevation has not been assigned.
<b>Coordinate quality</b>	Select a coordinate quality to assign to the imported data. Control quality data is fixed, mapping and survey quality data is weighted, and unknown quality data is not used during the computation of the project.

**Tip:** For large point files, select *unknown* quality; the program will perform faster.  
**Note:** If you want to be able to edit the point quality within the software, select *control* or *survey* quality on import.

### Related topics

- ❑ [Definition Options](#) (on page 210)
- ❑ [Description and Search Type Options](#) (on page 211)

- ❑ [Fields Options](#) (on page 213)
- ❑ [Import Data in a Custom Format](#) (on page 209)
- ❑ [Reference: Regular Expressions](#) (on page 214)

## Fields Options

Use these options to define the fields that you want to import, and their field order, and units. The options vary slightly based on the type of converter you are creating. They are available in the fourth dialog of the *Import Format Editor*.

### Options

#### Fields

Click **Fields** to display a drop-down list of data properties. Select one and a tag appears as a field in the row of data. Continue to select all the fields that you want to import.

---

**Note:** If you select the properties out of order, you can click and drag them into the proper order.

---

#### Units

#### Apply to all

To select the distance units for all data, select the units and enable the **Apply to all** check box. You can also disable the **Apply to all** check box, and select a unit for each exported field.

#### For Fixed Width (only)

Click on each field, and enter a **Start** and **End** value or a **Start** and **Width** value - the third value will be filled in automatically.

#### For Search for Text (only)

Click on each field, and enter text values to search for the **Start** and **End** the field.

---

**Note:** Spaces will not be visible in the *Start* and *End* fields, but you can see them in the *Preview* area.

---

#### For Regular expression (only)

Click **Next** to display the dialog box for entering a [regular expression](#) (see "Reference: Regular Expressions" on page 214).

#### Test

#### Enter file name

Click **Test** to open the testing display area. To test the current format on a specific file:

#### Read file

4. Browse for the file.
5. Click **Read file** to view the results.

You can continue to modify the format setting and **Read file** the results until you are satisfied.

### Related topics

- ❑ [Definition Options](#) (on page 210)
- ❑ [Description and Search Type Options](#) (on page 211)
- ❑ [General Properties Options](#) (on page 212)
- ❑ [Import Data in a Custom Format](#) (on page 209)
- ❑ [Reference: Regular Expressions](#) (on page 214)

## Reference: Regular Expressions

A regular expression is a formula composed of characters and operators that represent a specific pattern. This formula is used to locate text strings that match this pattern.

A simple example is searching your computer for a list of all the files that have the .txt extension. To do this, you use the substitution formula \*.txt where \* represents any alphanumeric characters A-Z or 0-9. Similarly, regular expressions allow you to create a formula that represents the text pattern to search for.

Regular expressions can be simple or very complex. For example, you can write expressions to search for a:

- specific sequence of characters
- specific format such as (999)999-9999 to find phone numbers
- special characters such as spaces or tabs
- repeated words (or any text string)
- one text string always followed by another text string

By using operators in your expression, you can find text that matches a pattern or text that does NOT match the pattern.

For a quick tutorial on regular expressions, visit:

<http://www.codeproject.com/dotnet/RegexTutorial.asp>  
<http://www.codeproject.com/dotnet/RegexTutorial.asp>

### Syntax

The syntax for writing regular expressions is comprised of several subsets, including:

- **Substitutions** - characters and operators used in replacement patterns
- **Character Classes** - used to match Unicode, white- space characters, non-word characters, etc.
- **Regular Expression Options** - for modifying how a pattern is matched
- **Character Escapes** - used to indicate that a special character is to be matched
- **Quantifiers** - used to specify the number of matches to find

**Grouping Constructs** - used to match groups and sub- groups of text strings Syntax details are located at:

[http://en.wikipedia.org/wiki/Regular\\_expression](http://en.wikipedia.org/wiki/Regular_expression)  
[http://en.wikipedia.org/wiki/Regular\\_expression](http://en.wikipedia.org/wiki/Regular_expression)

### Related topics

- ❑ [Definition Options](#) (on page 210)
- ❑ [Description and Search Type Options](#) (on page 211)

- ❑ [General Properties Options](#) (on page 212)
- ❑ [Fields Options](#) (on page 213)
- ❑ [Import Data in a Custom Format](#) (on page 209)

## Run an Import Report

Generate an *Import Report* to see a project summary, details on imported files, and any associated errors or warning messages.

### To run an Import Report:

- Select **Reports > Import Report**.
- Select **Reports > Report Options**. Select *Import Report* in the command pane, and click **OK**.

The *Import Report* displays in your default Web browser.

**Tip:** Click a file name in the report to jump to the creation and import dates and times.

### Related topics

- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Troubleshoot an Import Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
LandXML data imports in the wrong location or configuration.	Project units are not set correctly.	Check the units shown on the status bar. If they are not correct, undo the import. Then, click the units name to open the <i>Project Settings</i> dialog. Change to the correct type of units, and reimport the file.
A LandXML file will not import. The program's importer says it is a LandXML file, but when you try to import it, a message says the file is invalid.	The file is valid XML, but not valid LandXML.	Open the file in your default Web browser. If it is corrupt, you will get an error message. If you do not get a message, it may be valid XML, but not valid LandXML, in which case the file needs to be recreated in a valid LandXML format.
Custom point data (.csv) imports at the wrong location.	<ol style="list-style-type: none"> <li>1. Your project units are not set correctly.</li> <li>2. The wrong custom import definition was selected; the northing and</li> </ol>	<ol style="list-style-type: none"> <li>1. Make sure that your project units are set correctly.</li> <li>2. Check the custom import definition you used to import the points. If you accidentally</li> </ol>

	easting order are reversed.	selected the definition with the northing and easting reversed, undo the import and reimport with the correct definition.
Duplicate points were created for points in an imported text file and points already in the project that have the same ID (that is, points were not merged as expected).	If you import a text file with "Unknown" or "Mapping" coordinate quality into a project that already contains point data, duplicate points will be created for points in the text file ( <a href="#">lightweight points</a> (see "Understanding Point Types" on page 269)) and points already in the project ( <a href="#">normal points</a> (see "Understanding Point Types" on page 269)) that have the same ID.	Import the text file into the project first to create the lightweight points, then import the other point data. The lightweight points from the text file will merge with the normal points from the other point data to create normal non-duplicated points.



## CHAPTER 7

# Transfer/Synchronize Data

## Prepare to Connect a Field Device

Transfer and synchronize data between field devices and your computer using one of two methods: Direct connection via the **Device** pane, or Office Synchronizer via the data synchronization area.

### Options

#### **Device pane** (on page 11)

Use this and a direct connection to update files directly on the field device and to import data directly from the field device.

#### **Office Synchronizer and the data synchronization area** (see "Office Synchronizer" on page 219)

Use these to manually or automatically synchronize data between a field device and your computer. Using the data synchronization area allows you to prepare field data and create export files without physical connection to the field device. The upload process is a separate automated step.

### To prepare for using a direct connection:

1. If necessary, install Microsoft® ActiveSync® software. If this wasn't done during installation, then do so now. (Enter `ActiveSync` as a search topic on the <http://www.microsoft.com/downloads> site.)

**Note:** For Microsoft Vista users, ActiveSync technology is not required. The communication functionality needed is included with Vista.

2. Start this software.
3. Connect the field device to the computer using a USB connection or a serial connection.
4. If the device asks if you want to be connected, click **Yes**. The **Device** pane appears, listing the files on the field device.

**Note:** When a field device is in direct connection mode, the *Office Copy* mode is not available.

### To prepare for data synchronization:

1. If necessary, install Microsoft® ActiveSync® software. If this wasn't done during installation, then do so now. (Enter ActiveSync as a search topic on the <http://www.microsoft.com/downloads> site.)
2. If necessary, install Office Synchronizer. If this wasn't done during installation, then install it from this software's installation CD.
3. Run Office Synchronizer, and select **Tools > Synchronizer Options**. Verify the following options:
  - **Sync options tab > Sync Mode = Manually**
  - **Display tab > Check Set ActiveSync to work in "Guest Only" mode**
4. Connect the field device to the computer using a USB connection or a serial connection. When you connect a device for the first time, enter the following data:
  - Device name - a unique name to be associated with the device
  - Field crew information (optional)
  - System root folder location - a folder on the computer or network location accessible to the computer. This folder will store the synchronized data. This data is stored on the field device.

### Related topics

- [Upload Files \(via Direct Connect\)](#) (see "Upload Files (via Direct Connection)" on page 221)
- [Download Files \(via Direct Connect\)](#) (see "Download Files (via Direct Connection)" on page 221)
- [Upload Files \(via data synchronization\)](#) (on page 224)
- [Download Files \(via data synchronization\)](#) (on page 223)

### Synchronizing Data for SCS

In preparing and managing site data, design data, and work orders for site controller software (SCS), you **do not** need to use the *Device* pane to synchronize with the data synchronization area. The synchronization happens automatically when you resolve conflicts and finish SCS commands.

## Office Synchronizer

Office Synchronizer is a separate utility that transfers data files between your computer and your CE-based field device/site controller, and verifies that the data in both locations is the same, or synchronized.

### Data synchronization area

The data synchronization area (also known as the synchronizer root folder) is the folder structure that stores synchronized field data on your computer or computer network. This folder is used by the Office Synchronizer utility, any field devices that have been synchronized with the Office Synchronizer, and this software. Files to be uploaded to field devices and files downloaded from field devices are located in the folder.

If necessary, you can enter, change, or verify the location of the data synchronization area by selecting **Tools > Options** and clicking **General** and **File Locations**. Check the **Synchronizer root folder** path.

### Related topics


- ❑ Export Data Formats
- ❑ [File Location Options](#) (on page 142)
- ❑ Import Data Formats
- ❑ [Prepare to Connect a Field Device](#) (on page 217)

## Device Pane

The **Device** pane enables you to directly access Microsoft® Windows® CE-based field devices or the data synchronization area (also known as the **root sync folder** in the Office Synchronizer utility), which contains the files maintained by Office Synchronizer.

### To display the Device pane:

Do one of the following:

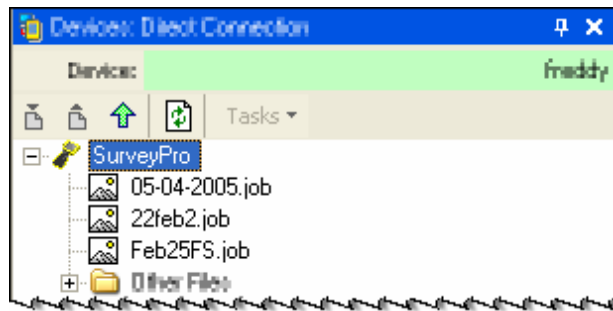
- Click the  icon on the toolbar.
- Select **View > Device Pane**.
- Press **[F10]** on the keyboard.

The **Devices** pane displays, docked on the left side of the application window, or where you positioned it last.

### To connect to a field device:

1. Connect the field device to the computer using a USB or serial connection.

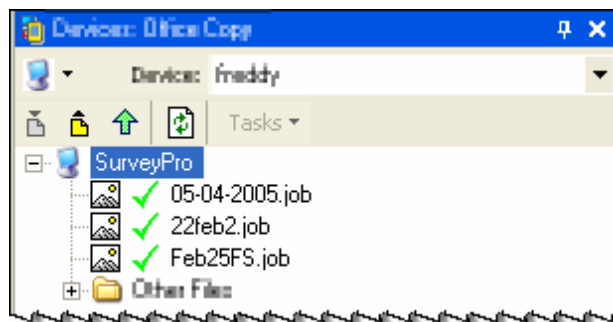
2. If the device asks if you want it to be connected, click **Yes**. The **Device** pane displays, showing a list of files on the device.




3. As needed, [upload](#) (see "Upload Files (via Direct Connection)" on page 221) or [download](#) (see "Download Files (via Direct Connection)" on page 221) files via this direct connection.

### To connect to field data in the data synchronization area:

In Office copy mode, the **Device** pane points to a folder on your office computer that contains the data previously synchronized from the field device, using the [Office Synchronizer utility](#) (see "Office Synchronizer" on page 219).



1. To verify that data in the synchronization area is selected, click the  icon, and verify that **Office Copy** is checked in the drop-down list. The contents of the synchronizer root folder display.
2. As needed, [upload files](#) (see "Upload Files (via data synchronization)" on page 224), [upload tasks](#) (see "Upload Tasks (via data synchronization)" on page 225), or [download files](#) (see "Download Files (via data synchronization)" on page 223) from the data synchronization area.

### Related topics


- ❑ [Office Synchronizer](#) (on page 219)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Prepare to Connect a Field Device](#) (on page 217)

## Direct Connection

### Download Files (via Direct Connection)

Download files to copy them from a connected field device to your office computer.

#### To download a file and import it into a project:

1. Open a project, or start a new project.
2. Connect the field device to the computer using a USB connection or a serial connection.
3. If the device asks if you want it to be connected, click **Yes**. The **Device** pane displays, listing the files on the field device.
4. Select one or more files to download from the device.
5. Do one of the following to import into the project:
  - Click the  icon on the toolbar.
  - Click and drag the selected file(s) into the plan view of the project.


#### Related topics


- ❑ Import Data Formats
- ❑ [Prepare to Connect a Field Device](#) (on page 217)
- ❑ [Upload Files \(via Direct Connection\)](#) (on page 221)

### Upload Files (via Direct Connection)

Upload files to copy them from your office computer to a connected field device.

#### To upload a file:

1. Open the project from which you want to export data.
2. If the field device is not connected:
  - Connect the field device to the computer using a USB connection or a serial connection.
  - If the device asks if you want it to be connected, click **Yes**. The **Device** pane appears within this office software, and lists the files currently on this field device.
3. Click the  icon on the **Device** pane toolbar. The **Export** pane displays, listing compatible formats in the **File Format** list.
4. Select the format to export.
5. If data has not been selected, select it, or click **Options** for selection options.

**Note:** You can also select the data to export prior to clicking the  icon.

6. Verify the default file name is correct, or enter a different name for the exported file in the **File Name** box.
7. Select any file specific settings in the **Settings** group.
8. Click **OK**. The exported file is converted to the appropriate format and uploaded to the field device. The file list in the **Device** pane updates, showing the new file name.

### Related topics

- ❑ Export Data Formats
- ❑ [Select from Plan View](#) (see "Select from 2D Views" on page 44)
- ❑ [Select via Command](#) (see "Selection Methods and Options" on page 43)
- ❑ [Prepare to Connect a Field Device](#) (on page 217)
- ❑ [Download Files \(via Direct Connection\)](#) (on page 221)

### Upload Tasks (via Direct Connection)

Upload files to field devices using the **Task** list on the **Device** pane toolbar. You can upload to:

- Trimble® Survey Controller™

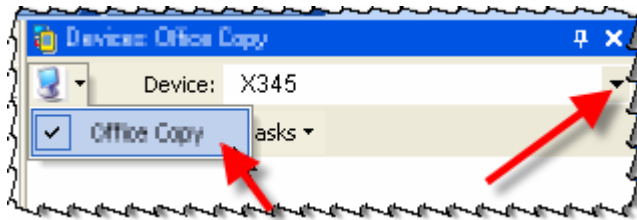
Trimble® Digital Fieldbook™ These file types can be uploaded:


- Feature code library (.fxl) files (converted to .fal files for Survey Controller versions prior to 11.3)
- Data dictionary (.ddf) files (converted to .fal files for Survey Controller versions prior to 11.3)
- Antenna (.ini) files
- Geoid (.ggf) files (including sub-grids)
- Datum grid files (.dgm)

#### To upload a file:

1. Connect the field device to the computer using a USB connection or a serial connection.
2. If the device asks if you want it to be connected, click **Yes**. The **Device** pane displays in this software.

3. Select the device to which you want to export the file in the *Device* list. If there is only one device available, this is not necessary.



4. Click **Tasks**, and select the file type from the list. The *Open* dialog displays for .fxl, .ddf, and .ini files. For .dgg files, the *Select Datum Grid Files for Upload* dialog displays, and for .ggf files, the *Geoid Sub-Gridding* command pane displays.
5. Browse to the file you wish to upload, and click **OK**, or click **Upload** in the *Geoid Sub-Gridding* command pane.
6. Click the  icon on the *Device* pane toolbar to see the exported file list.

#### Related topics

- ❑ [Prepare to Connect a Field Device](#) (on page 217)
- ❑ [Download Files \(via Direct Connection\)](#) (on page 221)
- ❑ [Upload Files \(via Direct Connection\)](#) (on page 221)

## Data Synchronization

### Synchronizing Data for SCS

In preparing and managing site data, design data, and work orders for site controller software (SCS), you **do not** need to use the *Device* pane to synchronize with the data synchronization area. The synchronization happens automatically when you resolve conflicts and finish SCS commands.


### Download Files (via data synchronization)

Before you download files, use [Office Synchronizer](#) (on page 219) to synchronize the field device from which you are importing a file.

Downloading a file creates an export file, and copies it to the data synchronization area for synchronizing.


#### To download a file and import it into a project:

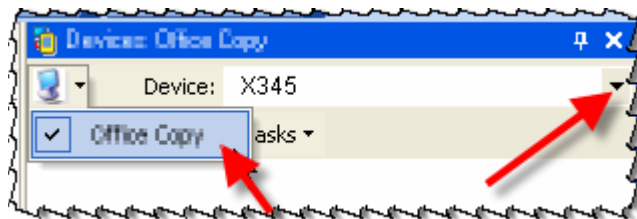
1. Open the project into which you want to download the file, or start a new project.
2. Open the *Device* pane by doing one of the following:
  - Select **View > Device Pane**.


- Click the  icon.
- Press [F10].

The **Device** pane displays.

**Note:** If a field device is directly connected, you will not be able to continue. Disconnect the device and start over.

3. Click the  icon, and verify that **Office Copy** is checked in the drop-down list. The contents of the data synchronization area display.
4. Select the device from which to import the files.



5. Select the files to import and do one of the following to import into the project:
  - Click the  icon.
  - Click and drag the selected file(s) to the plan view of the project.

### Related topics


- ❑ Import Data Formats
- ❑ [Prepare to Connect a Field Device](#) (on page 217)
- ❑ [Upload files \(via data synchronization\)](#) (on page 224)

## Upload Files (via data synchronization)

Upload files to copy them from your office computer to the data synchronization area. The synchronizer root folder is created when you synchronize field devices using the [Office Synchronizer utility](#) (see "Office Synchronizer" on page 219).

**Note:** To view or change the location of your data synchronization area, select **Tools > Options > File locations**.


### To upload a file:

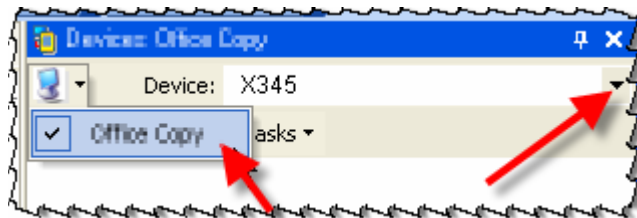
1. Open the project from which you want to export data.
2. Open the **Device** pane by doing one of the following:
  - Select **View > Device Pane**.
  - Click the  icon on the toolbar.





- Press [F10].

The **Device** pane displays.

3. Click the  icon and verify that **Office Copy** is checked in the drop-down list.
4. Select the device to which you want to export the file.



5. Select the data to export.
6. Click the  icon on the **Device** pane toolbar. The **Export** pane opens and displays a list of possible formats.
7. Select the file format to export. If data has not been selected, select it.
8. Verify the default file name is correct, or enter a different file name for the exported file.
9. Verify the settings options.
10. Click **OK**, and then close the **Export** pane to view the **Device** pane underneath.
11. Click the  icon on the **Device** pane toolbar to see the exported file list.

**Note:** Before taking the field device out to the field, the Office Synchronizer must synchronize the field device to which you are exporting a file.

### Related topics

- ❑ [Prepare to Connect a Field Device](#) (on page 217)
- ❑ [Download files \(via data synchronization\)](#) (on page 223)
- ❑ Export Data Formats

### Upload Tasks (via data synchronization)

Upload files to field devices using the **Task** list on the **Device** pane toolbar. You can upload to:

- Trimble® Survey Controller™
- Trimble® Digital Fieldbook™


**Note:** To activate the **Tasks** drop-down list, you must first [synchronize](#) (see "Office Synchronizer" on page 219) your field device.

These file types can be uploaded:


- Feature code library (.fxl) files (converted to .fal files for Survey Controller versions prior to 11.3)
- Data dictionary (.ddf) files (converted to .fal files for Survey Controller versions prior to 11.3)
- Antenna (.ini) files
- Geoid (.ggf) files (including sub-grids)
- Datum grid files (.dgm)

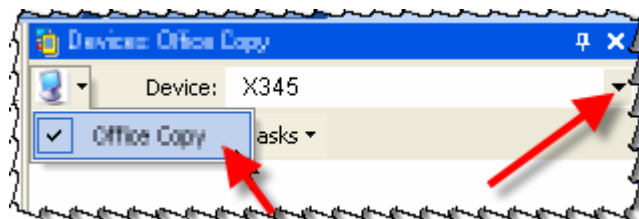
#### To upload a file:


1. Open the **Device** pane by doing one of the following:

- Select **View > Device Pane**.
- Click the  icon on the toolbar.
- Press **[F10]**.

The **Device** pane displays.

2. Click the  icon, and verify that **Office Copy** is checked in the drop-down list. The contents of the folder display.
3. Select the device to which you want to export the file.



4. Click **Tasks**, and select the file type from the list.
- For .fxl, .ddf, and .ini files, the **Open** dialog displays.
  - For .dgm files, the **Select Datum Grid Files for Upload** dialog displays.
  - For .ggf files, the **Geoid Sub-Gridding** command pane displays.
5. Browse to the file you wish to upload, and click **OK**, or click **Upload** in the **Geoid Sub-Gridding** command pane.
6. Click the  icon on the **Device** pane toolbar to see the exported file list. The new file will be copied to the field device the next time you synchronize.

### Related topics


- ❑ [Prepare to Connect a Field Device](#) (on page 217)
- ❑ [Upload files \(via data synchronization\)](#) (on page 224)

## Upload Geodetic Reference Data

### Upload a Datum Grid File

Select a datum grid file (.dgg) based on the datum used in the project coordinate system, and upload it to a field device.

#### To upload a datum grid file:

1. Connect the field device to which you want to upload a geoid. The **Device** pane displays.
2. On the pane's toolbar, click **Tasks**, and select **Upload datum (.dgg) file**. The **Datum Gridding** command pane displays.
3. If needed, select the folder containing the installed datum files in the **Folder** list, or click the  icon and navigate to the folder.

**Note:** The default location for .dgg files depends on your operating system:

In Windows® XP or earlier: **C:\Documents and Settings\All Users\Application Data\Trimble\GeoData** unless you have previously installed Trimble® Geomatics Office™ (TGO). If you have installed TGO and then this software, the path is **C:\Program Files\Common Files\Trimble\GeoData**.

In Windows Vista™: **C:\ProgramData\Trimble\GeoData\** or **C:\Program Files\Common Files\Trimble\GeoData\**.

4. Select a datum grid file in the **Datum Grid Files (.dgg)** list, if the datum you need isn't in the list, [create a datum grid file](#) (on page 174).
5. Click **Upload**. The datum file appears in the **Other Files** folder of the **Device** pane's tree.


**Note:** If you use the datum associated with your project's coordinate system, but do not create a datum grid file, it will not be saved in the list of datum grid files available the next time you upload. If it is a datum you will need again, create a datum grid file that will be saved.

### Related topics

- ❑ [Create a Datum Grid File](#) (on page 174)
- ❑ [Datum Grid Options](#) (on page 176)
- ❑ [Upload Tasks \(via Direct Connection\)](#) (on page 222)

## Upload a Geoid File

Select a geoid grid file (.ggf) based on the geoid used in the project coordinate system, and upload it to a field device.

1. Connect the field device to which you want to upload. The **Device** pane displays.
2. On the pane's toolbar, click **Tasks**, and select **Upload geoid (ggf) file**. The **Geoid Sub-Gridding** command pane displays.
3. If needed, select the folder containing the installed geoid files in the **Folder** list, or click the  icon and navigate to a folder containing .ggf files.
4. Select a geoid file in the **Geoid File** list.
5. Click **Upload**. The geoid file appears in the **Other Files** folder of the **Device** pane's tree.

**Note:** If the geoid grid file is larger than 1 MB, a confirmation message displays, asking if you really want to upload it. Consider [defining a sub-grid](#) (see "Define a Geoid Sub-Grid" on page 177) of the geoid to make the file smaller.

### Related topics

- ❑ [Geoid Options](#) (on page 177)
- ❑ [Define a Geoid Subgrid](#) (see "Define a Geoid Sub-Grid" on page 177)
- ❑ [Upload Tasks \(via Direct Connection\)](#) (on page 222)

## Troubleshoot a Data Transfer/Synchronization Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
Active Sync 4.5 will not run.	You are running as a limited user. Active Sync 4.5 is not compatible with limited user accounts.	Change you permissions to the administrator level, or download and use Active Sync 4.0. <b>Start &gt; Control Panel &gt; User Accounts &gt; User Accounts.</b> In the <b>User Accounts</b> dialog, select your user name in the list and click <b>Properties</b> . In the <b>Properties</b> dialog, click the <b>Group Membership</b> tab. Select <b>Other</b> , and <b>Administrators</b> in the list. Click <b>OK</b> twice to close the dialogs.

## CHAPTER 8

# Work with GNSS Data

## Planning Utility

Use the *Planning* utility to plan and schedule a GPS project based on good and bad satellite coverage information.

**To access the utility:**

- Select **Tools > Planning**.

**Note:** The *Planning* software has its own help system. Open the utility and select **Help > Index** from the *Planning* menu or press **[F1]** within the software.

## Check GNSS Data

### Run a Point List Report

Generate a *Point List* to see a simple summary of the coordinates for each point in your project.

**To run a Point List report:**

- Select **Reports > Point List**.

The *Point List* displays in your default Web browser.

**To modify the report:**

- Select **Reports > Report Options**. Select **Point List** in the command pane, and click **OK**.
- In the **Settings** group at the bottom of the command pane, you can specify the type of coordinates (grid, local, or global), and the type of data to display. The data options include quality control information such as scale factors and convergence angle.

**Tip:** Click a point ID in the report to select the point in graphic views and the **Project Explorer**.

**Related topics**

- [Customize and Run a Report](#) (see "Customize a Report" on page 436)

**Run a Point Derivation Report**

Generate a **Point Derivation Report** to see details on the survey data used to calculate the final coordinates of points in your project.

**To run a Point Derivation Report:**

- Select **Reports > Point Derivation Report**.
- Select **Reports > Report Options**. Select **Point Derivation Report** in the command pane, and click **OK**.

The **Point Derivation Report** displays in your default Web browser.

**Tip:** Click a point ID or coordinate in the report to select the point in graphic views and the **Project Explorer**.

**Related topics**


- [Customize and Run a Report](#) (see "Customize a Report" on page 436)

**Calculate the Inverse Between Points**

Calculate and report inverse values between any two points in your project, such as:

- Grid distance
- Change in elevation

Geodetic azimuth To calculate the inverse between two points:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Survey > Inverse**.

The **Inverse** command pane displays.


2. Select **Sequential** to calculate values from point to point in series (as if drawing a multi-segment line), or **Radial** to calculate values from one point to multiple other points (as if drawing a fan).



**Note:** You can switch between **Sequential** and **Radial** after picking any pair of points.

3. Pick the first point in a graphic view, or type a point ID in the **From** box.

**Note:** You can also right-click in the view to access [COGO controls](#) (see "Understanding COGO Controls" on page 86) and [snaps](#) (see "Snaps Modes and Commands" on page 89) when picking points.

4. Pick another point, or type a point ID in the **To** box. The point IDs appear in the **Reported Points** group, and the inverse values appear in the **Details** group.

**Note:** If **Free** appears in the **Reported Points** list, no point with a point ID was within the pick aperture. To prevent picking where there are no points, click the  icon on the **Inverse** command pane's toolbar. In the **Snap Mode** dialog, uncheck **Free**, and click **OK**.

5. To calculate additional inverses, continue picking **To** points.
6. To review the details for any inverse, click in the first column of the **Reported Points** list.
7. To change inverse report options, click the  icon to display the **Report Options** command pane. When you are done, click **OK** to return to the **Inverse** command.
8. To generate the **Inverse Results** report, click the  icon at the top of the **Reported Points** group. The **Inverse Results** report displays in your default web browser.

**Note:** If no coordinate system is defined, the **Select Coordinate System** dialog displays. Define a coordinate system and run the report again.

9. Click **Close**.

### Related topics

- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)
- ❑ [Inverse Options](#) (on page 231)
- ❑ [Measure Values Between Points](#) (on page 135)

## Inverse Options

Use these options to calculate and report the azimuth, distance, and other relationships between any two points. They are available in the **Inverse** command pane.

## Options



Click this on the **Inverse** command pane's toolbar to display the **Snap Mode** dialog, where you can enable and disable running snap modes.

## Inverse

### From/To

Pick points in graphic views, or type point IDs in the boxes and click **Apply** or press **[Enter]**.

---

**Note:** If **Free** appears in the **Reported Points** list, no point with a point ID was within the pick aperture. To prevent picking where there are no points, click the icon on the **Inverse** command pane's toolbar. In the **Snap Mode** dialog, uncheck **Free**, and click **OK**.

---

**Sequential** – Data is collected in a line, and you want to verify distances around the traverse.

- For example, you will need to click on:  
A to B, B to C, C to D, D to E, and E to A.

**Radial** – Data is collected in a ray, and you want to check distance from the base station.

- For example, you will need to click on: A to B, A to C, A to D, and A to E.

## Reported points



Click this to display the **Inverse Results** report in your default web browser.



Click this to display the **Report Options** command pan, in which you can specify heading, footer, and format settings for the **Inverse Report**.

### From point ID/

### To point ID

Click in the first column of any row to list details for the inverse of the points.

### Details

This shows the azimuths, changes in elevation and height, and three distances of the selected inverse:

- Grid
- Ellipsoidal
- Ground

### Apply

This acts as the **[Enter]** key, when specifying points, moving the focus between **From**, **To**, and **Reported Points**.

## Related topics

- ❑ [Calculate the Inverse Between Points](#) (on page 230)



## CHAPTER 9

# Work with Total Station Data



## Understanding Total Station Data



Survey data acquired in the field using a total station and contained in a data file can be imported into the software and integrated, as necessary, with other data collected as part of a survey project (for example, GNSS or leveling data).

Total station data is displayed in the following areas of the software:

- **Project Explorer**  
Imported data is displayed in expandable nodes displayed beneath the *Import Files* node. Data used to define points is displayed in expandable nodes displayed beneath the *Points* node. You can expand and collapse nodes as necessary to view information. For more information, see [View Total Station Data in Project Explorer](#) (on page 240).
- **Properties** pane  
Click any node in the *Project Explorer* to display its properties in the *Properties* pane. You can edit certain properties in the pane.
- **Mean Angle Residuals** dialog  
Right-click any mean angle node in the *Project Explorer* and select **Mean Angle Residuals** to view residuals for the mean angle and, if you want, disable any outlying observations. For more information, see [View and Edit Mean Angle Residuals](#) (on page 248).
- **Mean Angle Report**  
Select **Reports > Mean Angle Report** to view details of how each mean angle was computed. For more information, see [Run a Mean Angle Report](#) (on page 249).

- Graphical and spreadsheet views

View point information in any of the available graphic or spreadsheet views.

The type of total station data displayed in the software depends on the [station setup type](#) (see "Total Station Setup Types" on page 234) and [measurement types](#) (see "Total Station Measurement Types" on page 236) used for the survey.

### Related topics

- ❑ [Total Station Setup Types](#) (on page 234)
- ❑ [Total Station Measurement Types](#) (on page 236)
- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)
- ❑ [View and Edit Mean Angle Residuals](#) (on page 248)
- ❑ [Run a Mean Angle Report](#) (on page 249)

## Total Station Setup Types



The software supports any of the following types of station setups in the imported total station data file. The different station setups support specific geometrics of known points for orientation.

**Station setup type**

**Description**

**single backsight**

The coordinate for the occupied point must be keyed in, previously measured, or filled in at a later time.

The coordinate for the backsight or the azimuth to the backsight must be keyed in or previously measured.

All horizontal angles are measured relative to the backsight.

**station setup plus (multiple backsight)**

The coordinate for the occupied point must be keyed in, previously measured, or filled in at a later time.

The coordinates for the backsights or the azimuths to the backsights must be keyed in or previously measured.

All horizontal angles are measured relative to the first backsight.

This setup has redundant information so it is easy to detect errors in the backsight coordinates, azimuths, or instrument setup.

**standard resection**

This setup type requires at least two backsight points.

The coordinates for the backsights or the azimuths to the backsights must be keyed in or previously measured.

All angles are measured relative to the first backsight.

This setup can have redundant information so it is easy to detect errors in the backsight coordinates, azimuths, or instrument setup.

**Helmert resection**

This setup type is a variation of a standard resection setup type. Instead of using least squares to find the best fit of known points and observed data, a Helmert coordinate transformation is used.

**refline**

This setup type requires two points on a line, or one point and the azimuth of the line.

The coordinate for the first point on the line must be keyed in, previously measured, or filled in at a later time.

The azimuth to the second point or the coordinate for the second point must be keyed in or previously measured.

### Related topics

- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [Total Station Measurement Types](#) (on page 236)
- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)
- ❑ [View and Edit Mean Angle Residuals](#) (on page 248)
- ❑ [Run a Mean Angle Report](#) (on page 249)

## Total Station Measurement Types



The software supports any of the following types of measurements in the imported total station data file.

**Measurement type**

**Description**

**distance offset**

The surveyor measures to a convenient point near the object and enters Left/Right, Up/Down and In/Out position offsets so the object position can be calculated from the measured point.

You can edit the three offset values and direction after importing the data.

**horizontal angle offset**

The surveyor measures to the side of the object, then he turns a horizontal angle to the center. The offset is a combination of the slope readings taken to a prism and the horizontal angle after aiming the instrument to the center of the measured object.

Because these are measured values, you cannot edit them after importing the data.

**vertical angle offset**

The surveyor measures to a point above or below the object, then he turns a vertical angle to the center. The offset is a combination of the slope readings taken to a prism and the vertical angle after aiming the instrument to the center of the measured object.

Because these are measured values, you cannot edit them after importing the data.

**angle offset**

The surveyor measures to a point to the side of the object and above/below the object, then he turns a horizontal and vertical angle to the center. The offset is a combination of the slope readings taken to a prism and the angles after aiming the instrument to the center of the measured object.

Because these are measured values, you cannot edit them after importing the data.

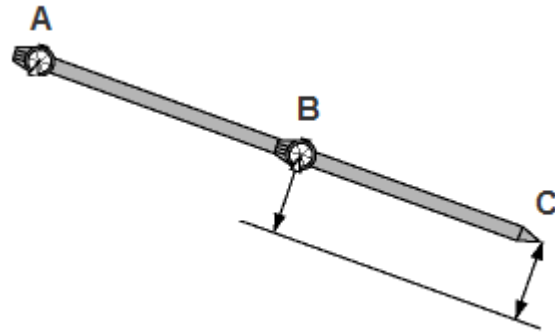
**circular object**

The surveyor measures to a point directly in front of a circular object, then he turns an angle to either edge. The center and radius are calculated from these two measurements.

Because these are measured values, you cannot edit them after importing the data.

**dual prism offset**

The surveyor measures to prism A and then to prism B. The position of the rod point is calculated from these measurements and from the B-C distance (the distance from the rod point to the nearest prism) the surveyor keys in.



You can edit the B-C distance and prism constants after importing the data.

### Related topics

- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [Total Station Setup Types](#) (on page 234)
- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)
- ❑ [View and Edit Mean Angle Residuals](#) (on page 248)
- ❑ [Run a Mean Angle Report](#) (on page 249)

## Workflow for Total Station Data



Use the following steps as a guideline for working with total station data you have imported into a project:

1. [Import](#) (see "Import Data" on page 193) the total station data file into your project.  
The data is displayed as [nodes](#) (see "View Total Station Data in Project Explorer" on page 240) located beneath the *Imported Files* node and the *Points* node in the *Project Explorer*.
2. Click **Reports > Import Report** to view information about the import and any related messages. The *Import Summary* report displays in a browser window.
3. Run a *Survey Report* to view complete information about imported total station data contained in a .job or .jxl file. For instructions, see Run a Job File Report. Be sure to select the *Survey Report* style sheet for the report.
4. In *Project Explorer*, click the new imported data file node, which is located beneath the *Imported Files* node, and verify job information in the *Properties* pane. (See [View Total Station Data in Project Explorer](#) (on page 240).)

5. If necessary, fill in any coordinates from standard references.
6. Review the chronology and correctness of the imported data in the *Project Explorer*.  
Expand the nodes under the *Imported Files* node and review in chronological order (top to bottom) to ensure data was gathered correctly and good survey practices were followed.
7. As necessary, view information in the *Project Explorer* and in the *Properties* pane to verify information is correct.

In the *Properties* pane, you can edit data entered manually in the field. But, you cannot edit measurements made by the device. If you find an error that cannot be corrected because the *Properties* pane field is read-only, the field crew must re-survey that portion of the project. You can delete the old station nodes that contain errors and import the file that contains the re-surveyed stations. If necessary, you can delete any redundant information.

**Tip:** If a coordinate or azimuth appears to be missing, look for it earlier in the file. It could have been keyed in as part of an earlier setup or it may have been determined from a resection. Coordinates determined as part of a traverse appear later in the file because they can only be determined after the traverse is complete.

**Tip:** To determine the angle between two observations, expand the nodes under the station from which the first observation is made. Click on that observation to view the horizontal angle to the first backsight. Then, click on the station setup node to view the orientation angle of the first backsight. Add these two numbers to determine the orientation angle of the first observation. Repeat this procedure for the second observation and subtract the two orientation numbers.

8. In *Project Explorer*, right-click any mean angle node and select **Mean Angle Residuals** from the context menu to display the *Mean Angle Residuals* dialog.  
Use this dialog to [view and edit mean angle residuals](#) (on page 248).
9. Select **Reports > Mean Angle Report** to [view details](#) (see "Run a Mean Angle Report" on page 249) of how each mean angle was computed.
10. Select **Reports > Point Derivation Report** to see how multiple observed measurements to the same point are being used by the software to compute the resultant coordinate, and how the measurements compare to each other.

**Tip:** When viewing the *Point Derivation Report*, you are especially looking for unacceptably high residuals.

After total station data is imported into your project, you can view it in the graphical and spreadsheet views.

### Related topics

- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)

- [View and Edit Mean Angle Residuals](#) (on page 248)
- [Run a Mean Angle Report](#) (on page 249)

## CHAPTER

### View Total Station Data in Project Explorer



Total station data is displayed as individual nodes under two top-level nodes in *Project Explorer*:

- **Imported Files** node

This top-level node includes one or more data file nodes, each specifying a file containing GNSS, total station, and/or level field data that has been imported into the project. Individual total station nodes display beneath the file node(s) for imported total station data. Total station nodes are typically listed in the same chronological sequence as data was gathered in the field. For more information, see [Total Station Nodes Sequence in Project Explorer](#) (on page 246).

- **Points** node

This top-level node includes a node for each point in the project. Individual total station nodes are displayed as appropriate beneath the points to which they pertain.

You can double-click any node to display its properties in the *Properties* pane. You can right-click any node to display a context menu that includes other options, depending on the node type.

For a description of how total station nodes are displayed in the *Project Explorer*, see the following topics:

- [View Imported Total Station Data in Project Explorer](#) (on page 241)
- [View Total Station Data Associated with a Point in Project Explorer](#) (on page 244)

#### Related topics

- [Understanding Total Station Data](#) (on page 233)
- [Workflow for Total Station Data](#) (on page 238)
- [Total Station Nodes Sequence in Project Explorer](#) (on page 246)
- [Total Station Data Errors](#) (on page 247)
- [View and Edit Mean Angle Residuals](#) (on page 248)



- [Run a Mean Angle Report](#) (on page 249)

## View Imported Total Station Data in Project Explorer



Each imported total station data file displays as an expandable node located beneath the *Imported Files* node in *Project Explorer* (select **View > Project Explorer**).

The types of nodes displayed beneath the data file node depend on the station setup type and measurement types contained in the file.

The sequence of nodes (top to bottom) displayed beneath the imported data file node reflects the chronological order in which data was gathered in the field. For more information, see [Total Station Nodes Sequence in Project Explorer](#) (on page 246).

The following table shows the possible node types and their relative position beneath the *Imported Files* node in *Project Explorer*.

**Note:** The following table is not intended to represent a single data file.

Node type example	Description
Imported Files	This is the Imported Files node.
Test.jxl	This is the imported data file node. The icon represents the source of the data. It is followed by the name of the file.
100 (S1)	This is a station node. The icon represents the survey instrument type. It is followed by the point ID and an "S" number in parenthesis that uniquely identifies the station.
Used last station setup	This is a note node. The icon is followed by the first few words of the note.
Single Backsight (99)	This is the station setup node. The icon is followed by the setup type (in this example, "single backsight") and the associated backsight point ID(s) in parenthesis. Nodes displayed beneath this one are typically coordinate and azimuth nodes, which show data that was keyed in during station setup.



100

This is a coordinate node.

The icon is followed by a coordinate point ID.

This node is typically displayed beneath a station setup node.

This is a coordinate node from a benchmark. The icon is followed by a coordinate point ID.

---

**Note:** To preserve the integrity of raw field data, you cannot edit coordinate properties for an imported coordinate displayed beneath the *Imported Files* node. However, you can edit coordinates displayed beneath the *Points* node.

---



100-99 (A1)

This is an azimuth node.

The icon is followed by the occupation point ID, a dash, and the the observed point ID. This is followed by an "A" number in parenthesis that uniquely identifies the azimuth.

This node is typically displayed beneath a station setup node.



Backbearing 99 (R1)

This is a backbearing node.

The icon and name are followed by the point ID of the first backsight point. This is followed by an "R" number in parenthesis that uniquely identifies the backbearing.

Nodes displayed beneath this one are typically related observation nodes.



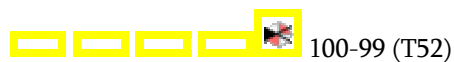
100-200 (T41)

These are face 1 and face 2 total station observation nodes.



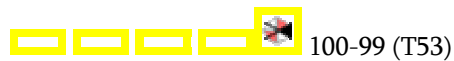
100-200 (T42)

The icon is followed by the occupation point ID, a dash, and the the observed point ID. This is followed by a "T" number in parenthesis that uniquely identifies the observation.



100-99 (T52)

These are face 1 and face 2 backsight observation nodes.

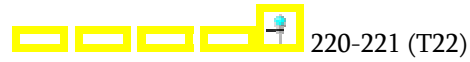


100-99 (T53)

The icon is followed by the occupation point ID, a dash, and the the observed point ID. This is followed by a "T" number in parenthesis that uniquely



201-202 (T54)



220-221 (T22)



110-111 (T5)



198-199 (T28)



99-100-200 (M3)



Rounds



Set 1

identifies the observation.

This is a dual prism observation node.

The icon is followed by the occupation point ID, a dash, and the observed point ID. This is followed by a "T" number in parenthesis that uniquely identifies the observation.

These are distance, angle, and circle offset nodes.

The icon is followed by the occupation point ID, a dash, and the observed point ID. This is followed by a "T" number in parenthesis that uniquely identifies the observation.

This is a mean angle node. It represents the combining and averaging of redundant observations to the same point.

The icon is followed by the backsight point ID, a dash, the occupation point ID, another dash, and the observed point ID. This is followed by an "M" number in parenthesis that uniquely identifies the mean angle.

For more information on viewing and working with mean angles, see [View and Edit Mean Angle Residuals](#) (on page 248) and [Run a Mean Angle Report](#) (on page 249).

This is a rounds node. The icon is followed by the word "Rounds".

Displayed beneath it are two or more sets of observations, which are represented by set nodes.

This is a set node. It is nested under a rounds node.

The icon is followed by the word "Set" and a number that identifies the set.

Displayed beneath it are nodes representing observations made during the set.

## Related topics

- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)
- ❑ [View Total Station Data Associated with a Point in Project Explorer](#) (on page 244)
- ❑ [Total Station Nodes Sequence in Project Explorer](#) (on page 246)
- ❑ [View and Edit Mean Angle Residuals](#) (on page 248)
- ❑ [Run a Mean Angle Report](#) (on page 249)

## View Total Station Data Associated with a Point in Project Explorer



Individual point nodes are displayed beneath the **Points** node in *Project Explorer*.

Beneath each point node, all total station, GNSS, and/or level data used to define the point are displayed as individual nodes. For total station data, this includes observation, mean angle, coordinate, and azimuth nodes.


**Note:** Station, station setup, backbearing, round, and set nodes, which are displayed beneath the **Imported Files** node, are not displayed under the **Points** node in *Project Explorer*.

The following table shows the possible node types and their relative nested position beneath the Points node in *Project Explorer*.

### Node type example

 Points

 100

 Grid (Single Backsight.jxl)

### Description

This is the points node.

This is an individual point node. The icon is followed by the point ID.

Observations to and from the point are listed as observation nodes beneath it. See the **Point Derivation Report (Reports > Point Derivation Report)** to see how observations were used to establish the point.

If a point ID is displayed in red, there is a computational error associated with the point. For instructions on viewing computational error messages, see [Select from the Flags Pane](#) (on page 48). For a description of total station errors that can cause a red point ID, see [Total Station Data Errors](#) (on page 247).

This is a coordinate node.

The icon is followed by the coordinate

 100-99 (A1)

type and the source of the coordinate in parenthesis.

This is an azimuth node.

The icon is followed by the occupation point ID, a dash, and the observed point ID. This is followed by an "A" number in parenthesis that uniquely identifies the azimuth.

 99-100-200 (M3)

This is a mean angle node. It represents the combining and averaging of redundant observations to the same point.

The icon is followed by the backsight point ID, a dash, the occupation point ID, another dash, and the the observed point ID. This is followed by an "M" number in parenthesis that uniquely identifies the mean angle.

For more information on viewing and working with mean angles, see [View and Edit Mean Angle Residuals](#) (on page 248) and [Run a Mean Angle Report](#) (on page 249).

 100-200 (T41)

This is a face 1 observation node.

The icon is followed by the occupation point ID, a dash, and the observed point ID. This is followed by a "T" number in parenthesis that uniquely identifies the observation.

 100-99 (T52)

This is a face 1 backsight observation node.

The icon is followed by the occupation point ID, a dash, and the observed point ID. This is followed by a "T" number in parenthesis that uniquely identifies the observation.

 201-202 (T54)

This is a dual prism observation node.

The icon is followed by the occupation point ID, a dash, and the observed point ID. This is followed by a "T" number in parenthesis that uniquely identifies the observation.

 220-221 (T22)

These are distance, angle, and circle offset nodes.

 110-111 (T5)

The icon is followed by the occupation point ID, a dash, and the the observed



198-199 (T28)

point ID. This is followed by a "T" number in parenthesis that uniquely identifies the observation.

### Related topics

- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)
- ❑ [View Imported Total Station Data in Project Explorer](#) (on page 241)
- ❑ [Total Station Data Errors](#) (on page 247)
- ❑ [View and Edit Mean Angle Residuals](#) (on page 248)
- ❑ [Run a Mean Angle Report](#) (on page 249)

## Total Station Nodes Sequence in Project Explorer



The sequence of nodes (top to bottom) nested beneath the imported data file node in the *Project Explorer* reflects the chronological order in which data was gathered. For example, the station setup node is above observations nodes because this is the sequence of events in the field.

Mean angle nodes are beneath rounds and backbearing nodes because observations must occur before calculations.

Even a backbearing node follows this rule, although it may appear not to. Consider a station setup plus in which observations to the backsight points are made to establish the orientation. It may seem as though the observations are collected first and then a backbearing is determined from them. But, what actually happens is very much like a surveyor setting the horizontal plate of a theodolite before making observations. This fixes or establishes the horizontal circle reading until the next time he changes it. Each plate change coincides with a backbearing node.

Performing a new setup is also like changing the plate because the instrument may not be put back exactly the same as a previous setup. Survey Controller will reset the horizontal circle reading to a user value under certain circumstances. This, of course, means a new backbearing node.

In a single backsight setup, this occurs whenever a face 1 observation to the backsight is made. In other setup types, different rules apply.

### Related topics

- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)

- ❑ [View Imported Total Station Data in Project Explorer](#) (on page 241)

## Total Station Data Errors



If a point ID is displayed in red in the *Project Explorer*, there is a computational error associated with the point. For instructions on viewing computational error messages, see [Select from the Flags Pane](#) (on page 48).

The following total station-related computational errors can occur:

Error message	Description
Setup failed to compute	An undefined problem caused all setup computations to fail. No further information is possible.
Using check observation to backsight as fallback	This error occurs when there is only one check observation available to the backsight and it is being used to define the backsight. (Typically, the software does not use check observations for any real computations.)
Azimuth to backsight missing or not computable	This error occurs when the backsight azimuth is not present or some coordinates defining the azimuth are disabled.
Some backsight observations are missing or disabled	This error occurs when some or all backsight observations are either disabled or not present.
Resection computation not possible	This error occurs when numerical problems occurred in the resection mathematics.

### Related topics

- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)
- ❑ [View and Edit Mean Angle Residuals](#) (on page 248)
- ❑ [Run a Mean Angle Report](#) (on page 249)

## View and Edit Mean Angle Residuals





A mean angle represents the combining and averaging of redundant observations to the same point. You can view residuals for any mean angle and, if you want, disable any outlying observations.

**Note:** For mean angle residuals, all angles are normalized to the range of 0 to 360 degrees expressed in project units. All distances are displayed in project units.

**Note:** If you delete a mean angle from your project (that is, right-click the mean angle node in the *Project Explorer* and click *Delete*), all observations associated with the mean angle are also deleted.

### To view and edit residuals for a mean angle:

- To display the *Mean Angle Residuals* dialog, do either of the following:
  - Right-click the mean angle node icon  in *Project Explorer* and select **Mean Angle Residuals** from the context menu.
  - Select the mean angle node icon  in *Project Explorer* and select **Survey > Mean Angle Residuals**.
- In the *Mean Angle Residuals* dialog, review the read-only data displayed in the various boxes in the top half of the dialog.
- In the table, review the observations used to compute the mean angle and their residual values.
- If necessary, uncheck the box in the **Enabled** column for any observation that you do not want used in the computation of the mean angle.

If an unchecked observation has an opposite face observation in the same set, the opposite face observation should also be unchecked to ensure maximum accuracy.

As you enable or disable observations, the means and residuals are re-calculated so you can immediately see the effects of the change. However, these changes are temporary and do not apply to the project until you click **OK**.

- Optionally, click **Report** to view the [Mean Angle Report](#) (see "Run a Mean Angle Report" on page 249).
- When you are done, click **OK**.



**Note:** Mean angles are computed differently in the software than in Survey Controller. In the software, turned angles are displayed with their standard errors. In the Survey Controller, they are displayed with residual and standard errors of backsight and foresight circle readings. This is not a problem, except that it makes comparisons of the two values impossible.

### Related topics


- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)
- ❑ [Run a Mean Angle Report](#) (on page 249)

## Run a Mean Angle Report



Run a *Mean Angle Report* to view details of how each mean angle was computed.

### To run a Mean Angle Report:

- Select **Reports > Mean Angle Report**.
- Right-click a mean angle node icon  in *Project Explorer* and select **Mean Angle Report** from the context menu.
- Click the **Report** button in the *Mean Angle Residuals* dialog.

The *Mean Angle Report* displays in your default Web browser.

**Note:** In the *Mean Angle Report*, all angles are normalized to the range of 0 to 360 degrees expressed in project units. All distances are displayed in project units.

The *Mean Angle Report* includes a separate table for each mean angle in the project. At the top of each table, the point ID, station ID, and backsight ID are displayed. Beneath that, the table includes a row of information for each enabled observation used to compute the mean angle. Information includes observed readings and residual values for the horizontal angle, vertical angle, and slope distance. The last row in the table displays the computed horizontal angle, vertical angle, and slope distance for the mean angle point.

**Tip:** Click any point in the report to select it in the *Project Explorer* and graphical and spreadsheet views, and display its properties in the *Properties* pane.

### Related topics

- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [Workflow for Total Station Data](#) (on page 238)

- ❑ [View Total Station Data in Project Explorer](#) (on page 240)
- ❑ [View and Edit Mean Angle Residuals](#) (on page 248)

## CHAPTER 10

# Work with Level Data







## Understanding Level Data




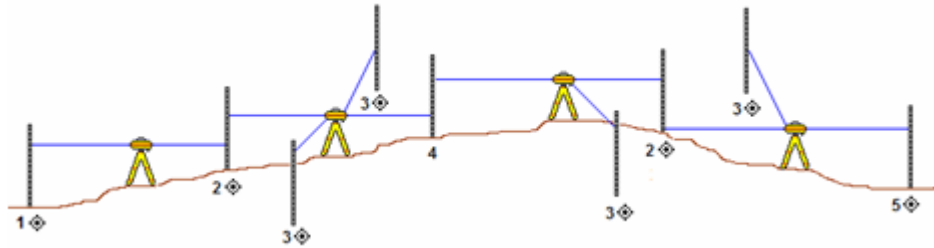
Level data acquired in the field using a Trimble DiNi level and contained in a data file can be imported into the software and integrated, as necessary, with other data collected as part of a survey project (for example, GNSS or total station data). During and after import, you can edit the level data as necessary.

The following diagram shows four segments in a typical level run.

Key to diagram:

-  identifies a control point (point of interest).
- 1  identifies a benchmark control point.
- 2  identifies a control point used for a foresight for the preceding segment and a backsight for the following segment.
- 3  identifies an intermediate control point.
- 4 identifies a turning point that is used as a foresight for the preceding segment and a backsight for the following segment.

- 5  identifies a control point that is used as a foresight for the preceding segment. This could also be a benchmark.



Level data is displayed in the following areas of the software:

- **Project Explorer**  
Imported data is displayed in expandable nodes nested beneath the *Import Files* node. Point data is displayed in expandable nodes located beneath the *Points* node. You can expand and collapse nodes as necessary to view information. (See [View Level Data in Project Explorer](#) (on page 260).)
- **Properties** pane  
Click any node in the *Project Explorer* to display its properties in the *Properties* pane. You can edit certain properties in the pane.
- **Level Editor** dialog  
Right-click the level data file node in the *Project Explorer* pane and select *Level Editor* from the context menu to view and edit level data in the *Level Editor* (see "View and Edit Level Data" on page 255) dialog.
- **Graphical and spreadsheet views**  
Points computed from level data are displayed in graphic (if they include x and y coordinates) and spreadsheet views.

### Related topics

- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View and Edit Level Data](#) (on page 255)
- ❑ [View Level Data in Project Explorer](#) (on page 260)
- ❑ [Adjust Level Runs](#) (on page 265)

## Workflow for Level Data



Use the following steps as a guideline for working with level data:

1. [Import](#) (see "Import DiNi Digital Level Files (.dat)" on page 207) the level data file into your project.

2. Review the imported level data in the **Level Editor** (see "View and Edit Level Data" on page 255) dialog, which displays automatically immediately following import, and make changes as necessary. For example, you can determine which points you want to create for the project, and enable or disable observations.

**Tip:** You can also display the **Level Editor** dialog any time after import by right-clicking the level data file node in the **Project Explorer** pane and selecting **Level Editor** from the context menu.

3. Review the **Import Summary** report to identify any errors, warnings, or messages associated with the import that require corrective action.
4. Review the chronology and correctness of the imported data in the **Project Explorer**.

Expand the nodes under the **Imported Files** node and review in chronological order (top to bottom) to ensure data was gathered correctly and good survey practices were followed. (See [View Level Data in Project Explorer](#) (on page 260).)

If necessary, edit leveling data properties in the **Properties** pane.

If you find an error that cannot be corrected because the **Properties** pane field is read-only, the field crew must re-survey that portion of the project. You can delete the old station nodes that contain errors and import the file that contains the re-surveyed stations. If necessary, you can delete any redundant information.

After level data is imported into your project, you can view the points computed from it in the graphic (if the points include x and y coordinates) and spreadsheet views.

### Related topics


- ❑ [Understanding Level Data](#) (on page 251)
- ❑ [View and Edit Level Data](#) (on page 255)
- ❑ [View Level Data in Project Explorer](#) (on page 260)
- ❑ [Adjust Level Runs](#) (on page 265)

## Import DiNi Digital Level Files (.dat)




DiNi digital level .dat (M5) files contain level data recorded in the field using a Trimble DiNi Digital level.

### To import DiNi Digital level .dat files:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **File > Import**.

The **Import** command pane displays.

2. Do one of the following:
  - Select a folder in the **Import Folder** list.
  - Click the  icon to browse for a folder.

The default is the folder that you last imported from. The files contained in the selected folder appear in the **Select File(s)** area. The file names and file types are listed.

3. Select the DiNi digital level .dat file(s) you want to import.
  1. In the **Settings** section at the bottom of the **Import** command pane, change the **Automatically Numbered Points** properties if necessary.

These properties help the software identify which points were auto-numbered by the DiNi level and which point IDs were entered by the user. Auto-numbered points typically are not points of interest and should not be created as points in the project.

- **Starting Point** - Specifies the first point ID number of the range of point ID numbers in the file you want to specify as automatically numbered points that should not be created in the project.
  - **Maximum Points** - Specifies the maximum number of points in the file you want to be included in the range.
  - **Increment** - Specifies the increment to be used when identifying point numbers in the range.
  - **Ending Point** - Specifies the calculated ending point for the point range based on the other specified properties.
2. When you are done, click **Import**. The **Level Editor** dialog displays. In this dialog, you can see which level points were entered by the user (points of interest) and will be created in the project, and which points were automatically numbered by the DiNi and will not be created in the project. If necessary, you can make changes by checking or unchecking any point to include or not include it in the project. For instructions, see [View and Edit Level Data](#) (on page 255).

**Note:** If you import a text file with "Unknown" or "Mapping" coordinate quality into a project that already contains level point data, duplicate points will be created for points in the text file ([lightweight points](#) (see "Understanding Point Types" on page 269)) and points already in the project ([normal points](#) (see "Understanding Point Types" on page 269)) that have the same ID (that is, points will not merge as expected). To avoid this problem, import the text file first to create the lightweight points in the project, then import the level point data. The lightweight points from the text file will merge with the normal points from the other point data to create normal non-duplicated points. For more information, see [Understanding Point Types](#) (on page 269).

### Related topics

- ❑ [Import Data](#) (on page 193)
- ❑ [View and Edit Level Data](#) (on page 255)

## View and Edit Level Data



The **Level Editor** dialog is displayed automatically immediately after you import level data. You can also display it any time after import by right-clicking the level data file in the **Project Explorer** pane and selecting **Level Editor** from the context menu. The **Level Editor** dialog enables you to do the following:

- View all level readings from the field, and select any reading to be "enabled" (used for computation of level adjustment) or "disabled" (not used for computation of level adjustment).
- Select which points to import.
- Select to use raw or adjusted elevations in the project.
- View the sums of backsight distances and foresight distances.
- View the total misclosure and individual residuals for benchmarks and coordinates.
- View the elevation type for each point (benchmark, computed, or coordinate), including the start and end points, and change the elevation type as necessary.
- Manually enter benchmark heights and qualities.
- Assign an elevation coordinate already in the project that specifies height and quality that cannot be changed.
- Adjust level runs to spread any misclosure proportionately throughout all the measurements
- Merge level runs.
- Specify whether or not to allow a network adjustment to the level data after import (that is, specify whether level data is imported as observations or coordinates).

### Before you get started:

Import level data into the project as described in [Import DiNi Digital Level Files \(.dat\)](#) (on page 207).

**To get started:**

1. When you [import a level data file](#) (see "Import DiNi Digital Level Files (.dat)" on page 207), the **Level Editor** dialog automatically displays. Otherwise, you can display it at any time by doing either of the following:
  - Right-click the imported level data file in the **Project Explorer** and select **Level Editor** from the context menu.
  - Click **Survey > Level Editor** and select the imported level data file in the **Select Leveling Files** dialog.

The **Level Editor** dialog displays a tab for each level run in the data file, allowing you to view and/or edit each run individually.

2. If more than one **Run** tab is displayed, select the tab whose run you want to view or edit.

At the top of the tab, you can view the sum of backsight distances and foresight distances and the total misclosure to identify issues that might require corrective action.

**To select options:**

1. At the top of the tab, select one of the following options:
  - Select **Use Adjusted Elevations** to use elevations in the project that have been adjusted for misclosure. When you select this option, the **Correction** column is displayed in the table instead of the **Misclosure** column. The **Correction** column shows the correction used for the computation of the elevation for each point based on the misclosure value associated with that point.
  - Select **Use Raw Elevations** to use elevations that have not been adjusted. When you select this option, the **Misclosure** column is displayed in the table instead of the **Correction** column. The **Misclosure** column shows the misclosure value for each benchmark or coordinate in the run.
2. If necessary, change the name of the run in the **Run Name** box.
3. In the **Columns** box in the lower left section of the dialog, select the columns you want to display in the table.
4. In the **Creation Options** box in the lower section of the dialog, select one of the following options:
  - **Allow Network Adjustment** – Select this option if you want all elevations of interest imported as delta elevations and, therefore, adjusted as part of a network adjustment.
  - **Prevent Further Adjustment** – Select this option if you want all elevations of interest imported as control coordinates and, therefore, not adjusted as part of a network adjustment.

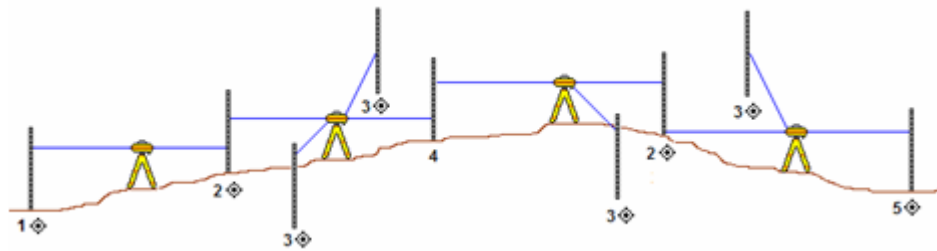


### To work with data in the table:


Each station point in the run is displayed in a table, along with the following information about the point (assuming all **Columns** boxes are checked in the lower left are of the dialog):

- The point ID
  - Backsight, foresight, and/or intermediate rod readings associated with the point
  - The change in elevation for the point based on the rod reading
  - The raw elevation of the point based on the rod reading
  - If the **Use Adjusted Elevation** box is checked, the correction required for the computation of the elevation for the point based on the associated misclosure value
  - If the **Use Raw Elevations** box is checked, the misclosure value for each benchmark or coordinate in the run
  - The adjusted elevation
  - The type of elevation used for the point: benchmark, coordinate, or computed
  - The distance from the instrument to the level rod for each reading
  - If applicable, a description entered for a reading associated with the point
1. Ensure the boxes in the **Create** column are checked appropriately.
    - Points you want imported into the project (for example, control points) should be checked.
    - Points you do not want imported into the project (for example, turning points) should not be checked.

See the following diagram for an example of control points (1, 2, 3, and 5) and a turning point (4).



2. If necessary, edit any point IDs in the **Point ID** column.

If the  icon displays following a point ID, the point ID is common to multiple runs. This can occur when the surveyor intentionally references the same point in multiple runs, in which case, no correction is required. It can also occur unintentionally when point IDs are automatically assigned in the level device for each run. In this case, you might need to rename one or more of the duplicate point IDs with unique point IDs, as necessary.

3. Ensure the enable/disable boxes in the **BS** (backsight), **IS** (intermediate), and **FS** (foresight) columns are checked appropriately.
  - Readings you want included in the computation of a level adjustment should be checked (enabled).
  - Readings you do not want included in the computation of a level adjustment should not be checked (disabled).

**Note:** Readings that were aborted in the field are imported as disabled (not checked).

**Note:** For each point, there must always be at least one backsight reading and one foresight or intermediate reading. Therefore, you can disable a reading for a point only if the point includes additional readings of the same observation type. For example, if a point includes two backsight readings, you can disable one of them, but not both.

4. If necessary, in the **Elevation Type** column change the type of elevation used for any point:
  - **Benchmark** – Select this option to specify that the elevation for the point be a manually entered benchmark elevation. Any point in the run can be designated as a benchmark.
  - **Coordinate** – Select this option to specify that the elevation for the point be a coordinate already assigned to the point in the project. A coordinate can be assigned to a point only if there is a corresponding point already in the project with a coordinate elevation.
  - **Computed** – Select this option to specify that the elevation for the point be computed based on rod readings, benchmark and coordinate elevations entered for the run, and any adjustments performed on the run.
5. If necessary, for any points assigned a **Benchmark** elevation type, do the following:
  - a. Enter or change the value in the **Elevation Type** column.
  - b. Click the **Quality** icon and select the appropriate quality option.
6. If necessary, enter or change any descriptions in the **Description** column.

**To adjust one or more of the level runs contained in the import file:**

When a level run is adjusted, any misclosures are distributed proportionately throughout all the measurements in the run.

**Note:** When multiple intermediate (*IS*) observations exist for the same point ID in a level run, the software includes the intermediate readings in the level adjustment (the *IS* boxes will be checked in the *Level Editor* dialog), just as it is done during a network adjustment. If you do not want these intermediate readings to be included in the level adjustment, either uncheck the *IS* boxes or rename the associated point IDs so they are not the same.

1. Click **Adjust Runs**. The *Adjust Runs* dialog displays.
2. In the *Adjust Runs* dialog, select the runs you want to adjust.  
 These runs will all be adjusted simultaneously as one network. When runs have points in common, the recommended procedure is to adjust them together.
3. Click **OK**.

#### **To merge level runs contained in the import file:**

You can merge two level runs into one run if the last point ID in the first run selected for the merge matches the first point ID in the second run selected for the merge. For example, if the first run selected ends on point "1", the second run selected must start on point "1". This is helpful if two or more level runs were accidentally specified in the field for what should have been a single run (for example, one of the runs does not include a benchmark point). If necessary, you can merge another existing run with the newly created merged run using the same guidelines, and repeat as necessary.

1. Click **Merge Runs**. The *Merge Runs* dialog displays.
2. In the *New Run Name* box, enter the name for the new run.  
 You can enter the name of one of the runs you are merging, or you can enter a new name.
3. In the **Start With** list, select the run whose observations you want to use for the first part of the new run.
4. In the **Add This** list, select the run whose observations you want to use for the second part of the new run.
5. Click **OK**. The *Merge Runs* dialog closes and the newly created run tab replaces the two merged run tabs in the *Level Editor* dialog.

If necessary, you can now merge another existing run with the newly created merged run using the same guidelines and instructions.

#### **To complete viewing and editing:**

1. Repeat the preceding procedures as appropriate for each run you want to view and/or edit.
2. When you are done working in the *Level Editor* dialog, click **OK**.

If the *Level Editor* dialog was displayed as part of a data file import, the import process completes and an *Import Summary* report displays, showing details of the import. Be sure to review any messages, warnings, or errors contained in the report to determine if any corrective action is required.

### Related topics

- ❑ [Understanding Level Data](#) (on page 251)
- ❑ [Workflow for Level Data](#) (on page 252)
- ❑ [View Level Data in Project Explorer](#) (on page 260)

## View Level Data in Project Explorer



Level data is displayed as individual nodes under two top-level nodes in *Project Explorer*:

- **Imported Files** node

This top-level node includes one or more data file nodes, each specifying a file containing GNSS, total station, and/or level field data that has been imported into the project. Individual level nodes display beneath the file node(s) for imported level data. Level nodes are typically listed in the same chronological sequence as data was gathered in the field.

- **Points** node

This top-level node includes a node for each point in the project. Individual level nodes are displayed as appropriate beneath the points to which they pertain.

You can double-click any node to display its properties in the *Properties* pane. You can right-click any node to display a context menu that includes other options, depending on the node type.

For a description of how leveling nodes are displayed in the *Project Explorer*, see one of the following topics:

- [View Imported Level Data in Project Explorer](#) (on page 261)
- [View Level Data Associated with a Point in Project Explorer](#) (on page 263)

### Related topics

- ❑ [Understanding Level Data](#) (on page 251)
- ❑ [Workflow for Level Data](#) (on page 252)
- ❑ [View and Edit Level Data](#) (on page 255)

## View Imported Level Data in Project Explorer



Each imported level data file displays as an expandable node located beneath the **Imported Files** node in **Project Explorer** (select **View > Project Explorer**). The way level data is displayed beneath the level data file node depends on whether the **Allow Network Adjustment** or **Prevent Further Adjustment** option was selected in the **Level Editor** dialog (see [View and Edit Level Data](#) (on page 255)).

For the following examples, assume the file being imported is named "test.dat" and contains one level run. The run began on BM1 and ended on BM2. In addition to the benchmarks, the surveyor specified three control points: 1, 2, and 100. The survey went in the direction of 1 toward 2 with 100 as an IS point between 1 and 2.

### Example 1: *Allow Network Adjustment* option selected in *Level Editor* dialog

This option should have been selected if you want all elevations of interest to be left as delta elevation observations and, therefore, adjusted as part of a network adjustment. The data would display in **Project Explorer** as follows:

#### Node type example

 Imported Files

 Test.dat

  BM1

  BM2

   BM1-BM2 (H1)

   BM1-1 (E1)

#### Description

This is the **Imported Files** node.

This is the imported leveling data file.

The icon is followed by the name of the file.

This is a coordinate node from a benchmark.

The icon is followed by a coordinate point ID.

---

**Note:** To preserve the integrity of raw field data, you cannot edit coordinate properties for an imported coordinate.

---

This is a coordinate node from a benchmark.




This is a run node.

The icon is followed by the run's first coordinate point ID, a dash, and the last coordinate point ID. This is followed by an "H" number in parenthesis that identifies the run.

Observation nodes are nested beneath it in chronological order. The data file can include multiple run nodes.

This is a delta elevation observation node, which represents the change of elevation between two control points.

The icon is followed by the backsight point ID, a dash, and the foresight or intermediate point ID. This is followed by an "E" number in parenthesis









 1-100 (E2)	that identifies the observation.
 1-2 (E3)	This is a delta elevation observation node.
 2-BM2 (E4)	This is a delta elevation observation node.

**Example 2: Prevent Further Adjustment option selected in Level Editor dialog**

This option should have been selected if you want all elevations of interest to be converted from delta elevation observations into coordinates and, therefore, not adjusted as part of a network adjustment. The data would display in *Project Explorer* as follows:

**Node type example**

**Description**

 Imported Files	This is the <i>Imported Files</i> node.
 Test.dat	This is the imported leveling data file. The icon is followed by the name of the file.
 BM1	This is a coordinate node from a benchmark. The icon is followed by a coordinate point ID.
 BM2	This is a coordinate node from a benchmark.
 BM1-BM2 (H1)	This is a run node. The icon is followed by the run's first coordinate point ID, a dash, and the last coordinate point ID. This is followed by an "H" number in parenthesis that identifies the run. Observation nodes are nested beneath it in chronological order. The data file can include multiple run nodes.
 1	This is a coordinate node from an adjustment. The icon is followed by the point ID.
 100	This is a coordinate node from an adjustment.
 2	This is a coordinate node from an adjustment.

**Related topics**

- [Understanding Level Data](#) (on page 251)
- [Workflow for Level Data](#) (on page 252)
- [View and Edit Level Data](#) (on page 255)
- [View Level Data in Project Explorer](#) (on page 260)

- [View Level Data Associated with Point in Project Explorer](#) (see "View Level Data Associated with a Point in Project Explorer" on page 263)

## View Level Data Associated with a Point in Project Explorer



The way imported level data is displayed beneath points in the **Points** node depends on whether the **Allow Network Adjustment** or **Prevent Further Adjustment** option was selected in the **Level Editor** dialog (see [View and Edit Level Data](#) (on page 255)).

### Example 1: **Allow Network Adjustment** option selected in **Level Editor** dialog

This option should have been selected if you want all elevations of interest to be left as delta elevation observations and, therefore, adjusted as part of a network adjustment. The data would display in **Project Explorer** as follows:

#### Node type example

#### Description

 Points

This is the **Points** node.

 1

All point nodes are located beneath it.

This is an individual point node.

The icon is followed by the point ID.

Observation nodes associated with the point are nested beneath it.

If a point ID is displayed in red, there is a computational error associated with the point. For instructions on viewing computational error messages, see [Select from the Flags Pane](#) (on page 48). For a description of level errors that can cause a red point ID, see [Level Data Errors](#) (on page 264).

  BM1-1 (E1)

This is a delta elevation observation node, which represents the change of elevation between two control points associated with point 1.

The icon is followed by the backsight point ID, a dash, and the foresight or intermediate point ID. This is followed by an "E" number in parenthesis that identifies the observation.

  1-100 (E2)

This is a delta elevation observation node.

  1-2 (E3)

This is a delta elevation observation node.

### Example 2: **Prevent Further Adjustment** option selected in **Level Editor** dialog

This option should have been selected if you want all elevations of interest to be converted from delta elevation observations into coordinates and, therefore, not adjusted as part of a network adjustment. The data would display in *Project Explorer* as follows:

### Node type example

 Points

 1

 1

### Description

This is the *Points* node.

All point nodes are nested beneath it.

This is an individual point node.

The icon is followed by the point ID. Observation nodes associated with the point are nested beneath it.

If a point ID is displayed in red, there is a computational error associated with the point. For instructions on viewing computational error messages, see [Select from the Flags Pane](#) (on page 48). For a description of level errors that can cause a red point ID, see [Level Errors](#) (see "Level Data Errors" on page 264).

This is a coordinate node from an adjustment.

The icon is followed by the point ID.

### Related topics

- ❑ [Understanding Level Data](#) (on page 251)
- ❑ [Workflow for Level Data](#) (on page 252)
- ❑ [View and Edit Level Data](#) (on page 255)
- ❑ [View Level Data in Project Explorer](#) (on page 260)
- ❑ [View Imported Level Data in Project Explorer](#) (on page 261)

## Level Data Errors



If a point ID is displayed in red in the *Project Explorer*, there is a computational error associated with the point. For instructions on viewing computational error messages, see [Select from the Flags Pane](#) (on page 48).

The following level-related computational errors can occur:

### Error message

Misclosure out of tolerance

### Description

This error occurs when there is enough redundancy in the leveling run to determine that the elevations computed in the run are out of tolerance. The usual reason for this is a misclosure that exceeds a limit.



Some delta elevation observations are missing or disabled

This error occurs when the elevation for a point cannot be determined because there is no path back to a benchmark. The usual reason is that an observation has been disabled or deleted.

### Related topics

- ❑ [Understanding Level Data](#) (on page 251)
- ❑ [Workflow for Level Data](#) (on page 252)
- ❑ [View and Edit Level Data](#) (on page 255)
- ❑ [View Level Data in Project Explorer](#) (on page 260)

## Adjust Level Runs



A level run can be adjusted during or after import to spread any misclosure proportionately throughout all the measurements in the run. You use the *Level Editor* dialog to adjust level runs.

**Note:** When multiple intermediate (*IS*) observations exist for the same point name in a level run, the software will include the intermediate readings in the level adjustment (the *IS* boxes will be checked in the *Level Editor* dialog). If you do not want these intermediate readings to be included in the level adjustment, either uncheck the *IS* boxes or rename the associated point IDs so they are not the same.

### To adjust level runs:

1. When you import a level data file, the *Level Editor* dialog automatically displays. Otherwise, you can display it at any time by doing either of the following:
  - Right-click the imported level data file in the *Project Explorer* and select **Level Editor** from the context menu.
  - Click **Survey > Level Editor** and select the imported level data file in the *Select Leveling Files* dialog.

The *Level Editor* dialog displays a tab for each run in the data file, allowing you to view and/or edit each run individually.

2. Click **Adjust Runs**. The *Adjust Runs* dialog displays.
3. In the *Adjust Runs* dialog, select the runs you want to adjust.

These runs will all be adjusted simultaneously as one network. When runs have points in common, the recommended procedure is to adjust them together.

4. Click **OK**.

**Note:** Performing an adjustment replaces any previously adjusted elevations, such as those from the instrument.

You can view the adjusted results in the *Level Editor* dialog by selecting the *Use Adjusted Elevations* option in the upper left area of the tab.

5. When you are done working in the *Level Editor* dialog, click **OK**.

For additional instructions on using the *Level Editor* dialog, see [View and Edit Level Data](#) (on page 255).

### Related topics

- ❑ [Understanding Level Data](#) (on page 251)
- ❑ [Workflow for Level Data](#) (on page 252)
- ❑ [View and Edit Level Data](#) (on page 255)
- ❑ [View Level Data in Project Explorer](#) (on page 260)

## Merge Level Runs



You can merge two level runs into one run if the last point ID in the first run selected for the merge matches the first point ID in the second run selected for the merge. For example, if the first run selected ends on point "1", the second run selected must start on point "1". This is helpful if two or more level runs were accidentally specified in the field for what should have been a single run (for example, one of the runs does not include a benchmark point). If necessary, you can merge another existing run with the newly created merged run using the same guidelines, and repeat as necessary.

**Note:** If a level data file you are importing includes a run that does not include a benchmark, see this [note](#) (see "Note on Level Runs Without Benchmarks" on page 267).

### To merge level runs contained in the import file:

1. When you import a level data file, the *Level Editor* dialog automatically displays. Otherwise, you can display it at any time by doing either of the following:
  - Right-click the imported level data file in the *Project Explorer* and select **Level Editor** from the context menu.
  - Click **Survey > Level Editor** and select the imported level data file in the *Select Leveling Files* dialog.

The *Level Editor* dialog displays a tab for each run in the data file, allowing you to view and/or edit each run individually.

2. Click **Merge Runs**. The *Merge Runs* dialog displays.
3. In the **New Run Name** box, enter the name for the new run.  
You can enter the name of one of the runs you are merging, or you can enter a new name.
4. In the **Start With** list, select the run whose observations you want to use for the first part of the new run.
5. In the **Add This** list, select the run whose observations you want to use for the second part of the new run.
6. Click **OK**. The *Merge Runs* dialog closes and the newly created run tab replaces the two merged run tabs in the *Level Editor* dialog.  
If necessary, you can now merge another existing run with the newly created merged run using the same procedure.
7. When you are done working in the *Level Editor* dialog, click **OK**.  
For additional instructions on using the *Level Editor* dialog, see [View and Edit Level Data](#) (on page 255).

### Related topics

- ❑ [Understanding Level Data](#) (on page 251)
- ❑ [Workflow for Level Data](#) (on page 252)
- ❑ [View and Edit Level Data](#) (on page 255)
- ❑ [View Level Data in Project Explorer](#) (on page 260)

## Note on Level Runs Without Benchmarks



If a level data file you are importing includes a level run that does not include a benchmark, the software will attempt to use a computed value for the first point in that run. It does this by locating the **most recent** point ID in a **preceding** run that matches the first point ID in the run that is missing a benchmark. If it finds a matching point ID, it copies the elevation from that point to the first point ID in the run that is missing a benchmark as a computed elevation.

For example, if the first point ID in the run that is missing a benchmark is 100, and the most recent previous instance of a point ID of 100 is found at the end of the preceding run, the elevation for point 100 in the preceding run is copied to the elevation for point 100 in the run that is missing a benchmark as a "Computed" value. In this case, you may not need to merge the runs.

**Related topics**

- ❑ [Understanding Level Data](#) (on page 251)
- ❑ [Workflow for Level Data](#) (on page 252)
- ❑ [View and Edit Level Data](#) (on page 255)
- ❑ [View Level Data in Project Explorer](#) (on page 260)

# CHAPTER 11

## Work with Point Data

### Understanding Point Types

The software supports two basic types of points:

- **Normal points** are created when you import into your project any type of point data other than coordinate point data contained in a text file (for example, .csv) with "Unknown" or "Mapping" quality. You can also create normal points manually using the **Point > Create Point** command.

A normal point includes one or more coordinate and/or observation nodes nested beneath it in *Project Explorer* that are used to compute the coordinate for the point. You can edit values for coordinate nodes, or change the coordinate quality level, in the *Properties* pane. These changes are reflected in the non-editable coordinates displayed for the point in the *Properties* pane.

- **Lightweight points** are created only when you import a text file (for example, .csv) containing coordinate point data with "Unknown" or "Mapping" quality (for example, topographic points).

A lightweight point does not include coordinate nodes nested beneath it in *Project Explorer*. A non-editable coordinate is displayed for the point in the *Properties* pane.

Note the following concerning lightweight points:

- You can add an editable coordinate to a lightweight point by right-clicking the point in *Project Explorer* and selecting **Add Coordinate**. This changes the lightweight point to a normal point.
- If you import a text file with "Unknown" or "Mapping" coordinate quality into a project that already contains point data, duplicate points will be created for points in the text file (lightweight points) and points already in the project (normal points) that have the same ID (that is, points will not merge as expected). To avoid this problem, import the text file first to create the lightweight points in the project, then import the other point data. The lightweight points from the text file will merge with the normal points from the other point data to create normal non-duplicated points.

## Related topics


- ❑ [Create and Edit Points and Coordinates](#) (see "Add and Edit Points and Coordinates" on page 270)

# Add and Edit Points and Coordinates

## Create a Point


Create a point in your project when you need to include one for control or stakeout that was not observed in the field or otherwise recorded and imported in a points file. When you create a point, in addition to the new point object, a coordinate object named 'Office entered' is created and appears in the *Project Explorer*. Each point can only have one office entered coordinate.

### To create a point:


1. Do one of the following:
  - Select **Point > Create Point**.
  - Click the  icon.

The *Create Point* command pane displays.
2. Type a name for the point in the **Point ID** box. Point IDs are **not** case sensitive.
3. Type a code in the **Feature Code** box, if you need to use the point in feature code processing.
4. Select the type of point you want to create in the **Coordinate type** list (see [Point Options](#) (on page 271) for details).
5. Pick a location in the plan view, or type a coordinate in the **Northing** and **Easting** (or **Latitude** and **Longitude**) boxes, or right-click for [coordinate geometry \(COGO\)](#) (see "Understanding COGO Controls" on page 86) options.

**Tip:** Once you have specified northing and easting values, you can click in the **Easting** box and pick a new point in the graphics window to change the easting value, while retaining the northing.


6. Click the  icon and select a planar quality for the coordinate. Qualities are used to determine the best point to use when multiple observations have been made at the same location.
7. Pick an object in a view to use its elevation at the location you pick, or type a value in the **Elevation** box, or right-click for options.

**Note:** Elevation is measured from sea level.

8. Click the  icon and select an elevation quality.

9. If needed, pick an object in a view, or type a value in the *Height* box, or right-click for options. Grid, local, and global coordinates appear at the bottom of the pane so you can check your new point's location in the other types.

**Note:** Height is measured from the geoid.

10. Click the  icon and select a height quality.
11. Select the appropriate point status from the **Status** list. The status determines whether and how the point is used during the computation process.
12. Click **OK**, or click **Apply** to continue creating additional points. The point and an office-entered coordinate appear in the *Project Explorer*, as shown.



**Note:** Only office-entered coordinates can be edited. If you try to edit imported coordinates, an office-entered coordinate object with the new location is created.







### Related topics

- ❑ [Add a Coordinate to a Point](#) (on page 272)
- ❑ [Understanding COGO Controls](#) (on page 86)
- ❑ [Point Options](#) (on page 271)

## Point Options

Use these options to define a new point. They are available in the **Add Point** command pane when you are adding a new point to a project.

## Options

<b>Point ID</b>	Type a unique identifier for the point.
<b>Feature code</b>	Type a code to use in feature code processing.
<b>Coordinate type</b>	<p><b>Grid</b> - Select this to enter northing, easting, elevation, and height values.</p> <p><b>Local</b> - Select this to enter latitude, longitude, height, and elevation values.</p> <p><b>Global</b> - Select this to enter global longitude, height, and elevation values.</p>
 Planar quality	<p>Click this and select a quality for the horizontal coordinate.</p> <p> <b>Control</b> - Select this for NGS surveyed coordinates of the highest quality.</p> <p> <b>Survey</b> - Select this for surveyed coordinates of the second highest quality.</p> <p> <b>Mapping</b> - Select this for coordinates of the low to average quality.</p> <p> <b>Unknown</b> - Select this for coordinates of the lowest or unverified quality.</p>
 Ellipsoidal quality	<p>Click this and select a quality for the vertical coordinate.</p> <p>See the quality descriptions above.</p>
<b>Status</b>	<p><b>Enabled</b> - Select this to include the point in project calculations.</p> <p><b>Disabled</b> - Select this to exclude the point in project calculations.</p> <p><b>Enabled as Check</b> - Select this to exclude the point in project calculations, but to include it in sideshot calculations.</p>

## Related topics

- ❑ [Create a Point](#) (on page 270)
- ❑ [Add a Coordinate to a Point](#) (on page 272)


## Add a Coordinate to a Point

Add a coordinate to a point when you need to include a position for control or stakeout that was not observed in the field or otherwise recorded and imported in a points file. When you add a coordinate to a point, a coordinate object named 'Office entered' is created under the point object.




**Note:** Only add a coordinate when you are sure that you want to use this coordinate instead of an observed coordinate. During the computation of the project, added coordinates are used in preference to observations of the same quality. This ensures that a designed point is used in preference to a staked point.

### To add a coordinate to a point:

- Do one of the following:
  - Pick the point in a graphic view or select it in the *Project Explorer*, right-click and select **Add Coordinate** from the context menu.
  - When a point is selected, and the *Properties* pane is open, click the  icon on the pane's toolbar.



The **Add Coordinate** command pane displays.

- Select the type of coordinate you want to add in the *Coordinate Type* list (see [Coordinate Options](#) (on page 274) for details).

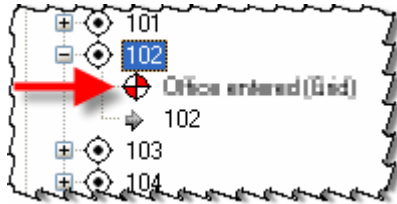
**Note:** You can only enter one grid, local, and global coordinate for each point. If you already have each type, the  icon is unavailable.


- Pick a location in the plan view, or type a coordinate in the *Northing* and *Easting* (or *Latitude* and *Longitude*) boxes, or right-click for [coordinate geometry \(COGO\)](#) (see "Understanding COGO Controls" on page 86) options.

**Tip:** Once you have specified northing and easting values, you can click in the *Easting* box and pick a new point in the graphics window to change the easting value, while retaining the northing.

- Click the  icon and select a planar quality for the coordinate. Qualities are used to determine the best coordinate to use when multiple coordinates exist for the same point.
- Pick an object in a view to use its elevation at the location you pick, or type a coordinate in the *Elevation* box, or right-click for options. Grid, local, and global coordinates appear at the bottom of the pane so you can cross check your new coordinate's location.
- Click the  icon and select an elevation quality.

7. Select the appropriate coordinate status from the drop-down **Status** list. The status determines whether and how the coordinate is used during the computation process.
8. Click **OK**. When you open the point in the **Project Explorer**, it displays the original observation data and the office-entered coordinate, as shown.



9. If the  icon appears on the status bar, click it to recompute and update the coordinates in the project.

**Note:** You can only edit point coordinates that you added in this software, not those which were observed and imported. If you try to edit an imported point, a coordinate object named 'office-entered' is created.

**Note:** You can edit the coordinates for imported *CAD points* in the **Properties** pane, but CAD points are not used in computations.

### Related topics

- ❑ [Create a Point](#) (on page 270)
- ❑ [Compute Project Command](#)
- ❑ [Coordinate Options](#) (on page 274)
- ❑ [Project Explorer](#) (on page 7)
- ❑ [Properties Pane](#) (on page 14)

### Coordinate Options

Use these options to define a coordinate that you are adding to a point. They are available in the **Add Coordinate** command pane.

## Options

### Coordinate type

**Grid** - Select this to enter northing, easting, elevation, and height values.

**Local** - Select this to enter latitude, longitude, height, and elevation values.

**Global** - Select this to enter global longitude, height, and elevation values.



(Planar quality)

Click this and select a quality for the horizontal coordinate.



**Control** - Select this for NGS surveyed coordinates of the highest quality.



**Survey** - Select this for surveyed coordinates of the second highest quality.



**Mapping** - Select this for coordinates of the low to average quality.



**Unknown** - Select this for coordinates of the lowest or unverified quality.



(Ellipsoidal quality)

Click this and select a quality for the vertical coordinate.

See the quality descriptions above.

### Status

**Enabled** - Select this to include the coordinate in project calculations.

**Disabled** - Select this to exclude the coordinate in project calculations.

**Enabled as Check** - Select this to exclude the coordinate in project calculations, but to include it in sideshot calculations.

## Related topics

- [Add a Coordinate to a Point](#) (on page 272)

## Point Coordinate Options

Use these options to edit point coordinates when the point is a member of a surface. The surface that references the edited member updates accordingly. These options are found in the *Properties* pane for a point coordinate.

**Member type****Properties that affect surfaces****Coordinate****Nothing** - Edit this to move a surface vertex north or south.**Easting** - Edit this to move a surface vertex east or west.**Elevation** - Edit to move a surface vertex up or down, relative to sea-level.**Latitude** - Edit this to move a surface vertex north or south.**Longitude** - Edit this to move a surface vertex east or west.**Height** - Edit this to move a surface vertex up or down.**Session****Height** (point) - Edit this to move a surface vertex up or down.**Related topics**

- ❑ [Edit a Surface by Changing a Point Coordinate](#) (on page 329)
- ❑ [Edit a Surface by Adding and Removing Members](#) (on page 328)


**Rename Points**

Rename points to ensure all the points in your project are unique. For example, if two rovers are set to automatic point numbering and contain identical points, use the **Rename Points** command to renumber one set of points. Points to be renamed can be selected before the command starts or within the command.

You can rename your points:

- Sequentially from a specific value
- By adding a prefix
- By adding a suffix

**By adding a constant To rename points:**

1. Select **Point > Rename Points**. The **Rename Points** dialog displays.
2. In the **Method** group, select a renaming option (see descriptions below).
3. If you do not want to generate a report, uncheck **Report**.
4. Select the points to rename.
5. To preview the results, click **Preview** to see the results of the settings before the command is run. Additional preview options are selectable using the  icon.
6. Click **OK** to rename the points. To automatically generate and display a report, check **Report**. When renaming completes, the **Renamed Point** list displays.

## Options


<b>Rename from</b>	Select this if you want to rename points using a specific starting number. Type the starting number in the box.  To use this method, the point ID you enter must end in a number. For example, if you type <i>GPS100</i> and select three points, they will be renamed <i>GPS100</i> , <i>GPS101</i> , and <i>GPS102</i> .
<b>Add prefix</b>	Select this if you want to insert a constant character(s) at the beginning of the point ID for the points you select.
<b>Add suffix</b>	Select this if you want to insert a constant character(s) at the end of the point ID for the points you select.
<b>Add constant</b>	Select this if you want to enter a value to increase each point ID by. For example, if you type 2, point ID <i>GPS101</i> becomes <i>GPS103</i> .
<b>Report</b>	Check this to generate the <b>Renamed Point List</b> report when you finish the command.

## Related topics

- ❑ [Run a Renamed Point List](#) (on page 277)
- ❑ [Merge Duplicate Points](#) (on page 278)
- ❑ [Merge Points](#) (on page 279)
- ❑ [Select Duplicate Points](#) (on page 52)
- ❑ [Select from 2D Views](#) (on page 44)

## Run a Renamed Point List

Generate a **Renamed Point List** to see a simple summary of the original and new names of points that you have renamed in your project.

1. Select **Reports > Report Options**. The **Report Options** command pane displays.
2. Select **Renamed Point List** in the **Reports** list.
3. Edit options in the **Settings** group as needed.
4. Click the  icon on the pane's toolbar. The **Renamed Point List** displays in your default web browser.

## Related topics

- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Edit the Point ID for a Station Setup and/or Backsight



When checking data in your project, you might find that the point ID for a station setup and/or backsight has been incorrectly labeled in the field. For example, the *Plan* view looks incorrect or a red flag indicates a point out of tolerance.

### To change the point ID for a station setup and/or backsight:

1. To change the point ID for a station setup:
  - a. In the *Imported Files* node in the *Project Explorer*, click to display properties in the *Properties* pane for the station setup whose point ID you want to change.
  - b. In the *Point ID* box in the *Properties* pane, type the correct ID for the station setup.

If the corrected ID is for a station you set up on previously or to a control point, no new point is created in the *Points* node (the station point is shared).

Otherwise, a duplicate point is created in the *Points* node (local points are not shared). In this case, merge them as described in [Merge Points](#) (on page 279).

2. To change the point ID for a backsight:
  - a. In the *Imported Files* node in the *Project Explorer*, click to display properties for the observation to the backsight whose point ID you want to change.
  - b. In the *Point ID* box in the *Properties* pane, type the correct ID for the backsight observation. The software updates the backsight with the new point ID.

If the corrected observation is to a station you set up on previously or to a control point, no new point is created in the *Points* node (the station point is shared). Otherwise, a duplicate point is created in the *Points* node (local points are not shared). In this case, merge them as described in [Merge Points](#) (on page 279).

3. Verify your corrections.

## Merge Duplicate Points

If you have multiple points with the same Point ID, you can merge them into a single point.

**Note:** This command can be run with the **Select Duplicate Points** command.

1. Select **Select > Select Duplicate Points**, with a distance tolerance and with **Point IDs identical** selected.

2. Select **Point > Merge Duplicate Points**. A list of the IDs with duplicate points appears.
3. Click **OK**.

**Caution:** Two points with the same ID but significantly different data may be the result of an error. Merging them may produce unexpected results.

### Related topics

- ❑ [Merge Points](#) (on page 279)
- ❑ [Rename Points](#) (on page 276)
- ❑ Select Duplicate Point IDs
- ❑ [Select from the 2D Views](#) (see "Select from 2D Views" on page 44)

## Merge Points


Use this command to create a single point from two or more points that share the same location data.

### To merge points (one set of points):

1. Select the points you want to merge together. Use the **Select Duplicate Points** command or select from the plan view or a spreadsheet view.
2. Select **Point > Merge Points**. If you have not selected points to merge yet, do it now. The points IDs and distance from the first listed point are displayed in the **Selected point** area.
3. Click on the points to merge. A green check displays.
4. (optional) Enter a point ID for the merged point. The default is the first listed point.
5. Click **OK**.

### To merge points (multiple sets of points):

1. Select the points that may be merged together. Use the **Select Duplicate Points** command to select all points that are within a defined distance from one another.
2. Select **Point > Merge Points**. The points IDs and distance from the first listed point are displayed in the **Selected point** area. If you have not selected points to merge yet, do it now. The points IDs and distance from the first listed point are displayed in the **Selected point** area.
3. Click on the points to merge. A green check displays.
4. (optional) Enter a point ID for the merged point. The default is the first listed point.
5. Click **Apply**. The checked points are merged together and removed from the list.

6. Repeat steps 1-5 until all points are merged. Click **OK** when you want to exit the command.
7. Click  if it appears on the status bar to recompute the project.

### Related commands


- Select Duplicate Points
- [Merge Duplicate Points](#) (on page 278)
- [Rename Points](#) (on page 276)
- [Select from the 2D Views](#) (see "Select from 2D Views" on page 44)

## Calculate the Inverse Between Points

Calculate and report inverse values between any two points in your project, such as:

- Grid distance
- Change in elevation

Geodetic azimuth To calculate the inverse between two points:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Survey > Inverse**.

The *Inverse* command pane displays.


2. Select **Sequential** to calculate values from point to point in series (as if drawing a multi-segment line), or **Radial** to calculate values from one point to multiple other points (as if drawing a fan).

**Note:** You can switch between **Sequential** and **Radial** after picking any pair of points.

3. Pick the first point in a graphic view, or type a point ID in the **From** box.



**Note:** You can also right-click in the view to access [COGO controls](#) (see "Understanding COGO Controls" on page 86) and [snaps](#) (see "Snaps Modes and Commands" on page 89) when picking points.

4. Pick another point, or type a point ID in the **To** box. The point IDs appear in the **Reported Points** group, and the inverse values appear in the **Details** group.

**Note:** If **Free** appears in the **Reported Points** list, no point with a point ID was within the pick aperture. To prevent picking where there are no points, click the  icon on the *Inverse* command pane's toolbar. In the **Snap Mode** dialog, uncheck **Free**, and click **OK**.

5. To calculate additional inverses, continue picking **To** points.



6. To review the details for any inverse, click in the first column of the *Reported Points* list.
7. To change inverse report options, click the  icon to display the *Report Options* command pane. When you are done, click **OK** to return to the *Inverse* command.
8. To generate the *Inverse Results* report, click the  icon at the top of the *Reported Points* group. The *Inverse Results* report displays in your default web browser.

**Note:** If no coordinate system is defined, the *Select Coordinate System* dialog displays. Define a coordinate system and run the report again.

9. Click **Close**.

### Related topics

- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)
- ❑ [Inverse Options](#) (on page 231)
- ❑ [Measure Values Between Points](#) (on page 135)

## Inverse Options

Use these options to calculate and report the azimuth, distance, and other relationships between any two points. They are available in the *Inverse* command pane.

## Options




Click this on the **Inverse** command pane's toolbar to display the **Snap Mode** dialog, where you can enable and disable running snap modes.

## Inverse

### From/To

Pick points in graphic views, or type point IDs in the boxes and click **Apply** or press **[Enter]**.

---

**Note:** If **Free** appears in the **Reported Points** list, no point with a point ID was within the pick aperture. To prevent picking where there are no points, click the  icon on the **Inverse** command pane's toolbar. In the **Snap Mode** dialog, uncheck **Free**, and click **OK**.

---

**Sequential** – Data is collected in a line, and you want to verify distances around the traverse.

- For example, you will need to click on:  
A to B, B to C, C to D, D to E, and E to A.

**Radial** – Data is collected in a ray, and you want to check distance from the base station.

- For example, you will need to click on: A to B, A to C, A to D, and A to E.

## Reported points



Click this to display the **Inverse Results** report in your default web browser.



Click this to display the **Report Options** command pan, in which you can specify heading, footer, and format settings for the **Inverse Report**.

### From point ID/

### To point ID

Click in the first column of any row to list details for the inverse of the points.

### Details

This shows the azimuths, changes in elevation and height, and three distances of the selected inverse:

- Grid
- Ellipsoidal
- Ground

### Apply

This acts as the **[Enter]** key, when specifying points, moving the focus between **From**, **To**, and **Reported Points**.

## Related topics

- ❑ [Calculate the Inverse Between Points](#) (on page 230)

# Measure Values Between Points

Calculate and report values between points in your project.

- In the plan view, the command measures bearing and distance.
- In the profile view, it measures the delta station, slope, and slope distance.
- In the cross-section and surface slicer views, it measures delta offset, slope, and slope distance.

### To measure values between two points:

1. Do one of the following:

- Click the  icon on the toolbar.
- Select **Tools > Measure**.

The *Measure* command pane displays.

2. Pick the first point in a graphic view or type a point ID or coordinate (in the format X,Y) in the *From* box.

**Note:** You can also right-click in the view to access [COGO controls](#) (see "Understanding COGO Controls" on page 86) and [snaps](#) (see "Snaps Modes and Commands" on page 89) when picking points.

3. Pick another point or type a point ID or coordinate in the *To* box. The measured values appear in the *Results* group.
4. To measure other values, continue picking *From* and *To* points.

**Note:** You can change to a different graphic view between measurements.

5. Click **Close**.

### Related topics

- ❑ [Measure Options](#) (on page 136)
- ❑ [Calculate the Inverse Between Points](#) (on page 230)
- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Measure Options

Use these options to calculate and report the bearing, distance, slope, slope distance, delta offset, and delta station between any two points, depending on which graphic view you use. They are available in the *Measure* command pane.

## Options

- From/To** Pick points in graphic views, or type point IDs or coordinates (in the format X,Y) in the boxes, and click **Measure** or press **[Enter]**.
- Results** This shows the values between the selected points:  
**Slope** - In the cross-section view, the slope is relative to the centerline. In the surface slicer view, the slope is relative to the first point of the line.  
**Offset** - In the cross-section view, the offset is relative to the centerline. In the surface slicer view, the offset is relative to the first point of the line.
- Measure** When you type in **To** and **From** points, this acts as the **[Enter]** key, calculating the **Results**.

## Related topics

- [Measure Values Between Points](#) (on page 135)

## Points Spreadsheet




The points spreadsheet view lists the survey points in the current project, enabling you to easily edit the data. The plan view and the **Properties** pane reflect all changes made to data in the point spreadsheet view.


### Using the spreadsheet

- **To select a point**, click in the left column for that row.



101	569588.439	949042.647	1550.108 m
101	569588.439	949042.647	1550.108 m
102	569582.094	949037.776	1549.864 m
102	569582.094	949037.776	1549.864 m
102	569582.094	949037.776	1549.864 m
103	569581.904	949046.519	1550.005 m
103	569581.904	949046.519	1550.005 m

- **To display more detail on a point** in the **Properties** pane, select the point and press **[F11]**, or right-click and select **Properties**.
- **To edit a point's ID, coordinate, elevation, or feature code**, select it by clicking on the cell. You can also tab from cell to cell and simply type over the value in the cell.
- **To sort points based on a criteria**, click on a column heading. Up  or down  icons appear on the selected column heading, indicating the current sort order (ascending or descending).
- **To filter the point data**, click on the  icon at the top of the column and select an option from the drop-down menu.

**Note:** If the filter for a column is active, the icon  appears blue.

- **To copy data to a text editor**, such as Microsoft® Notepad, select data, and copy and paste by using the right-click menu or by pressing **[Ctrl] + C** to copy and **[Ctrl] + V** to paste. You can select all data by pressing **[Ctrl] + A**.
- **To change the order of columns** across the spreadsheet, click and drag the column heading to a new location.

### Related topics

- ❑ [Data View Display Formats](#) (on page 32)
- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Select from Spreadsheet Views](#) (on page 46)
- ❑ [Tabbed View Arrangement](#) (on page 34)
- ❑ [Create a Point](#) (on page 270)

## Troubleshoot an Import Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
Duplicate points were created for points in an imported text file and points already in the project that have the same ID (that is, points were not merged as expected).	If you import a text file with "Unknown" or "Mapping" coordinate quality into a project that already contains point data, duplicate points will be created for points in the text file ( <a href="#">lightweight points</a> (see "Understanding Point Types" on page 269)) and points already in the project ( <a href="#">normal points</a> (see "Understanding Point Types" on page 269)) that have the same ID.	Import the text file into the project first to create the lightweight points, then import the other point data. The lightweight points from the text file will merge with the normal points from the other point data to create normal non-duplicated points.

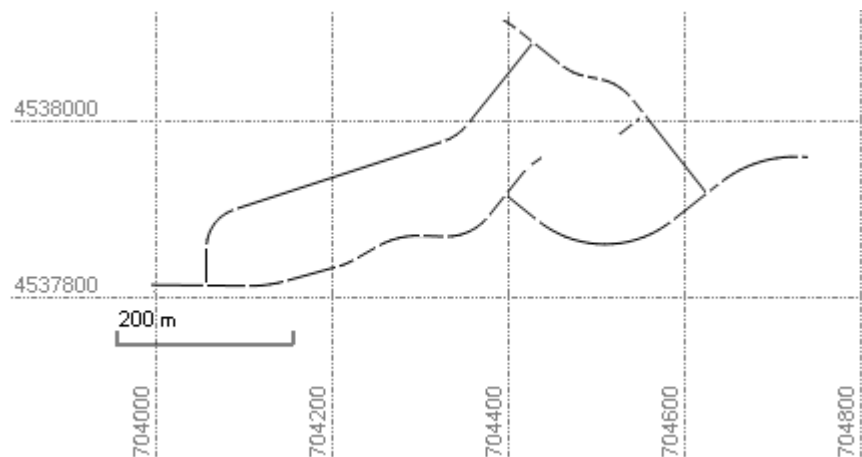
## CHAPTER 12

# Work with Line Data

## Create and Edit an Alignment

### Understanding Alignments

An alignment defines a linear feature, such as a road centerline. Alignments consist of horizontal geometry (a path in a horizontal plane) and optional vertical geometry (a path in a vertical plane). Alignments can also include station equations.



You can import existing alignments from LandXML or GENIO string files, or create them by specifying their horizontal and vertical components, or converting them from vertices, lines, and polylines in your project.

You can create horizontal alignments using lines/tangents, arcs, and clothoid spirals. You can create vertical alignments with lines (constant slope), arcs, and parabolas. Each vertical alignment is associated with a horizontal alignment, and there can be multiple vertical alignments for each horizontal alignment. The values you can enter depend on the alignment settings for the project.

Usually, the goal in creating an alignment is to create a digital file of the design that you can upload to a field device for staking.

### Related topics

- ❑ [Create an Alignment](#) (on page 288)
- ❑ [Workflow for Creating Alignments](#) (on page 287)
- ❑ [Workflow for Using Imported Alignments](#) (on page 287)

## Workflow for Using Imported Alignments

Save the effort of manually creating alignments by importing alignments created in other design applications. Alignments can be imported in GENIO or LandXML file formats.

The general workflow for using imported alignments is:

1. Import the file containing the alignment. The alignment appears as a single object in the *Project Explorer*.
2. Check the alignment to make sure it accurately reflects the design, and make edits as necessary.
  - Open the *Alignment Editor* and verify that coordinates, bearings, lengths, stations, and other values match the original paper or digital plan.
  - Open a new horizontal tab group and arrange your views so that you can see the *Alignment Editor* and the appropriate graphic view (either plan or profile) concurrently.
  - Edit horizontal, vertical, and station values as needed.
  - Open the *Properties* pane, and edit the alignment's properties, such as the name, appearance, and layer, as needed.

### Related Topics

- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Import LandXML Files \(.xml\)](#) (on page 200)
- ❑ [Import GENIO Files](#) (on page 198)

## Workflow for Creating Alignments

Create alignments by manually entering values from digital or paper plans. Here is the general workflow:

1. Use the *Create Alignment* command to create a new, blank alignment, and open it in the *Alignment Editor*.
2. Open a new horizontal tab group and arrange your views so that you can see the alignment spreadsheet and the appropriate graphic view (either plan or profile) concurrently.
3. Enter horizontal values from the paper or digital plans, or using the imported data as a reference. If necessary, enter vertical and station values as well.

4. Open the **Properties** pane, and edit the alignment's appearance properties, as needed.
5. Export the alignment to another application or upload it to a field device. You can export alignments as .dxf/.dwg, GENIO, or LandXML files.

**Note:** You can export an alignment to SCS software as a .dxf/.dwg foreground or background map (loses stationing and spirals), or you can export it to Terramodel as a 3D LandXML file, which you can then upload to SCS software.

**Note:** If you export an alignment as a .dwg and then import it into another application as a .dxf/.dwg, any vertical alignments in the file may not appear.

### Related Topics


- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Create an Alignment from a GENIO String](#) (on page 297)


## Create an Alignment

Build alignments by entering values to define horizontal line, arc, and spiral segments. If needed, define station equations and vertical segments as well.

You can also create alignments by picking points in graphic views, but for precise values it is more likely that you will enter them using the keyboard.

### To create an alignment:

1. Do one of the following:
  - Select **Line > Create Alignment**.
  - Click the  icon on the toolbar.

The **Create Alignment** command pane displays.
2. In the **Name** box, type an identifier for the new alignment.
3. In the **Layer** list, select the layer on which you want the alignment to reside.
4. Click **OK**. The **Alignment Editor** displays.
5. Click the  icon. The **Project Settings** dialog for alignments displays.
6. Change any setting formats to match the formats used in the design.
7. Click **OK**.
8. Referring to your design, type coordinate values in the **North** and **East** cells of the **POB** row. If you have station information, type it in the **Station** box.



**Note:** You can also pick a point, or right-click for snap options, in the graphic views to specify values.

**Note:** You can right-click in the active cell to access *Undo*, *Cut*, *Copy*, and *Paste* commands, or right-click at the beginning of a row to access *Insert Row*, *Delete Row*, *Copy*, *Editor Settings*, and *Float View* commands.

9. Press **[Tab]** or **[Enter]** to proceed to the next row.
10. Select a segment type in the **Type** list. The type you select determines which values you can enter in the other cells of the row.
11. Enter values for the available cells, based on the design.
12. Repeat the previous steps until you have created segments for the entire horizontal component of the alignment.

**Note:** Check the alignment in the graphic views as you enter values.

13. If the alignment has station equation information, click the **Station Equation** tab.

**Note:** Depending on how your data is arranged, you may want to enter station equations in between creating horizontal segments.

14. Click **Create Stations** to make the cells available.
15. Type values in the **Back** and **Ahead** cells.
16. If the alignment has a vertical component, click the **Vertical** tab.
17. Click **Create Profile** to make the cells available.
18. Type coordinates in the **Station** and **Elevation** cells of the **POB** row.
19. Press **[Tab]** to proceed to the next row.
20. Select a segment type in the **Type** list, and enter values in the available cells.

**Note:** In the vertical alignment, you are defining the PVI type, and entering PVI stations and elevations.

21. Press **[Enter]**. The alignment appears in the **Project Explorer**.

### Related topics

- [View Settings](#) (on page 152)

## Use Valid Segment Order

When you are adding, inserting, deleting, or editing horizontal alignment segments, there are valid and invalid ways in which segment types can be connected. For instance, a combining spiral must be preceded and followed by an arc; it cannot connect to any other segment type. Some typical, and valid, sequences of segment types include:

- Residential streets: Line > Arc > Line >
- High speed-streets and ramps: Line > Spiral in > Arc > Spiral out > Line
- Unusual high-speed ramps: Line > Spiral in > Arc > Combining spiral > Arc > Spiral out > Line

### Valid Segment Connections

	To	POB	Line	Arc	Spiral in	Spiral out	Combining spiral
<b>From</b>							
<b>POB</b>		—	OK	OK	OK	OK	—
<b>Line</b>		—	OK	OK	OK	OK	—
<b>Arc</b>		—	OK	OK	OK	OK	OK
<b>Spiral in</b>		—	OK	OK	—	OK	—
<b>Spiral out</b>		—	OK	OK	OK	—	—
<b>Combining spiral</b>		—	—	OK	—	—	—



**Note:** Tangency is assumed when a line transitions into an arc or a spiral.

### Related topics


- ❑ [Alignment Editor](#) (on page 30)
- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Edit an Alignment](#) (on page 290)
- ❑ [Horizontal Alignment Options](#) (on page 291)

## Edit an Alignment

You can edit an alignment's segment type and values, and you can also insert new segments into an alignment.

At the top of the **Alignment Editor**, there is a list of alignments in your project. Click the  icon and the arrow to open the list. Once you have selected the alignment you want to edit, click the  icon to lock the list again. This will prevent you from editing the wrong alignment.

### To edit an alignment:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Line > Alignment Editor**.
  - Pick an alignment in a graphic view, right-click and select **Alignment Editor** from the context menu.
  - Select an alignment in the **Project Explorer**, right-click and select **Alignment Editor** from the context menu.

The *Alignment Editor* displays.

**Note:** You can only edit one alignment at a time. If you have multiple alignments selected, only the active alignment (indicated in bold in the *Project Explorer*) appears in the *Alignment Editor*.

2. Select **Window > New Horizontal Tab Group**. A second pane opens so that the alignment spreadsheet and graphic views are visible concurrently. You may need to right click on a view tab and select **Move to Next Tab Group** from the context menu.
3. In the alignment spreadsheet, click the tab on which you want to edit data.
4. To edit values, click in a cell and specify a new value. The alignment updates in the graphic views as you make changes.

**Note:** If the spreadsheet requests values in different formats than those used in your plan, click **Settings** to access the *Project Settings* dialog, where you can change the display format and entry methods.

5. To change a segment type, click a cell in the *Type* column and select a new type from the list.
6. To insert or delete a segment, right-click at the beginning of a row.
7. Press **[Enter]** to save your changes.

### Related topics

- ❑ [Alignment Editor](#) (on page 30)
- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Edit an Alignment's Properties](#) (on page 295)

## Horizontal Alignment Options

Use these options to define each segment of a horizontal alignment. They are available on the *Horizontal* tab of the *Alignment Editor*.

Entry formats are defined in *Project Settings*, which can be accessed by clicking **Settings** in the *Alignment Editor*, or by right-clicking the last row in the spreadsheet and selecting **Editor Settings** from the context menu. Depending on the entry format that is set, some of the options below may not be required.

## Options


<b>Type</b>	<p><b>POB</b> (Point of Beginning) - Denotes the starting point for the alignment. Define the station, and northing and easting coordinates.</p> <p><b>Line</b> - Select this to enter a straight segment. Define the azimuth or bearing, and length.</p> <p><b>Arc</b> - Select this to enter a curved segment with a constant radius. Define the azimuth or bearing, radius, side from POB, and length/delta/station.</p> <p><b>Spiral In</b> - Select this to enter a transitional segment (clothoid spiral) with a decreasing radius. Generally, use this to connect a line with an arc. Define the azimuth or bearing, radius, side from POB, and length/station/A parameter.</p> <p><b>Spiral Out</b> - Select this to enter a transitional segment (clothoid spiral) with an increasing radius. Generally, use this to connect an arc with a line. Define the azimuth or bearing, radius, side from POB, and length/station or A parameter.</p> <p><b>Combining Spiral</b> - Select this to spiral between two arcs with different radii. Define the azimuth or bearing, side from the POB, radius 1, radius 2, and length/station/A parameter.</p>
<b>Azimuth (North or South) or Bearing</b>	<p>Specify a bearing or an azimuth value at the beginning of the segment, or accept the default of <b>Tangent</b>, which is the bearing from the previous segment.</p> <p>For lines, this is the bearing of the entire segment. For arcs and spirals, it is the tangent bearing into the segment.</p>
<b>Radius</b>	Specify a radius value for an arc.
<b>Left / Right</b>	Select the direction an arc should curve, left of right in the direction from the POB.
<b>Length</b>	Specify the length or distance of the segment. For arcs, and spirals, it is the actual length, not a chord length
<b>Delta</b>	Specify the central angle of the arc.
<b>A Param</b>	Specify the standard factor used as a roadway design criteria in establishing the required length of the spiral. The A parameter reflects the rate of change of the radius, in relation to the distance along the spiral.
<b>Station</b>	Specify the station value at the end of the POB segment. For other segment types, this displays the station.

<b>North</b>	Specify the northing coordinate at the end of the POB segment. For other segment types, this displays the northing coordinate. <hr/> <b>Note:</b> The order of the north and east are dictated by the format set in <i>Project Settings</i> . <hr/>
<b>East</b>	Specify the easting coordinate at the end of the POB segment. For other segment types, this displays the easting coordinate.

### Related topics

- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Horizontal Alignment Options](#) (on page 291)
- ❑ [Vertical Alignment Options](#) (on page 293)

## Vertical Alignment Options

Use these options to define each segment of a vertical alignment. They are available on the *Vertical* tab of the *Alignment Editor*. The entry formats are defined in *Project Settings*, which can be accessed by clicking the  icon in the alignment spreadsheet.

## Options

<b>PI Type</b>	<p><b>POB</b> (Point of Beginning) - Denotes the starting point for the alignment. Define the station and elevation.</p> <p><b>Grade Break</b> - Select this for PVIs that do not have a curve. Typically, this is used for small changes in grade, or for the end of the vertical alignment. Define the station and elevation.</p> <p><b>Symmetrical vertical curve</b> - Select this to create a vertical curve when the curves on either side of the point of intersection (PI) are of <b>equal</b> length. Define the station, elevation, and curve length/K factor.</p> <p><b>Asymmetrical vertical curve</b> - Select this to create a vertical curve when the curves on either side of the point of intersection (PI) are of <b>unequal</b> length. Define the station, elevation, and approach and departure curve lengths.</p> <p><b>Vertical Arc</b> - Select this to enter a vertical curve with a constant radius. Define the station, elevation, and radius/length.</p>
<b>Station</b>	Type a station value, or pick it in the profile view.
<b>Elevation</b>	Type an elevation value for the station, or pick it in the profile view.
<b>Curve Length</b>	Type a length for the vertical curve, or pick two points in the profile view to use the distance between them as the length.
<b>Radius</b>	Type a radius value for the arc.
<b>K Factor</b>	This displays the calculated ratio of change on the vertical curve.
<b>Approach Curve Length</b>	For asymmetric curves, type a value for the curve before the PVI. (from PVC to PVI)
<b>Departure Curve Length</b>	For asymmetric curves, type a value for the curve after the PVI. (from PVI to PVT)

**Note:** You can use the same station value for two consecutive PVIs (not just 0+00). This allows a vertical rise or fall in the alignment.

For example, if you need to model channel (drainage) flow-lines where you have a vertical drop, you can add the same station at different elevations.

Station	Elevation
2+25.00	125.00
2+25.00	120.00

This will draw the profile to elevation 125 when coming from the downstation, and elevation 120 when coming towards the upstation. The order of the entries in the editor determines the order in which they are used.

### Related topics

- ❑ [Alignment Profile View](#) (see "Profile View" on page 26)
- ❑ [Alignment Stationing Options](#) (on page 295)
- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Create a Profile of a Surface](#) (see "Create and View a Surface Profile" on page 325)
- ❑ [Horizontal Alignment Options](#) (on page 291)

## Alignment Stationing Options

Use these options to define an alignment's station values, generally when you add or remove curves. They are available on the **Stationing** tab of the **Alignment Editor**.

### Options

#### Station Equations

**Back** Type a station back value.

**Ahead** Type a station ahead value.

#### Station Zones

**Zone** Denotes the section from one station to the next. The zone number also appears after the colon in the **Start Station** and **End Station** values.

**Start Station** Shows the station at which the zone begins. The first station segment's value is derived from the **Horizontal** tab's POB station.

**End Station** Shows the station at which the zone ends.

**Progression** Indicates whether the station value increases or decreases after the station equation.

### Related topics

- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Horizontal Alignment Options](#) (on page 291)
- ❑ [Vertical Alignment Options](#) (on page 293)

## Edit an Alignment's Properties

Change alignment names, display properties, and layers in the **Properties** pane.

### To edit an alignment's properties:

1. Do one of the following:
  - Pick the alignment in a graphic view, right-click, and select **Properties** from the context menu.

- Double-click the alignment in the *Project Explorer*, or right-click it and select *Properties* from the context menu.

The *Properties* pane displays.

2. Click in an available box, and make changes as necessary.

**Note:** If the scale of your alignment is small in a graphic view, some linestyles display as solid lines to improve performance.

3. Click **Close**. The alignment updates based on the changes.

### Related topics

- [Alignment Editor](#) (on page 30)
- [Edit an Alignment](#) (on page 290)

### Alignment Properties

Use these options to change alignment names, display properties, and layers. They are available in the *Properties* pane when an alignment is selected.



## Options

<b>Linestyle</b>	Select an appearance for the line in graphic views. <hr/> <b>Note:</b> If the scale of your line is small in a graphic view, some linestyles display as solid lines to improve performance. <hr/>
<b>Color</b>	Select a display color for graphic views.
<b>Visible</b>	<b>True</b> - Select this to display the alignment in graphic views. <b>False</b> - Select this to hide the alignment in graphic views.
<b>Layer</b>	Select the layer on which you want the alignment to reside.
<b>Reference Location Northing</b>	Displays the north coordinate used in the alignment's POB.
<b>Reference Location Easting</b>	Displays the east coordinate used in the alignment's POB.
<b>Reference Station</b>	Display the station used in the alignment's POB.

## Related topics

- [Edit an Alignment's Properties](#) (on page 295)

## Create an Alignment from a GENIO String

GENIO strings are sets of 3D points connected to form linear features, such as street centerlines or curb lines. There are three types of GENIO strings that can be imported into Trimble® Business Center. Each type can be used in a different way.

## GENIO string

- 3D** Import these, and use them with other data to form a surface.  
They do not import as alignments, and cannot be converted into alignments.
- 6D** Import these, and convert them into alignments using the steps below.
- 12D** Import these. They are automatically converted into alignments that appear in graphic views and the *Project Explorer*.

### To create an alignment from a 6D string:

1. Import a GENIO 6D string.
2. In a graphic view, pick the string.
3. Select **Line > Create Alignment From a GENIO 6D String**.

The alignment is created. It appears in graphic views (coincident with the original string) and in the *Project Explorer*.

**Note:** After you convert a GENIO 6D string into an alignment, check the results thoroughly.

### Related topics

- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Import GENIO Files](#) (on page 198)


## Run an Alignment Geometry Report

Generate an *Alignment Geometry Report* to see a simple summary or detailed listing of the geometry of an alignment in your project. You can choose to report on just the horizontal component or both the horizontal and vertical components of alignments. If you have specified station equations, they will be reported as well.

### To run an Alignment Geometry Report:

1. Select **Reports > Alignment Geometry Report**. The *Alignment Geometry Report* command pane displays.
2. Select an alignment in the *Alignment* list.
3. Click **OK**. The *Alignment Geometry Report* displays in your default Web browser.

### To customize the report:

1. Do one of the following:
  - Click the  icon on the *Alignment Geometry Report* command pane's toolbar.
  - Select **Reports > Report Options**.

The *Report Options* command pane displays.

2. Select **Alignment Geometry Report** in the list.
3. Expand sections and specify output settings in the **Settings** group as needed.
4. Click **Apply** if you want to customize additional reports, or **OK** to close the command pane.

**Tip:** You can also change the abbreviations used for horizontal and vertical alignment classifications in the *Alignment Geometry Report*. Select **Project > Project Settings**, and click **Abbreviations** in the left pane. Edit any of the abbreviations in the right column and click **OK**. Rerun the report to see your changes.

**Related topics**

- [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- [Customize and Run a Report](#) (see "Customize a Report" on page 436)

**Alignment Editor**

The *Alignment Editor* enables you to edit the horizontal, vertical, and stationing values of existing alignments. The graphic views reflect all changes made to alignments in the editor.

		Type	North Azimuth	Radius	Lt / Rt	Length	Delta	A Param	Station	North	East
Horizontal	POB								-0+048.8	289423.659	392776.747
	Arc	83°18'04"	464.425	Right	29.051	3°35'02"			-0+019.8	289426.143	392805.687
Vertical	Line	80°45'24"			70.321				0+050.49	289437.439	392875.094
	Spiral	63°23'01"	370.000	Left	65.188		155.305		0+115.68	289468.332	392932.472
Stationing	Arc	58°20'10"	370.000	Left	39.451	6°06'33"			0+155.13	289490.790	392964.884
	Spiral	52°13'37"	370.000	Left	65.188		155.305		0+220.32	289533.661	393013.962
	Line	47°10'47"			296.002				0+516.32	289734.854	393231.077
	Spiral	37°49'42"	400.000	Right	60.299		155.305		0+576.62	289781.526	393269.233
	Arc	42°08'49"	400.000	Right	6.879	0°59'07"			0+583.50	289786.586	393273.893
	Spiral	43°07'56"	400.000	Right	60.299		155.305		0+643.80	289828.454	393317.266
	Spiral	47°27'03"	265.000	Left	91.018		155.305		0+734.82	289893.652	393380.605
	Arc	37°36'41"	265.000	Left	79.408	17°10'08"			0+814.23	289962.826	393418.992
	Line	20°26'33"			50.009				0+864.24	290009.686	393436.458
	*										

**Related topics**

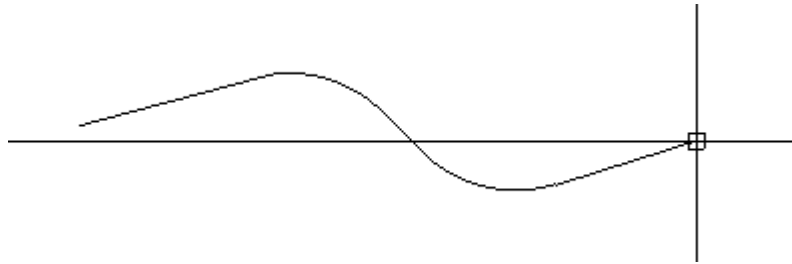
- [Edit an Alignment](#) (on page 290)
- [Data View Display Formats](#) (on page 32)

- ❑ [Pane and Data View Positioning](#) (on page 31)
- ❑ [Tabbed View Arrangement](#) (on page 34)


## Create and Edit a Linestring

### Create a Linestring

Create linestrings (versatile single or multi-segmented linear or curvilinear objects) to represent 2D or 3D linear objects. 3D lines can be defined and queried entirely from within the *Plan View* (on page 25), and they provide a unique and versatile way of establishing the line's elevation or its vertical alignment. You can create linestrings in this software, or you can convert imported lines, such as CAD polylines, into linestrings as you edit them. In creating a linestring, you specify the location of each point along the line, and how the connections (segments) between them are formed.



#### To create a linestring:

1. Do one of the following:
  - Select **Line > Create Linestring**.
  - Click the  icon.

The *Create Linestring* command pane displays.
2. In the **Name** box, type an identifier for the linestring as you want it to appear in the *Selection Explorer* and graphic views. You can also use the name to select the linestring in the *Advanced Select* command.
3. Select the layer on which you want the linestring to reside in the **Layer** list, or select **<<New layer>>** to create a new layer for the linestring.
4. Click **OK**. The *Edit Linestring* command pane displays. Continue to create the linestring using either of the options below.

#### To enter the first point by specifying a coordinate:

This option starts the linestring with the coordinate of any location (point) you pick, so that the linestring's geometry remains fixed at that point until you edit its coordinate.

1. In the **Start Point** group's **Type** list, select **Coordinate**.

2. Click in the **Coordinate** box, and pick a location in a graphic view, type a coordinate, or right-click for [COGO coordinate options](#) (see "Enter a Coordinate" on page 97).
3. Enter the elevation for the starting point in the **Elevation** box, or right-click for [COGO elevation options](#) (see "Enter an Elevation" on page 125).
4. Click **Save** or press **[Enter]**. The first point of the linestring is saved and you are prompted to add the second point, designating how the segment between them is formed.
5. Add the second point and the segment between them using the operations in *Edit a Linestring's Horizontal Segments* (on page 301).

### To enter the first point by specifying a point ID:

This option starts the linestring with a named point (point with a point ID) so that the linestring's geometry always reflects the location of the point. If you edit the point, the linestring updates in response.

1. In the **Start Point** group's **Type** list, select **Point ID**.
2. Click in the **Point ID** box, and pick a named point in a graphic view or type a point ID. The elevation of the point you specified is automatically used.
3. Click **Save** or press **[Enter]**. The first point of the linestring is saved and you are prompted to add the second point, designating how the segment between them is formed.
4. Add the second point and the segment between them using the operations in *Edit a Linestring's Horizontal Components* (see "Edit a Linestring's Horizontal Segments" on page 301).

### Related topics

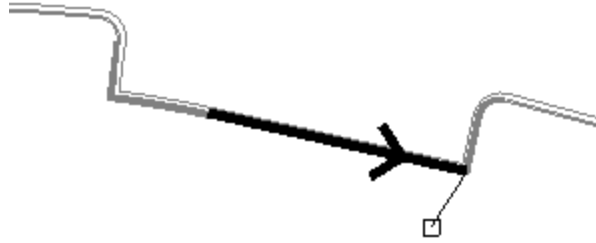
- ❑ [Edit a Linestring's Horizontal Segments](#) (on page 301)

## Edit a Linestring's Horizontal Segments

The linestring editing command provides one familiar tool with which you can edit a large array of imported CAD lines of many types, such as polylines, arcs, and splines, automatically converting them in the process to linestrings, and enriching them as needed with geometric attributes not otherwise supported by the source objects.

Edit linestrings by redefining their points and the connections (segments) between the points, as well as by adding, inserting, and deleting segments.


To define vertical points of intersection (independent of the segment end points) at specific distances along the linestring, add vertical control points using the operations in *Edit a Linestring's Vertical Control Points* (on page 309).



The current segment, end point, and direction of the linestring are indicated during editing. An X indicates the linestring's start point.

### To access these options:

If you have just created a linestring, skip to one of the operations below. Otherwise, select a line or linestring and do one of the following:

- Select **Line > Edit Linestring**.
- Click the  icon on the toolbar, or on the *Properties* pane toolbar when a linestring is selected.
- Right-click, and select **Edit** from the context menu.

The *Edit Linestring* command pane displays. Click the *Horizontal* tab.

### To view and select each horizontal segment in sequence:

- Click the back and forward arrows on either side of the *Current segment* list, or move your cursor along the linestring in a graphic view and click to select the segment.

### To add a straight segment:

1. In the *Segment* group's *Type* box, select **Straight**.
2. Follow the steps in *To specify a segment's end (or start) point* below.
3. Click **Save** or press **[Enter]**. The segment is saved and you are prompted to specify the end point for the next segment.

### To add a curved segment:

1. In the *Segment* group's *Type* list, select **Arc** or **3 Point Arc**.
  - **Arc** - Select this to create a curved segment between two points. Then, specify the radius of the arc, whether it curves left or right from the previous segment, and whether to use the larger arc (> 180 °) or smaller arc (< 180 °) that is created between the points.

- **3 Point Arc** - Select this to create a curved segment from the preceding endpoint, passing through a specified intermediate point, to a specified segment endpoint.
2. Follow the steps in *To specify a segment's end (or start) point* to specify the end point of the curved segment.

#### To add a straight segment at a deflection angle from the preceding segment:

1. In the **Segment** group's **Type** list, select **Deflection**.
2. For the **Direction**, select the perpendicular left (-90 °), perpendicular right (90 °), straight ahead (0 °) deflection angle option, or select **Specified angle** to enter a specific deflection angle.


**Note:** Positive deflection angles are measured clockwise from the direction of the previous segment.

3. Pick a point in the graphic view to specify the length, or type a value in the **Length** box.
4. Type an elevation for the end point in the **Elevation** box, and click **Save** or press **[Enter]**.



#### To specify a segment's end (or start) point:

1. In the **End Point** (or **Start Point**) group's **Type** list, select **Coordinate** or **Point ID**.
2. Depending on the type you selected, click in either the **Coordinate** box or the **Point ID** box.
3. For a coordinate, pick a location in a graphic view, type a coordinate, or right-click for [COGO coordinate options](#) (see "Enter a Coordinate" on page 97). Then enter the elevation for the starting point in the **Elevation** box, or right-click for [COGO elevation options](#) (see "Enter an Elevation" on page 125).
4. For a point ID, pick a named point (point with an ID) in a graphic view or type a point ID. The elevation of the point you specified (if defined) is automatically used to establish a vertical point of intersection on the line at that point.
5. Click **Save**.

#### To edit a segment:



1. Select the segment by picking it in the plan view, selecting it in the **Current segment** list, or clicking  and selecting it in the **Browse Horizontal Segments** list.
2. Modify any of the options for the segment, and click **Save**.

**To remove a segment:**


1. Select the segment by picking it in the plan view, selecting it in the *Current segment* list, or clicking  and selecting it in the *Browse Horizontal Segments* list.
2. Click the  icon. The segment and its end point are removed, and the adjoining segments' end points are joined.

**Note:** To delete a segment without joining the adjoining segments, use the *Delete Line Segment* (see "Delete a Line Segment" on page 316) command.


**To insert a segment before the current segment:**

1. Select the segment before which you want to insert a new segment by picking it in the plan view, selecting it in the *Current segment* list, or clicking  and selecting it in the *Browse Horizontal Segments* list.
2. Click the  icon.
3. Follow the steps in *To specify a segment's end (or start) point* above to specify the new segment's end point, which becomes the new location of the start point for the segment you selected. The new segment is inserted before the selected segment.


**To insert a new first segment:**

1. Select the linestrings 'start' coordinate or point by picking it in the plan view or selecting it in the *Current segment* list.
2. Click the  icon.
3. Follow the steps in *To specify a segment's end (or start) point* above to specify the start point for the new, first segment in the linestring.

**To add a segment onto the end:**


1. Select the last segment by picking it in the plan view or selecting it in the *Current segment* list.
2. Click the  icon.
3. Follow the steps in one of the *To add a \* segment* operations above.

**To view and select a segment in a list:**



1. Click the  icon. The *Browse Horizontal Segments* dialog displays.
2. Select the segment you need, and click **OK**.



### To reverse the segment order/switch the linestring's start and end points:



- Click the  icon on the pane's toolbar. You can confirm the order by moving your cursor along the linestring in the graphic view. The direction of the linestring is shown in the plan view an arrow on the selected segment.

### To convert two consecutive straight segments to a 3 point arc segment:

1. Select the segment whose end point will become the 'point on curve' by picking it in the plan view, selecting it in the *Current segment* list, or clicking  and selecting it in the *Browse Horizontal Segments* list.
2. Click the  icon. The selected segment and the one following it are converted to a 3 point arc.

**Note:** Since you must select the first of two segments to be used in the 3 point arc, a linestring's final segment cannot be selected for this function.

### To convert a 3 point arc segment to two straight segments:

1. Select the segment that is currently defined as a 3 point arc by picking it in the plan view, selecting it in the *Current segment* list, or clicking  and selecting it in the *Browse Horizontal Segments* list.
2. Click the  icon. The selected segment is converted into two straight segments, joined at the previous 'point on curve' of the 3 point arc.

### Related topics

- ❑ [Create a Linestring](#) (on page 300)
- ❑ [Edit a Linestring's Vertical Control Points](#) (on page 309)
- ❑ [Horizontal Linestring Segment Options](#) (on page 305)

## Horizontal Linestring Segment Options

Use these options to create and edit horizontal linestring segments. They are available on the *Horizontal* tab of the *Edit Linestring* command pane. Linestring properties can be edited in the *Properties pane* (on page 14).

## Options



Click this to display the properties of the selected linestring in the **Properties** pane.



Click this to start the **Create Linestring** (see "Create a Linestring" on page 300) command.



Click this to reverse the order of the linestring's segments and the start and end points.



Click this to start the **Set Line Elevation** (see "Set a Line Elevation" on page 317) command.



Click this to start the **Break Line** (see "Break a Line" on page 314) command.



Click this to start the **Join Lines** (on page 315) command.

## Horizontal tab

### Current segment

This shows the number, segment type, and end point type of the currently selected horizontal segment of the linestring. Click the back or forward arrow on either side of the listed segment (or scroll the mouse wheel) to select the previous or next segment.

A segment is defined by the location of its end point, the elevation of the point (if defined), and the type of connection to the previous segment's end point (or the linestring's starting point).



Click this to display the **Browse Horizontal Segments** dialog, a list in which you can view and select from all of the horizontal alignment segments in the linestring.



Click this to insert a new segment before the selected segment or start point.



Click this to add a new segment onto the end of the linestring.



Click this to remove the selected segment.

The segment and its end point are removed, and the adjoining segments' end points are joined.



When displayed, click the icon to toggle between two states, depending on the current segment type.

Click to convert a selected 3 point arc segment into two straight segments, converting the specified **point on curve** into a new segment's end point.

Click to convert the selected segment and next segment into a 3 point arc, converting the end point of the first segment into the arc's **point on curve**.

### Start point

This group appears only when you are specifying the beginning location of a linestring. Use the **Straight Segment Options** below.

### Segment type

Select an option for how the current segment end point will be joined to the previous segment end point or the line's start point.

**Straight** - Select this to create a straight segment between the points.

**Arc** - Select this to create a curved segment defined by a radius between the points.

**3 Point Arc** - Select this to create a curved segment defined by a *point on curve* and an end point.

**Deflection** - Select this to add a straight segment at a deflection angle from the preceding segment.

**Straight segment/End point options**

<b>End point type</b>	<b>Coordinate</b> - Select this to end the segment by specifying a fixed coordinate location. <b>Point ID</b> - Select this to end the segment by specifying a named point using a point ID. The linestring segment will be dynamically attached to that point object.
<b>Coordinate</b>	Pick a point in a graphic view, type a coordinate, or right-click for more options
<b>Elevation</b>	Optionally, type an elevation for the end point of the segment. If you do, it establishes a vertical point of intersection on the line at the segment's end point.
<b>Point ID</b>	Pick a named point in a graphic view or type a point ID.

**Arc segment options**

<b>Radius</b>	Type a value for the radius of the arc.
<b>Left/right</b>	Select whether the arc should curve right or left, in relation to the direction of the previous segment.
<b>Large/Small</b>	Select whether to use the larger arc ( $> 180^\circ$ ) or smaller arc ( $< 180^\circ$ ) that is created between the points.

**3 Point arc segment options**

<b>Coordinate</b>	Select this to use any coordinate as the intermediate point ('point on curve') of the arc.
<b>Point on curve</b>	Pick a point in a graphic view, type a coordinate, or right-click for more options to specify a point through which the arc must pass.
<b>Elevation</b>	Optionally, type an elevation for the <b>point on curve</b> of the arc. If you do, it establishes a vertical point of intersection on the line at the segment's end point.
<b>Point ID</b>	Select this to use a named point as the intermediate point. Then, pick a named point in a graphic view or type a point ID in the box.

**Deflection segment options**

<b>Direction</b>	Select the perpendicular left ( $-90^\circ$ ), perpendicular right ( $90^\circ$ ), straight ahead ( $0^\circ$ ) deflection angle option, or select <b>Specified angle</b> to enter a specific deflection angle. <hr/> <b>Note:</b> Positive deflection angles are measured clockwise from the direction of the previous segment. <hr/>
<b>Length</b>	Type a value in the box or pick a point in the graphic view to specify the length of the segment.
<b>Elevation</b>	Optionally, type an elevation for the end point of the segment. If you do, it establishes a vertical point of intersection on the line at the segment's end point.
<b>Save</b>	Click this to save the current segment. You are prompted to specify the end point for the next segment.

## Related topics

- ❑ [Edit a Linestring's Horizontal Segments](#) (on page 301)
- ❑ [Edit a Linestring's Vertical Control Points](#) (on page 309)
- ❑ [Vertical Linestring Options](#) (on page 310)


## Edit a Linestring's Vertical Control Points

If you need to define vertical points of intersection (VPIs) on a linestring that are located other than at the horizontal segment end points or the line's start point, add vertical control points along the linestring. A vertical control point is defined by its distance along the linestring, the elevation of the vertical point of intersection, and an indication as to whether a symmetrical parabolic vertical curve is involved at that VPI and if so, its length. Vertical points of intersection defined in this manner have no effect on the linestring's horizontal alignment.

**Note:** If a horizontal segment's end point and a vertical control point coincide, the elevation of the vertical control point overrides the elevation that may be assigned to the end point or to the point object to which it may be attached.


### To access these options:

Select a line or linestring and do one of the following:


- Select **Line > Edit Linestring**.
- Click the  icon on the toolbar, or on the *Properties* pane toolbar when a linestring is selected.
- Right-click, and select **Edit** from the context menu.

The *Edit Linestring* command pane displays. Click the **Vertical** tab.

### To add a vertical control point with no vertical curve:


1. Click the  icon.
2. In the *Curve* group's *Type* box, select **No curve**.
3. Type a distance from the linestring's start point in the *Distance along* box, or pick a point along the linestring in the graphic view.
4. Type an elevation for the vertical control point in the *Elevation* box.
5. Click **Save** or press **[Enter]**. The vertical control point is added with simple, straight transitions from and to the point using the straight segments created by the point.

### To add a vertical control point with a parabolic curve:

1. Click the  icon.

2. In the **Curve** group's **Type** box, select **Parabolic**.
3. In the **Length** box, type a length for the parabolic vertical curve to be applied at the vertical control point.
4. Type a distance from the start point in the **Distance along** box, or pick a point along the linestring in the graphics view.
5. Type an elevation for the vertical control point in the **Elevation** box, and click **Save** or press **[Enter]**. The vertical control point is added with a parabolic curve.


#### To view and select a vertical control point in a list:

1. Click the  icon. The **Browse Vertical Information** dialog displays.
2. Select the point you need, and click **OK**.



#### To step through each vertical control point in sequence:

- Click the back and forward arrows on either side of the **Current vertical control point** list.

#### To edit a vertical control point:

1. Select the point by selecting it in the **Current VPI** list or clicking  and selecting it in the **Browse Vertical Information** list.
2. Modify any of the points options, and click **Save**.

#### To delete a vertical control point:

1. Select the point by selecting it in the **Current VPI** list or clicking  and selecting it in the **Browse Vertical Information** list.
2. Click the  icon. The point is removed, eliminating the affect of the vertical control point on the linestring's vertical alignment.

#### Related topics

- ❑ [Edit a Linestring's Horizontal Segments](#) (on page 301)
- ❑ [Vertical Linestring Control Point Options](#) (see "Vertical Linestring Options" on page 310)

### Vertical Linestring Options

Use these options to create and edit vertical control points along a linestring. They are available on the **Vertical** tab of the **Edit Linestring** command pane. Linestring properties can be edited in the **Properties pane** (on page 14).

## Vertical tab options

### Current VPI

This shows the distance along the linestring and curve type of the currently selected vertical control point on the linestring. Click the back or forward arrow on either side of the listed point to select the previous or next point.

A vertical control point is defined by its distance along the linestring, the elevation of the point, and how the linestring's geometry vertically transitions at the point.

---

**Note:** The locations of vertical control points are independent of the locations and elevations of horizontal segment end points; when you move horizontal segments, vertical control points remain at the distance along the line at which they were defined. If a horizontal segment's end point and a vertical control point coincide, the elevation of the vertical control point overrides the elevation of the end point.

---



Click this to display the **Browse Vertical Information** dialog, a list in which you can view and select from all of the vertical control points in the linestring.



Click this to add a new a vertical control point.



Click this to remove the selected vertical control point.

### Curve type

Select an option for how the linestring's geometry vertically transitions at the vertical control point.

**No curve** - Select this to simply transition at the vertical control point using the two straight segments created by the point.

**Parabolic** - Select this to transition more smoothly by rounding off the two straight segments created by the point using a parabolic curve.

### No curve type

#### Distance along

Type a distance from the linestring's start point, or pick a point along the linestring in the graphic view.

#### Elevation

Type an elevation for the vertical control point.

### Parabolic curve type

#### Length

Type a length for the parabolic curve that transitions the two straight segments into the vertical control point.

The parabolic curve is tangent to both incoming and outgoing segments.

#### Save

Click this to save the current vertical control point.

You are prompted to specify the location of the next vertical control point.

## Related topics

- [Edit a Linestring's Horizontal Segments](#) (on page 301)


- ❑ [Edit a Linestring's Vertical Components](#) (see "Edit a Linestring's Vertical Control Points" on page 309)
- ❑ [Horizontal Linestring Options](#) (see "Horizontal Linestring Segment Options" on page 305)

## Create and Edit a Simple Breakline

Simple breaklines are simplified linestrings that help to define the shape of a surface as do other 3D line types that act as breaklines, by controlling how triangles in the surface mesh are formed. Triangles forming a surface never cross a breakline. Use breaklines of any type to more accurately reflect where surface topography changes.

Create a simple breakline is to add linestring segments between points where surface triangles converge (vertices), but you can also add simple breakline segments freely between any two locations on or near the surface. The addition, deletion or any edit to a 3D line serving as a surface breakline will cause the entire surface to be reformed. Unlike more complex linestrings, simple breaklines do not use vertical control points; you can enter elevations for each new vertex or derive elevations from existing vertices.

### To create a simple breakline:

1. Do one of the following:
  - Select **Surface > Create Breakline**.
  - Click the  icon on the toolbar.

The *Create Breakline* command pane displays.
2. In the **Name** box, type an identifier for the linestring as you want it to appear in the *Selection Explorer* and in graphic views when you select from multiple objects. You can also use the name to select the linestring in the *Advanced Select* command.
3. Select a layer on which to create the linestring in the **Layer** list, or select **<<New layer>>** to create a new layer for the linestring.
4. Select an option for how smoothly the surface is rendered at this particular linestring in the **Surface sharpness** list.
5. Select the surface to which the linestring will be assigned as a member in the **Add to surface** list.
6. Click **OK**. The *Edit Linestring* command pane displays with reduced options appropriate for creating a simple breakline.
7. In the **Start Point** group's **Type** list, select **Coordinate** to start the linestring at any location, or select **Point ID** to attach the linestring to a named point.



8. Click in either the **Coordinate** box or **Point ID** box (depending on what you chose for the starting point type), and use one of these ways to specify each point along the linestring:
  - In a graphic view, pick a vertex on the surface, or pick a named point (which may be a member of the surface and therefore one of its vertices)
  - Pick any location on or near the surface in the graphic view, and then type an elevation in the **Elevation** box. Then, click **Save** or press **[Enter]**.
  - Type a coordinate, press **[Tab]**, and type an elevation in the **Elevation** box. Click **Save** or press **[Enter]**.

The first point of the simple breakline is saved and you are prompted to add the second point and specify how the segment between them is formed.

**Note:** To use the elevation from a point you pick anywhere on a surface, right-click and select **From Surface Snap** from the context menu. You are prompted to pick the point in the **From Surface Snap** command pane. Then, click **Save** or press **[Enter]**.

**Note:** To avoid picking points off of the surface (with no elevation), uncheck the **Free** snap in the **Snap Mode** (see "Snaps Modes and Commands" on page 89) dialog when you create a simple breakline or linestring. If you are prompted for an elevation, you may have missed the vertex point you were trying to pick.

**Note:** You can also draw a 'freehand' breakline without using any existing data. In fact, you do not even need to have a surface to create a breakline.

9. Keep adding segments to the linestring using one of the methods in step 8.
10. Click **Close**. The simple breakline appears on the surface in graphic views, but does not appear in the **Project Explorer**.

**Note:** Once created, linestrings can be graphically selected and added to other surfaces.

**Tip:** To view changes to your surface more clearly as you work, open a 3D view and move it to a new tab group next to your plan view so you can see both views concurrently.

#### To edit a simple breakline:

- Use the operations in **Edit a Linestring's Horizontal Segments** (on page 301).

#### Related topics

- ❑ [Edit a Linestring's Horizontal Segments](#) (on page 301)
- ❑ [Create a Linestring](#) (on page 300)
- ❑ [Understanding Breaklines](#) (on page 330)

## Breakline Options

Use these options and those listed in *Horizontal Linestring Options* to create a breakline. They are available in the **Create Breakline** and **Edit Linestring** command panes. After creating a breakline, these properties and others can be edited in the **Properties** pane.

### Options

<b>Name</b>	Type an identifier for the breakline.  This name will appear in the <b>Selection Explorer</b> and in graphic views when you select from multiple objects. In addition, you can use the name to select the breakline using the <b>Advanced Select</b> command.
<b>Layer</b>	Select the layer on which you want the breakline to reside, or select << <b>New layer</b> >> to create a new layer for the breakline.  Segregating data onto logical layers makes it easier to filter your graphic views and select related data.
<b>Surface sharpness</b>	Select an option for how smoothly surface colors graduate at breaklines.  <b>Soft</b> - Select this to have shading gradually change at the breakline, giving the visual appearance of a soft edge. <b>Sharp</b> - Select this to have shading abruptly change at the breakline, giving the visual appearance of a sharp edge.
<b>Add to surface</b>	Select the surface that you want the breakline to modify.

### Related topics


- [Create and Edit a Breakline](#) (see "Create and Edit a Simple Breakline" on page 312)
- [Edit a Surface by Creating a Breakline](#) (on page 334)

## Break a Line

Break a line to split it into two separate linestrings at the point you specify. The linestrings remain coincidental at the break point, but must be edited separately thereafter.

**Note:** This command and others used on lines, such as **Join**, **Edit**, and **Delete Line Segment**, convert the lines into linestrings. Linestrings are generic lines which offer more editable properties than the original lines. For more information on linestrings, see *Understanding Linestrings*.

### To break a line:

1. Do one of the following:
  - Select **Line > Break Line**.
  - Click the  icon on the toolbar.

The **Break Line** command pane displays.

2. Pick a line in a graphic view. The line is converted into a linestring which is recorded in the **Line** box, and a "rubber-band" line appears between your cursor and the linestring.
3. Using the rubber-band line, pick a location along the linestring at which to break it, or distance from the beginning point of the line in the **Location** box and click **Break**. The linestring is broken at the point you specified.
4. To break another line, click in the **Line** box, or click **Close** to end the command.

### Related topics

- ❑ [Join Lines](#) (on page 315)
- ❑ [Create a Linestring](#) (on page 300)
- ❑ [Edit a Linestring's Horizontal Segments](#) (on page 301)

## Join Lines

Join two or more contiguous lines into individual linestrings using manual and automated methods. When lines are joined, they are converted into linestrings, thereby increasing their versatility. The resulting linestring's direction and properties are determined by the first (base) line that you choose.


There are two ways lines join:

- By making their end points coincident if they are within 0.0001 m of each other
- By drawing a straight segment between the nearest end points if they are further than 0.0001 m apart

**Note:** End points with different elevations cannot be joined.

**Note:** This command works on 2D data in the plan view; end points with an elevation of 0 (zero) are converted to 2D end points.

### To join two lines manually:

1. Do one of the following:
  - Select **Line > Join Lines**.
  - Click the  icon on the toolbar.

The **Join Lines** command pane displays.
2. Select the **Two lines** option.
3. Pick a line in a graphic view. The line's name is recorded in the **Base line** box.
4. Pick the line you want to join to the base line. The line's name is recorded in the **Line to join** box. The lines are joined and converted into a single linestring.
5. To join additional pairs of lines, continue to pick lines to connect to the current base line.

6. Continue to join additional lines, or click **Close**.

#### To join one line to many consecutive lines:

1. After step 1 above, select the **One line to selection** option.
2. Pick a line in a graphic view. The line's name is recorded in the **Base line** box.
3. Pick a set of lines to join to the base line, or click **Options** and choose a selection option. The number of lines selected is recorded in the **Lines to join** box.
4. Press **[Enter]** or click **Join**. The command searches from the base line to each consecutive line, joining them until it reaches a break in the lines or a branch to multiple lines where it stops. The number of lines that were successfully joined to the base line is reported at the bottom of the pane.

**Note:** Lines that are not within the 0.0001 m tolerance of each other cannot be joined using this method.

#### To join all consecutive lines:

1. After step 1 in the first procedure, select the **All selected lines** option.
2. Pick a set of lines to join in a graphic view, or click **Options** and choose a selection option. The number of lines selected is recorded in the **Lines to join** box.
3. Press **[Enter]** or click **Join**. The command searches from each selected line to each consecutive line, joining them until it reaches a break in the lines or a branch to multiple lines where it stops. The number of lines that were successfully joined to the base line is reported at the bottom of the pane.

**Note:** Lines that are not within the 0.0001 m tolerance of each other cannot be joined using this method.

#### Related topics

- [Break a Line](#) (on page 314)

## Delete a Line Segment

Delete a segment from a multi-segmented line or linestring to form one or two linestrings, depending on whether you delete an interior or an end segment. If a line consists of only one segment, the entire line will be deleted.

**Note:** This command can be especially helpful in fixing crossing breaklines in a surface.

#### To delete a segment:

1. Do one of the following:
  - Select **Line > Delete Line Segment**.

- Click the  icon on the toolbar.

The **Delete Line Segment** command pane displays.

2. Pick a line or linestring in a graphic view. The name is recorded in the **Line** box, and a "rubber-band" line appears between your cursor and the line.
3. Using the rubber-band line, move along the line and pick the segment you want to delete. The segment disappears.

**Tip:** You can also roll the mouse wheel to scroll through the line's segments.

4. Continue picking linestrings and then segments to delete, or click **Close**.

### Related topics

- [Create a Linestring](#) (on page 300)
- [Edit a Linestring's Horizontal Segments](#) (on page 301)

## Set a Line Elevation


Apply a single, constant elevation to a 2D line to make it a 3D linestring. You can also specify a vertical offset from the elevation that you enter and a surface to which the linestring will be added.

**Note:** You cannot set a line elevation on a line that already has elevations. To modify the existing elevations, add vertical control points using the **Edit Linestring** command.

**Tip:** The **Set Line Elevation** command is especially efficient when you have annotations that contain elevations next to your lines. You can simply pick each object and annotation consecutively.

**Tip:** If you need to apply the same elevation to multiple consecutive linestrings, you may want to join them into a single linestring first using the **Join** command.

### To set a constant elevation on a linestring:

1. Do one of the following:
  - Select **Line > Set Line Elevation**.
  - Click the  icon on the toolbar.

The **Set Line Elevation** command pane displays.

2. To apply a standard vertical offset to each elevation you enter, type a value in the **Vertical offset** box.

**Note:** This can be especially helpful when elevation values are specified by clicking on a text object, and when the resulting value is to be above or below the elevation reflected on that text. This commonly occurs in setting building pad elevations where the finished floor elevation is labeled, and the top of pad is to be beneath that by a specified slab thickness.

3. To make each line you pick a [member](#) (see "Edit a Surface by Adding and Removing Members" on page 328) of a specific surface, select the surface in the **Add to Surface** list.
4. Pick a line in a graphic view. Any CAD line is converted to a linestring.
5. Pick another object, such as an elevation annotation or contour line, that has an elevation you want to use, or type a value in the **Elevation** box. The elevation is applied to the entire linestring after adding the specified vertical offset.
6. Repeat steps 4 and 5 to add elevations to additional objects as needed.
7. Click **Close**.


### Related topics

- ❑ [Join Lines](#) (on page 315)
- ❑ [Edit a Linestring's Horizontal Segments](#) (on page 301)

## Explore an Object

Explore objects, such as lines and alignments, by viewing geometric values calculated at specific locations based on the position of the cursor on the object in a graphic view. Certain values are displayed dynamically at the cursor as you move it along the object. Additional values are displayed statically in the command pane when you click in the view to specify a fixed location.

### To explore an object:

1. Do one of the following:
  - Select **Tools > Explore Object**.
  - Click the  icon on the toolbar.

The **Explore Object** command pane displays.
2. Pick an object in a graphic view. The object is recorded in the **Object** box and values appear to the right of the cursor.
3. Move the cursor to see values at various locations along the object.
4. Click a location to report additional values in the command pane.
5. Click **Close**.

### Related Topics

- ❑ [Explore an Object](#) (on page 318)
- ❑ Coordinates Scroll Command

## Explode a Block

After you import CAD data, explode any blocks that contain objects that you want to move, modify, or delete individually. Exploding breaks apart group objects that contain multiple individual objects (and sometimes additional nested blocks).

### To explode a block using options:

1. Select **Edit > Explode Blocks**. The *Explode Blocks* command pane displays.
2. Select a block object in the *Project Explorer*, *Selection Explorer*, or a graphic view, or click **Options** and select an option in the list.
1. Uncheck the *Delete blocks after exploding* box if you want to retain the original block, as well as its individual component objects, after exploding it.


**Note:** It is unlikely that you will need or want to retain the original block.

2. Check the *Remove block definitions* box if you want to delete the hidden definition of the original block that you deleted by checking the *Delete blocks after exploding* box.

**Note:** This box is unavailable if you do not delete the exploded blocks. Block definitions cannot be removed if your project still contains blocks that reference them.

3. Click **Apply** if you want to explode additional blocks, or **OK** to close the command pane.

### To explode a block automatically:

1. Select a block in a graphic view or in the *Selection Explorer*.
2. Do one of the following:
  - Click the  icon on the toolbar.
  - Right-click and select **Explode Blocks** from the context menu.

The selected block is exploded using the options defined in the *Explode Blocks* command pane.

### Related topics

- ❑ Import CAD Files (.dwg/.dxf)
- ❑ [Import MicroStation Files \(.dgn\)](#) (on page 204)

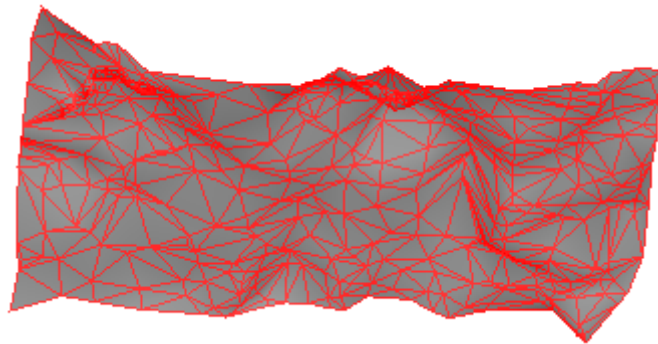
## CHAPTER 13

# Work with Surface Data

## Create and Edit a Surface

### Understanding Surfaces

A surface is a 3D digital representation of topography, formed by a mesh of contiguous triangles, which is known as a triangulated irregular network (TIN). The triangles are connected at their vertices, which are defined by points with horizontal positions (X and Y values) and elevations (Z values). You can either import surfaces, or create them using existing data in your project.



### Use surfaces to:

- Visualize and analyze the topography at different phases in your project:
  - Existing/as built terrain
  - Work in progress terrain
  - Finished earth
  - Proposed terrain (design)



- Compare one phase to another to generate volume reports for cut and fill earthwork operations.
- Represent stockpiles and depressions
- Create contour maps from topographic surveys.
- Upload a surface to a field device to check grades, or for accurate staking of an alignment or daylight boundary.

### Related topics

- ❑ [Create a Surface](#) (on page 323)
- ❑ [Workflow for Using Imported Surfaces](#) (on page 321)
- ❑ [Workflow for Created Surfaces](#) (see "Workflow for Creating Surfaces" on page 322)

## Workflow for Using Imported Surfaces

You may receive surface files in .ttm or .xml format from colleagues, clients, or field crews using field software. You can also create surfaces from point data when you import using the *Import Format Editor*. The general workflow for using imported surfaces is:

1. Import the file containing the surface. The surface is created as a single object, and appears in the *Project Explorer*.
2. Check the surface for quality to make sure it accurately reflects the project topography.
  - View the surface in plan view with contours.
  - Orbit and view the surface in 3D view.
  - Use the coordinate scroll to check the elevation at specific locations.
3. Edit surface properties, such as the classification, color, and other display characteristics.
4. Compute volumes by comparing the surface to another surface.
5. Upload the surface to a field device for staking, or export it to another application.

### Related Topics

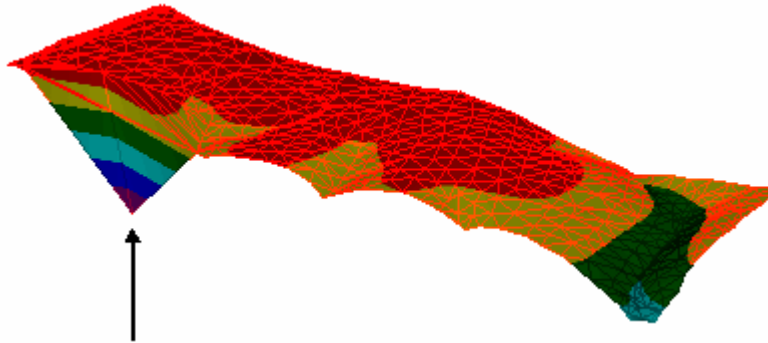
- ❑ [Import LandXML Files \(.xml\)](#) (on page 200)
- ❑ [Import Terrain Model Files \(.ttm\)](#) (see "Import Trimble Surface Files (.ttm)" on page 207)
- ❑ [Check Imported Surfaces](#) (on page 321)

## Check Imported Surfaces

You can import surfaces existing in Trimble DTMs (.ttm), LandXML files (.xml), or ASCII point files. Once you do, check the surface for quality and accuracy.

**To check an imported surface:**

1. Add contours to visualize the topography of the surface.
2. Spot check elevations. Pick a point in a graphic view to find the elevation of the surface at that location. The coordinate scroll also shows the elevation of the current cursor location.
3. In the 3D view, orbit around the surface and zoom in to see if everything looks valid. If it helps, change the vertical exaggeration.
4. Pick the surface and view its properties in the *Properties* pane.



Example of a surface with a point at an invalid elevation.

**Related Topics**

- ❑ [Import LandXML Files](#) (see "Import LandXML Files (.xml)" on page 200)
- ❑ [Import TTM Files](#) (see "Import Trimble Surface Files (.ttm)" on page 207)
- ❑ [Create Surface Contours at Intervals](#) (on page 343)
- ❑ [Edit a Surface by Changing Its Properties](#) (on page 338)
- ❑ [3D View Navigation](#) (on page 37)

**Workflow for Creating Surfaces**

You can create surfaces from data in your project. Here is the general workflow:

1. Import the data from which you want to create a surface. Objects that can be used to create surfaces include:
  - Points
  - Feature coded points
  - CAD data
2. Check and prepare the data.
  - Delete unneeded objects.
  - Organize data onto layers.
  - Add elevations to 2D CAD data.

3. Create a surface.
  - Select objects by layer, by elevation, or by picking them in a graphic view.
4. Check the surface for quality to make sure it accurately reflects the project topography.
  - View the surface in plan view with contours.
  - View the surface in 3D view.
  - Use the coordinate scroll to check the elevation at specific locations.
5. Edit the surface.
  - Add or remove members that form the surface.
  - Edit properties, such as elevations, of surface members.
  - Add breaklines to the surface.
  - Edit triangles at the surface's edge.
6. Compute volumes by comparing the surface to another surface.

## Related Topics

### Prepare Layered Data for a Surface

- ❑ Add and Edit Elevations on a 2D Line
- ❑ [Create a Surface](#) (see "Understanding Surfaces" on page 320)
- ❑ [Create Surface Contours](#) (see "Create a Surface Boundary and Contours" on page 339)

## Create a Surface

Certain types of objects in your data can be used to define a surface, including:


- Points
- Arcs, lines with elevation data, and polylines
- Alignments
- Sessions and vectors

When you create a surface, you select a set of these objects. These selected objects are called "members" of the surface. Members do not become part of the surface, they simply define it. The surface is created as a separate object. To change a surface, you add, remove, or edit the positions of members in the set. As a result, the surface object updates to reflect the changes.

**Note:** Before creating a surface, set the default *Maximum Edge Length* and the *Maximum Edge Angle* of surfaces in *Project Settings*. Select **Project > Project Settings**. Then click **Computational Settings** and **Surface** in the *Project Settings dialog*. Setting these can reduce the amount of trimming needed on the surface edge.

**Tip:** You can import points as a surface in the [Import Format Editor](#) (see "Import Data in a Custom Format" on page 209).

### To create a surface:

1. Do one of the following:
  - Select **Surface > Create Surface**.
  - Click the  icon on the toolbar.

The *Create Surface* command pane displays.
2. Type a name in the *Name* box.
3. Select a type in the *Surface Classification* list. This classification will be used to compare the surface to another surface with a different classification to calculate volumes for the *Earthwork Report*.
4. In a graphic view, pick objects to include as member of the surface, or click **Options** and select an option in the list.

**Note:** If you have organized your project into layers, select **ByLayer**.

5. Click **OK**. The surface appears in the graphic views and the *Project Explorer*.

**Tip:** To view surfaces more clearly, use a 3D view, and set your view filter to **Surfaces**.

6. Check the surface for quality to make sure it accurately reflects the project topography.
  - View the surface in a plan view with contours.
  - View the surface in 3D view.
  - Use the coordinate scroll to check the elevation at specific locations.

### Related topics

- ❑ Prepare Layered Data for a Surface
- ❑ [Surface Options](#) (on page 324)
- ❑ [Workflow for Creating Surfaces](#) (on page 322)

## Surface Options

Use these options to identify the state of a surface in the construction process. They are available in the *Create Surface* command pane.

## Options

**Surface Classification** *Unclassified* - Select this when none of the classifications apply, or when you do not need to compare the surface to another classification of surface.

*Original* - Select this when the surface represents the current state of the site's topography, i.e. the existing terrain.

*Work-in-progress* - Select this when the surface represents a state in between the other, more defined, states.

*Finished earth* - Select when the surface represents the “top-of-dirt” of the finished design. This is usually the finished product for dirt-moving contractors.

*Design* - Select this when the surface represents the proposed surface of a project, typically running across the top of pavement, building pads, concrete sidewalks etc.

In most cases, this is the surface defined by the contours and spot elevations on plans from the engineer.

*As-built* - Select this when the surface represents the completed project.

Often this state is used to verify to an owner or regulatory agency that the site construction conforms to the plans.

*Stockpile* - Select this when the surface represents a storage area for earthwork material.

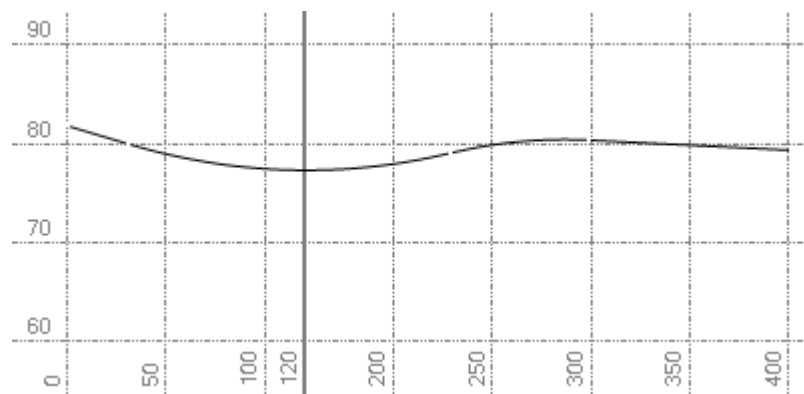
*Depression* - Select this when the surface represents a hole where material has been removed.

## Related topics


- ❑ [Create a Surface](#) (see "Understanding Surfaces" on page 320)

## Create and View a Surface Profile

Create a surface profile to review the elevation of a surface along a vertical alignment. To do this, you must have a surface and an alignment that coincide. If you simply want to see a surface profile without relation to an alignment, use the **Surface Slicer View** (see "View a Slice of a Surface" on page 327).



**To create a surface profile:**

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Surface > Create Surface Profile**.

The *Create Profile of Surface* command pane displays.
2. Select a surface in the *Surface* list.
3. Select an alignment in the *Alignment* list.
4. Click **OK**. The profile of the surface displays with the alignment in the profile view.

**To view a surface cross-section along an alignment:**

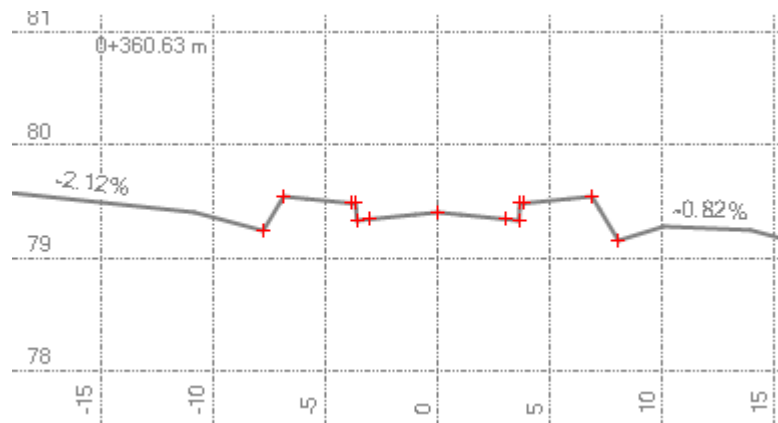
1. In the *Project Explorer*, select the alignment that coincides with the surface.
2. Right-click and select **New Profile View** from the context menu. The profile view of the alignment displays.

**Related topics**


- ❑ [Profile View](#) (on page 26)
- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Create a Surface](#) (see "Understanding Surfaces" on page 320)
- ❑ [View a Slice of a Surface](#) (on page 327)

**Create and View a Surface Cross-Section**

Create a surface cross-section to check geometry along a single, specific alignment anywhere it coincides with a single, specific surface. To do this, the surface and alignment must be coincident. The view changes depending on where you are along the alignment. When the alignment or surface that the view is based on is modified or deleted, the view updates accordingly. Multiple cross-section views can be open concurrently.



**To create a surface cross-section:**

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Surface > Create Surface Cross-Section**.

The *Create Surface Cross-Section* command pane displays.
2. Select a surface in the *Surface* list.
3. Select the coincident alignment in the *Alignment* list.
4. Click OK.

**To view a surface cross-section along an alignment:**

1. In the *Project Explorer*, select the alignment that coincides with the surface.
2. Right-click and select **New Cross-Section View** from the context menu. A cross-section view displays.
3. Click and drag the slider at the bottom of the view to see the cross-sections along the alignment.

or

Type a station value in the box to the left of the slider to see the cross-section at a specific station. After you click in the station box, you can also move the cursor into the plan view and click anywhere along the alignment to specify the station.

**Note:** The cross-section view maintains the same scale as you move the station slider. Red tick marks denote where the cross-section crosses points or breaklines. Bold vertical lines denote station equations. At certain view magnifications, slope values appear above segments. To hide cross-section slope values, right-click the cross-section, and select **Properties** from the context menu. In the *Properties* pane, select **Hide** in the *Label slope* list.

**Related topics**

- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Create a Surface](#) (see "Understanding Surfaces" on page 320)
- ❑ [Profile View](#) (on page 26)
- ❑ [View a Slice of a Surface](#) (on page 327)


**View a Slice of a Surface**

Use the surface slicer view to check any surface cross-section by slicing vertically through the surface. You need to have the plan view and the surface slicer view open concurrently. Multiple surface slicer views can be open at a time, and you can view multiple surfaces in the view concurrently.

**Note:** If you use a command while viewing a surface slice, the focus will be in the **Command** pane. To return focus to the surface slicer view, click in the view's **From** or **To** box.

### To view a surface slice:

1. Do one of the following:

- Select **Surface > Surface Slicer View**.
- Click the  icon on the toolbar.

The surface slicer view displays.

2. Click in the **From** box and pick a starting point for the slice in the plan view, or type a coordinate (in the format X,Y) or point ID in the box.
3. Pick an ending point for the slice, or type a coordinate or point ID in the **To** box.

**Tip:** After picking the **From** point, you can also move the cursor across the surface to view the cross-section slice dynamically, without picking a **To** point. The surface slicer view automatically scales to fill the view as you move the **cursor**.

**Note:** Red tic marks denote where the slice crosses points or breaklines. At certain view magnifications, slope values appear above segments. To hide cross-section slope values, right-click the cross-section, and select **Properties** from the context menu. In the **Properties** pane, select **Hide** in the **Label slope** list.

**Note:** You can also right-click in the view to access COGO controls and snaps when picking points.

4. To add additional surfaces to the view, click **Surfaces**. The **Select Surfaces** dialog displays.
5. Check boxes for the surfaces to include in the surface slicer view, and click **OK**.

### Related topics

- ❑ [Create and View a Cross-Section](#) (see "Create and View a Surface Cross-Section" on page 326)

## Edit a Surface by Adding and Removing Members


Members are not part of a surface; they are simply used to define the surface. Edit a surface by adding to, and removing from, the set of members. As a result, the surface updates to reflect the changes.

You may want to prepare your data by organizing members by layer before using this command.

**Warning:** If you add objects on or beyond the outer edge of a surface, the surface is recalculated and all prior edge trimming is lost. Similarly, if you remove objects from a surface edge, all prior trimming is lost.



**To add and remove surface members:**

1. Do one of the following:
  - Select **Surface > Add/Remove Surface Members**.
  - Click the  icon on the toolbar.
  - Select a surface in the *Project Explorer*, right-click, and select **Add/Remove Surface Members**.

The *Add/Remove Surface Members* command pane displays.

2. Select a surface to edit in the **Surface** list, unless you have already selected one.
3. In a graphic view, pick objects to add to the set of surface members, or click **Options** and choose a selection method in the context menu. You can also select objects in the *Project Explorer*.

**Tip:** Sometimes it is difficult to pick surface members without picking the surface itself. Here are some ways to make it easier:  
Use the **View Filter** to control the visibility and selectability of surfaces and their members. To view surfaces more clearly, use a 3D view, and set your view filter to **Surfaces**. If more than one object is in the pick aperture when you pick, a list of the possible selections appears. Refine your pick by selecting from the list. Use [windowing and crossing](#) (see "Graphic Selection Methods" on page 43) selection methods to pick only what you need.

4. Click **Add** to update the surface based on the selected objects. The surface changes in the 3D view.
5. In a graphics view, pick surface members to remove, or click **Options** and choose a selection method in the context menu.
6. Click **Remove** to update the surface based on the selected objects.
7. Click **Close**.


**Related topics**

- ❑ [Edit a Surface by Creating a Breakline](#) (on page 334)
- ❑ [Edit a Surface by Changing a Point Coordinate](#) (on page 329)
- ❑ [Edit a Surface by Changing Its Properties](#) (on page 338)
- ❑ [Edit a Surface by Trimming Edge Triangles](#) (on page 336)
- ❑ Prepare Layered Data for a Surface

**Edit a Surface by Changing a Point Coordinate**

Make your surface more accurate by editing inaccurate coordinate heights in the *Properties* pane.

**To edit surface member properties:**

1. Do one of the following:
  - Pick the point(s) in a graphic view, or select it in the *Project Explorer*.
2. Click the  icon next to the point in the *Project Explorer* to expand it, and select the coordinate.
3. Double-click the coordinate, or right-click it and select **Properties** from the context menu. The *Properties* pane displays.
4. Edit any of the values in the **Coordinates** group as needed.
5. Click **Close**. The surface updates accordingly.

**Related topics**

- [Edit a Surface by Adding and Removing Members](#) (on page 328)
- [Point Coordinate Options](#) (on page 275)

**Point Coordinate Options**

Use these options to edit point coordinates when the point is a member of a surface. The surface that references the edited member updates accordingly. These options are found in the *Properties* pane for a point coordinate.

**Member type****Properties that affect surfaces****Coordinate**

**Northing** - Edit this to move a surface vertex north or south.

**Easting** - Edit this to move a surface vertex east or west.

**Elevation** - Edit to move a surface vertex up or down, relative to sea-level.

**Latitude** - Edit this to move a surface vertex north or south.

**Longitude** - Edit this to move a surface vertex east or west.

**Height** - Edit this to move a surface vertex up or down.

**Session**

**Height (point)** - Edit this to move a surface vertex up or down.

**Related topics**

- [Edit a Surface by Changing a Point Coordinate](#) (on page 329)
- [Edit a Surface by Adding and Removing Members](#) (on page 328)

**Understanding Breaklines**

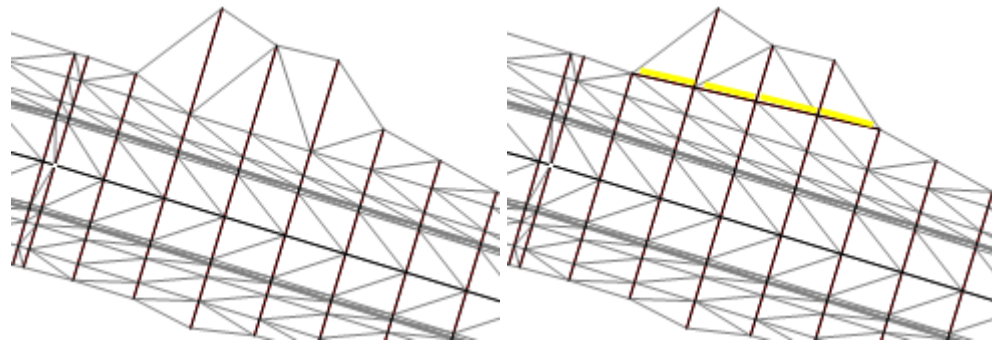
Breaklines are lines that help to define the shape of surfaces by controlling how triangles in the surface mesh are formed; triangles forming a surface never cross a breakline. Use breaklines to make surfaces more accurate. For example, if a square area on your surface is formed by two triangles using the wrong opposing points, you can switch the direction of the diagonal to reform the two triangles. This might be done to get drainage to work as the site designer intended, or simply to 'smooth' the surface.

The simplest way to create a breakline is to add it between points where surface triangles converge (vertices). You can also create breaklines by picking locations on the surface that are not vertices. The surface triangles are redrawn, creating vertices at the points you picked.

You might add a breakline to a surface to represent:

- A stream bed
- A curb line

A linear feature that was not recorded in the field Here is an example of how adding a breakline (in yellow) can change a surface:



In addition to the breaklines that you actively create, other breaklines are automatically created in surfaces when you import or select linear objects to create a surface. Any linear object, such as a polyline or an alignment, that you use in the creation of a surface will be represented as a breakline in the surface. If you modify the linear object, the breakline will update in response.

**Note:** When you use an alignment in the creation of a surface, you can constrain the resulting breaklines horizontal and vertical distance from the alignment by setting tolerances in the *Project Settings*. Select **Project > Project Settings**. Then click **Computational Settings** and **Surface** in the left pane.

### Related topics


- ❑ [Edit a Surface by Creating a Breakline](#) (on page 334)

## Create and Edit a Simple Breakline

Simple breaklines are simplified linestrings that help to define the shape of a surface as do other 3D line types that act as breaklines, by controlling how triangles in the surface mesh are formed. Triangles forming a surface never cross a breakline. Use breaklines of any type to more accurately reflect where surface topography changes.

Create a simple breakline is to add linestring segments between points where surface triangles converge (vertices), but you can also add simple breakline segments freely between any two locations on or near the surface. The addition, deletion or any edit to a 3D line serving as a surface breakline will cause the entire surface to be reformed. Unlike more complex linestrings, simple breaklines do not use vertical control points; you can enter elevations for each new vertex or derive elevations from existing vertices.

### To create a simple breakline:

1. Do one of the following:
  - Select **Surface > Create Breakline**.
  - Click the  icon on the toolbar.

The *Create Breakline* command pane displays.
2. In the *Name* box, type an identifier for the linestring as you want it to appear in the *Selection Explorer* and in graphic views when you select from multiple objects. You can also use the name to select the linestring in the *Advanced Select* command.
3. Select a layer on which to create the linestring in the *Layer* list, or select **<<New layer>>** to create a new layer for the linestring.
4. Select an option for how smoothly the surface is rendered at this particular linestring in the *Surface sharpness* list.
5. Select the surface to which the linestring will be assigned as a member in the *Add to surface* list.
6. Click **OK**. The *Edit Linestring* command pane displays with reduced options appropriate for creating a simple breakline.
7. In the *Start Point* group's *Type* list, select **Coordinate** to start the linestring at any location, or select **Point ID** to attach the linestring to a named point.
8. Click in either the *Coordinate* box or *Point ID* box (depending on what you chose for the starting point type), and use one of these ways to specify each point along the linestring:
  - In a graphic view, pick a vertex on the surface, or pick a named point (which may be a member of the surface and therefore one of its vertices)
  - Pick any location on or near the surface in the graphic view, and then type an elevation in the *Elevation* box. Then, click **Save** or press **[Enter]**.
  - Type a coordinate, press **[Tab]**, and type an elevation in the *Elevation* box. Click **Save** or press **[Enter]**.

The first point of the simple breakline is saved and you are prompted to add the second point and specify how the segment between them is formed.

**Note:** To use the elevation from a point you pick anywhere on a surface, right-click and select **From Surface Snap** from the context menu. You are prompted to pick the point in the **From Surface Snap** command pane. Then, click **Save** or press **[Enter]**.

**Note:** To avoid picking points off of the surface (with no elevation), uncheck the **Free** snap in the **Snap Mode** (see "Snaps Modes and Commands" on page 89) dialog when you create a simple breakline or linestring. If you are prompted for an elevation, you may have missed the vertex point you were trying to pick.

**Note:** You can also draw a 'freehand' breakline without using any existing data. In fact, you do not even need to have a surface to create a breakline.

9. Keep adding segments to the linestring using one of the methods in step 8.
10. Click **Close**. The simple breakline appears on the surface in graphic views, but does not appear in the **Project Explorer**.

**Note:** Once created, linestrings can be graphically selected and added to other surfaces.

**Tip:** To view changes to your surface more clearly as you work, open a 3D view and move it to a new tab group next to your plan view so you can see both views concurrently.

#### To edit a simple breakline:

- Use the operations in **Edit a Linestring's Horizontal Segments** (on page 301).

#### Related topics

- ❑ [Edit a Linestring's Horizontal Segments](#) (on page 301)
- ❑ [Create a Linestring](#) (on page 300)
- ❑ [Understanding Breaklines](#) (on page 330)

### Breakline Options

Use these options and those listed in **Horizontal Linestring Options** to create a breakline. They are available in the **Create Breakline** and **Edit Linestring** command panes. After creating a breakline, these properties and others can be edited in the **Properties** pane.

## Options

<b>Name</b>	Type an identifier for the breakline.  This name will appear in the <b>Selection Explorer</b> and in graphic views when you select from multiple objects. In addition, you can use the name to select the breakline using the <b>Advanced Select</b> command.
<b>Layer</b>	Select the layer on which you want the breakline to reside, or select << <b>New layer</b> >> to create a new layer for the breakline.  Segregating data onto logical layers makes it easier to filter your graphic views and select related data.
<b>Surface sharpness</b>	Select an option for how smoothly surface colors graduate at breaklines.  <b>Soft</b> - Select this to have shading gradually change at the breakline, giving the visual appearance of a soft edge.  <b>Sharp</b> - Select this to have shading abruptly change at the breakline, giving the visual appearance of a sharp edge.
<b>Add to surface</b>	Select the surface that you want the breakline to modify.

## Related topics

- ❑ [Create and Edit a Breakline](#) (see "Create and Edit a Simple Breakline" on page 312)
- ❑ [Edit a Surface by Creating a Breakline](#) (on page 334)


## Edit a Surface by Creating a Breakline

3D lines of several types can be designated as members of a surface. They serve to influence the shape of that surface by acting in the classic manner as breaklines. These lines can exhibit complex curvilinear geometry and still serve as breaklines, by employing the breakline approximation parameters as defined in the computational settings for a surface in the **Project Settings** (see "Choose Project Settings" on page 147) dialog.

One of the types of lines that can serve as a member of a surface, thereby acting as a breakline, is the linestring. Like alignments, linestrings can exhibit rather complex curvilinear geometries. But they can also exist as simple strings of straight line segments, connecting 3D coordinate locations, as in the case of the more classic breaklines associated with typical TIN models. A linestring of that nature is referred to here as a simple breakline; a specific command has been devised to allow you to create a linestring of that nature and assign it as a member of a surface at the same time, while defining its surface sharpness. In using the Create Simple Breakline command, you will in fact be creating a linestring object, though through a simpler user interface and one that supports the line's association with a surface as a part of its creation.

The easiest way to create a simple breakline is to add segments between surface vertices (the points where surface triangles converge), but you can also add linestring segments freely between any two locations on or near the surface. Any geometric edit to a breakline of any kind will cause the entire surface to be reformed.

**To create a simple breakline:**

1. Do one of the following:
  - Select **Surface > Create Breakline**.
  - Click the  icon on the toolbar.

The **Create Breakline** command pane displays.

2. In the **Name** box, type an identifier for the linestring as you want it to appear in the **Selection Explorer** and in graphic views when you select from multiple objects. You can also use the name to select the linestring in the **Advanced Select** command.
3. Select a layer on which to create the linestring in the **Layer** list, or select **<<New layer>>** to create a new layer for the linestring.
4. Select an option for how smoothly the surface is rendered at the linestring in the **Surface sharpness** list.
5. Select the surface to which the linestring will be assigned as a member in the **Add to surface** list.
6. Click **OK**. The **Edit Linestring** command pane displays with reduced options appropriate for creating a simple breakline
7. In the **Start Point** group's **Type** list, select **Coordinate** to start the breakline using any location, or select **Point ID** to start the breakline using a named point.
8. Click in either the **Coordinate** box or **Point ID** box (depending on what you chose for the starting point type), and use one of these ways to specify each point along the breakline:
  - In a graphic view, pick a vertex on the surface, or pick a named point (may also be a vertex)
  - Pick any location on or near the surface in the graphic view, and then type an elevation in the **Elevation** box. Then, click **Save** or press **[Enter]**.
  - Type a coordinate, press **[Tab]**, and type an elevation in the **Elevation** box or right-click for [COGO elevation options](#) (see "Enter an Elevation" on page 125). Click **Save** or press **[Enter]**.

The first point of the linestring is saved and you are prompted to add the second point and specify how the segment between them is formed.

**Note:** To use the elevation from a point you pick anywhere on a surface, right-click and select **From Surface Snap** from the context menu. You are prompted to pick the point in the **From Surface Snap** command pane. Then, click **Save** or press **[Enter]**.

**Note:** To avoid picking points off of the surface (with no elevation), uncheck the **Free** snap in the **Snap Mode** (see "Snaps Modes and Commands" on page 89) dialog when you create a breakline. If you are prompted for an elevation, you may have missed the vertex point you were trying to pick.

**Note:** You can also draw a 'freehand' breakline without using any existing data using method below. In fact, you don't even need to have a surface to create a breakline.

9. Keep adding segments to the breakline using one of the methods in step 8.
10. Click **Close**. The breakline appears on the surface in graphic views, but does not appear in the **Project Explorer**.

**Note:** Once created, breaklines can be graphically selected and added to other surfaces.

**Note:** To see the properties of a breakline (in the **Properties** pane) you have actively created, select the breakline in a graphic view.

**Tip:** To view changes to your surface more clearly as you work, open a 3D view and move it to a new tab group next to your plan view so you can see both views concurrently.

#### To edit a simple breakline:

- Use the operations in **Edit a Linestring's Horizontal Components** (see "Edit a Linestring's Horizontal Segments" on page 301).

#### Related topics

- [Edit a Surface by Adding and Removing Members](#) (on page 328)
- [Edit a Surface by Changing Its Properties](#) (on page 338)
- [Edit a Surface by Changing a Point Coordinate](#) (on page 329)
- [Edit a Surface by Trimming Edge Triangles](#) (on page 336)

### Edit a Surface by Trimming Edge Triangles

Clean-up the edges of surfaces in your project so that you can accurately calculate volumes when comparing surfaces.




**Note:** Before trimming the triangles at a surface's edge, adjust the *Maximum edge length* and the *Maximum edge angle* of the surface in the *Properties* pane. Setting these can reduce the amount of trimming needed on the surface edge. In general, trim the edge triangles of a surface after you have made all other surface edits.

**Warning:** If the program is forced to recompute the edge of a surface, then all edge trimming edits will be lost.

Things that would cause this are:

- Changing the *Maximum edge length* of the surface in the *Properties* pane.
- Changing the *Maximum edge angle* of the surface in the *Properties* pane.
- Adding a new surface member outside of the edge of the surface.
- Removing surface members that lie on the edge of the surface.

### To trim a triangle:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Surface > Trim Surface Edge**.
  - Right-click the surface in a graphic view, or the *Project Explorer* and select **Trim Surface Edge** from the context menu.

The *Trim Surface Edge* command pane displays.

2. Select a surface to trim in the *Surface to Trim* list.
3. In a graphic view, pick a point to start the trim line or right click for options, or type a coordinate in the *Outside location* box.
4. In the view, pick a point to end the trim line or right click for options, or type a coordinate in the *Inside location* box.
5. Click **Apply** to keep trimming, or **Close**. Any triangle edges occurring outside of the outermost breakline are trimmed, removing the related triangle from the surface.

**Tip:** Trimming stops when a breakline on the surface is encountered. To trim deeper into the surface, remove the breakline using the Edit Surface command. Conversely, you can add breaklines to a surface to prevent trimming an edge beyond a certain point.

**Note:** Trimming maintains a valid surface based on the members used to form the surface. Because any point on the surface will always be connected to a triangle, the order in which you trim edge triangles can result in different formations. If you find that you cannot trim a certain triangle, you may need to remove the member on which it is based.

**Warning:** If you add objects on or beyond the outer edge of a surface, the surface is recalculated and all prior edge trimming is lost. Similarly, if you remove objects from a surface edge, all prior trimming is lost.

### Related topics

- ❑ [Edit a Surface by Adding and Removing Members](#) (on page 328)
- ❑ [Edit a Surface by Creating a Breakline](#) (on page 334)
- ❑ [Edit a Surface by Changing Its Properties](#) (on page 338)
- ❑ [Edit a Surface by Changing a Point Coordinate](#) (on page 329)
- ❑ [Surface Properties](#) (on page 338)

## Edit a Surface by Changing Its Properties

Edit a surface object by controlling its triangle distance and angle values, as well as display properties in the **Properties** pane.

### To edit surface properties:

1. Do one of the following:
  - Pick the surface in a graphics view, right-click, and select **Properties** from the context menu.
  - Double-click the surface in the **Project Explorer**, or right-click it and select **Properties** from the context menu.

The **Properties** pane displays.

2. Click in an available property box, and make changes as necessary.
3. Click **Close**.

### Related topics

- ❑ [Edit a Surface by Adding and Removing Members](#) (on page 328)
- ❑ [Edit a Surface by Creating a Breakline](#) (on page 334)
- ❑ [Edit a Surface by Changing Its Properties](#) (on page 338)
- ❑ [Edit a Surface by Changing a Point Coordinate](#) (on page 329)
- ❑ [Edit a Surface by Trimming Edge Triangles](#) (on page 336)
- ❑ [Surface Properties](#) (on page 338)

## Surface Properties

Use these options to change a surface's edge prior to trimming. They are available in the **Properties** pane when a surface is selected. In addition, you can set display properties for how a surface appears in graphic views.

## Options

**Max Edge Distance** Type a value for the maximum length that one side of a triangle can be if it lies on the edge of the surface.

Basically, this value defines how the longest distance that points can be connected on a surface edge.

**Max Internal Angle** Type a value for the maximum angle that a surface triangle can use.

Practically, this value limits the amount of long, but very narrow triangles that can form on the edge of a surface.


## Related topics

- ❑ [Edit a Surface by Changing Its Properties](#) (on page 338)
- ❑ [Edit a Surface by Trimming Edge Triangles](#) (on page 336)

## Run a Surface Report

Run a surface report to see surface measurements and limits, as well as the number of triangles, vertices, and other items in a surface in your project.

### To generate a surface report:

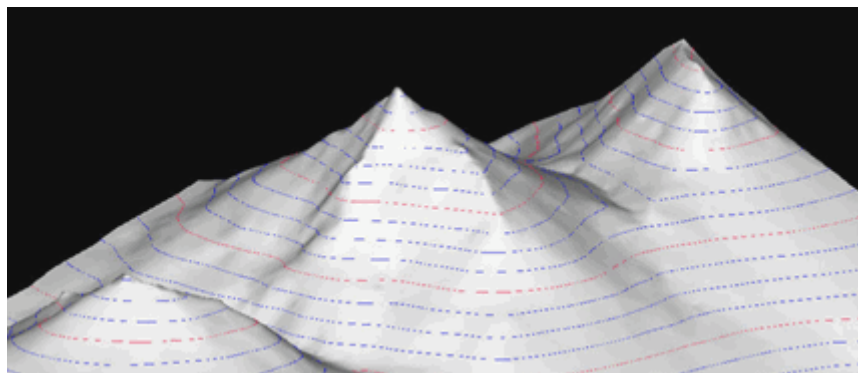
1. Do one of the following:
  - Select **Reports > Surface Information Report**.
  - Click the  icon.

The *Surface Information Report* command pane displays.
2. Select the surface you want to generate a report for in the **Surface** list.
3. Click **OK**. The report displays in your default Web browser.

## Related topics

- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Create a Surface Boundary and Contours



A boundary is a line that delineates a portion of a surface. Boundaries can be polygonal or defined by specifying an offset from an alignment.

Contours are lines that show the topography of a surface at a constant elevation. Contour objects are separate from the surface objects to which they are attached, but they are nested under surfaces in the *Project Explorer*. You can associate multiple contour objects with a single surface object. When a surface is changed, the contours on the surface update to reflect any changes in elevation.

### Related topics


- ❑ [Create a Surface Boundary](#) (see "Create a Boundary" on page 340)
- ❑ [Create a Surface Contour at an Elevation](#) (on page 344)
- ❑ [Create Surface Contours at Intervals](#) (on page 343)

## Create a Boundary

Create a boundary to delineate a portion of a surface. This is useful if you do not want to send an entire surface to field software; it enables you to include a smaller surface model with your design data.

**Note:** Boundaries cannot be used to create or modify a surface.

### To create a boundary by drawing a polygon:

1. Do one of the following:
  - Select **Line > Create Boundary**.
  - Click the  icon on the toolbar.

The **Create Boundary** command pane displays.
2. Type an identifier for the boundary in the **Name** box.
3. Select the layer on which you want to boundary to reside in the **Layer** box.

**Note:** If the layer does not exist in your project yet, you can create it by selecting **<New>** in the box.

4. Select **Polygon** in the **Creation Method** group to define the boundary by drawing a polyline around an area.
5. Click in the **Point** box and pick a starting point for the polygonal boundary in the plan view, or type a coordinate or point ID in the box and click **Apply**.
6. Pick another point, or type a coordinate or point ID in the **Additional Point** box to draw the first line of the boundary. A "rubber-band" line appears, showing you how the boundary will be closed when you click **Close**.
7. Continue specifying points until you have completed the boundary.
8. To begin another boundary, click in the **Name** box, and repeat steps 2 - 7.

9. Click **Close**. The boundary appears in the view and the *Project Explorer*.

#### To create a boundary offset from an alignment:

1. Follow steps 1 - 3 above.
2. Select **Alignment corridor** in the *Creation Method* group to define the boundary by specifying an offset around a selected alignment.
3. Select an alignment to create a corridor around in the **Alignment** box.
4. Click in the **Left offset** box and pick a point in the view, or type a distance in the box to specify the left boundary of the corridor parallel to the alignment.

**Note:** Since the alignment itself denotes a zero offset, typically the left offset value will be a negative number, unless you want to create a corridor on only one side of the alignment, such as for a sidewalk.

5. Pick a point in the view, or type a distance in the **Right offset** box to specify the right boundary of the corridor parallel to the alignment.
6. To create the boundary only between specific stations along the alignment, check the **Limit by station** box.
7. Click in the **Begin station** box and pick a point along the alignment, or type a value in the box.
8. Pick a point along the alignment, or type a value in the **End station** box.
9. Click **OK**. The boundary appears in the view and the *Project Explorer*.

#### Related topics

- [Boundary Options](#) (on page 341)
- [Add and Edit a Design Model](#) (on page 398)

### Boundary Options

Use these options to specify a subset of a surface by creating a boundary. They are available in the **Create Boundary** command pane.

## Options

<b>Name</b>	Type a unique name for the boundary.
<b>Layer</b>	Select the layer on which you want to boundary to reside. You can also select <b>&lt;New&gt;</b> to create a new layer for the boundary.
<b>Creation method</b>	<b>Polygon</b> - Select this to define the boundary by drawing a polyline around an area. <b>Alignment corridor</b> - Select this to define the boundary by specifying an offset from a selected alignment.

## Polygon options

<b>Point</b>	Pick a point in the view, or type a coordinate or point ID in the box to draw the first line of the boundary.
<b>Additional point</b>	Pick a point in the view, or type a coordinate or point ID in the box to draw additional lines along the boundary. Open polygons will be closed when you click in the <b>Name</b> box to begin a new boundary, or when you close the command.

## Alignment corridor options

<b>Alignment</b>	Select the alignment around which to create the boundary corridor.
<b>Left offset / Right offset</b>	Pick points in the view, or type distances in the boxes to specify the left and right boundaries of the corridor parallel to the alignment. <hr/> <b>Note:</b> Since the alignment itself denotes a zero offset, typically the left offset value will be a negative number, unless you want to create a corridor on only one side of the alignment, such as for a sidewalk. <hr/>
<b>Limit by station</b>	Check this box to create the boundary only between specific stations along the alignment.

**Begin station /  
End station**

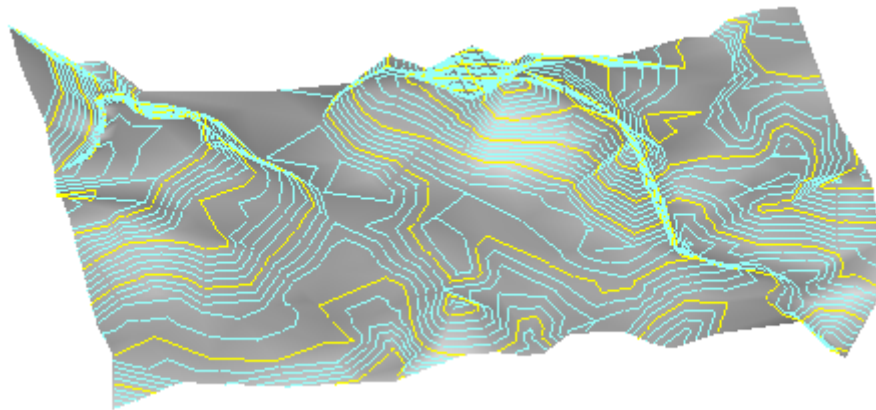
Pick points along the alignment, or type values in the boxes to specify the first and last stations in the corridor.


**Related topics**

- [Create a Boundary](#) (on page 340)

**Create Surface Contours at Intervals**

Add contour lines to a surface at regular elevation intervals to help visualize the topography.

**To create contours at intervals:**

1. Pick the surface you want to add contours to in the graphics view, or select it in the *Project Explorer*.
2. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Surface > Create Contours**.
  - Select a surface in the project explorer, right-click, and select **Create Contours** from the context menu.

The *Create Contours* command pane displays.

3. Confirm the surface to which you want to add contours in the **Surface** box.
4. Type a name for the contour object in the **Name** box.
5. Type a value for the vertical distance between contours in the **Contour interval** box. The *Estimated contours* value updates in the **Surface Information** group below.

6. Type a value for the spacing of index contours in the *Index frequency* box. Index contours are the major contours, while the other contours are the minor contours.
7. Select a layer on which to place the contour object in the *Layer* box.
8. Select a display color for the contours in the *Contour color* box.
9. Select a display color for the index contours in the *Index color* box.
10. Click **OK**. The contours appear on the surface in graphic views, and under the surface in the *Project Explorer*.

#### To edit contours:

1. Do one of the following:
  - Double-click the contour object in the *Project Explorer*, or right-click it and select **Properties** from the context menu.
  - Pick a contour line in a graphic view, right-click, and select **Properties** from the context menu.

The *Properties* pane displays.

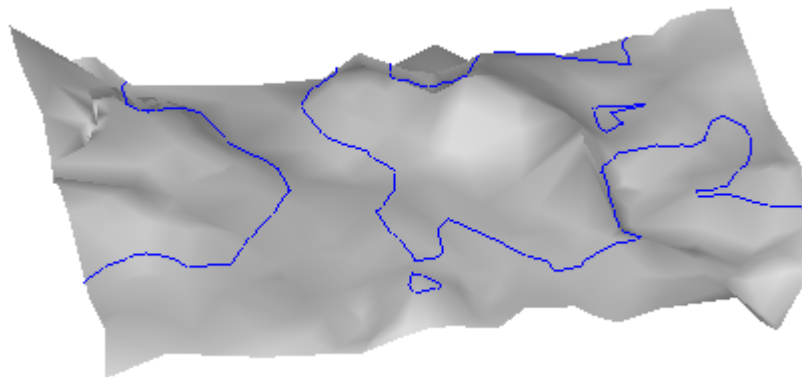
2. Edit any available properties as needed.
3. Click **Close**.

#### Related topics

- ❑ [Create a Surface Contour at an Elevation](#) (on page 344)
- ❑ [Surface Contour Options](#) (see "Contour Options" on page 347)


### Create a Surface Contour at an Elevation

Add a single contour to mark a specific elevation and help visualize its topography. For example, you might want to create a contour line to indicate a flood plain or a cut/fill line.





**To create a contour at an elevation:**

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Surface > Create Contour at Elevation**.
  - Pick a surface in a graphic view, or select it in the *Project Explorer*, right-click and select **Create Contour at Elevation** from the context menu.

The *Create Contours at Elevation* command pane displays.

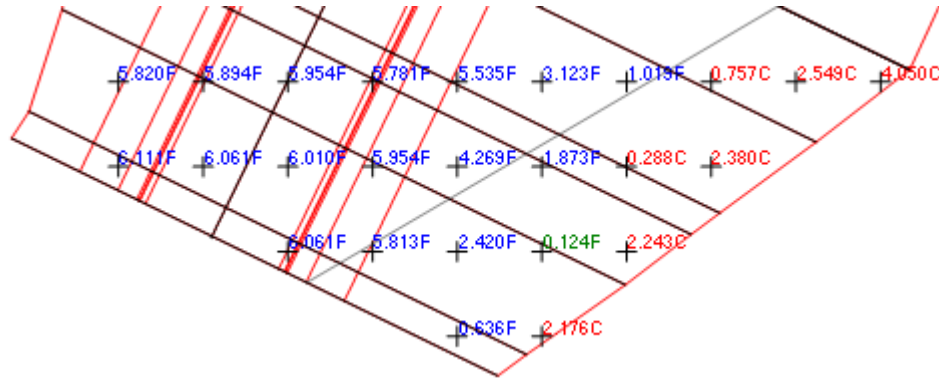
2. If necessary, select the surface to which you want to add a contour in the *Surface* box.
3. Type a name for the contour in the *Name* box.
4. In the view, pick a point, or right-click for options, or type an elevation in the *Elevation* box. The elevation should be between the *Minimum elevation* and *Maximum elevation* displayed in the *Surface Information* group.
5. Select a layer on which to place the contour in the *Layer* box.
6. Select a display color for the contour in the *Contour color* box.
7. Click **OK**. The contour appears on the surface in the graphics view, and under the surface in the *Project Explorer*.

**Related topics**


- [Create Surface Contours at Intervals](#) (on page 343)
- [Surface Contour Options](#) (see "Contour Options" on page 347)

## Create a Surface Cut/Fill Grid

Create a grid of values labeling the elevation differences between two surfaces. The measurements are color-coded to indicate where earth needs to be cut or filled. You can also create a grid of values labeling the absolute elevations of a single surface. The resulting grid is a single object that will dynamically update in response to edits to the related surfaces. The order in which you specify the surfaces, or their classifications (as defined when they are created, and available in the surface properties) will determine the *Initial* and *Final* surfaces.



### To create a cut/fill grid:

- Do one of the following:
  - Select **Surface > Create Surface Cut/Fill Grid**.
  - Click the  icon on the toolbar.

The *Create Surface Cut/Fill Grid* command pane displays.
- To compare two surfaces, select **Cut/Fill**. To show the elevations of a single surface, select **Elevation** and a surface, and skip to step 5.
- Select a surface in the **Initial surface** list. You can select a surface with any classification, but the initial surface should reflect the state of topography before the surface you select as the final surface.
- Select a surface in the **Final surface** box. Again, you can select a surface with any classification, but the final surface should reflect the state of topography after the surface you selected as the initial surface.
- Select the layer on which you want the grid to reside in the **Layer** box, or select **<<New layer>>** to create a new layer for the grid.
- Select a style that controls the text font, font style, justification, and size for the grid annotations in the **Text style** box, or select **<<New style>>** to define a new text style.
- In the **Grid spacing** box, type a value for the uniform interval at which the grid lines or tick marks for the measurements will be spaced.

8. Select an option for how the location of each measurement will be denoted in the *Grid style* box.
9. Select the number of decimals to use in the measurement in the *Decimal precision* list.
10. Click **OK**. The cut/fill grid is created and appears in graphic views and the *Project Explorer*. Additional properties, such as the grid's origin point and rotation, and cut, fill, and zero colors, can be set in the *Properties* pane.

#### Related topics

- ❑ [Surface Cut/Fill Grid Options](#) (on page 348)
- ❑ [Create Surface Contours at Intervals](#) (on page 343)

#### Contour Options

Use these options to define contour lines and their spacing. They are available in the *Create Contours* and *Create Contour at Elevation* command panes.

## Options

<b>Surface</b>	Select the surface to which you want to add the contours.
<b>Name</b>	Type an identifier for the contour object. Duplicate names are allowed.
<b>Elevation</b>	Type a value, or pick a point in a graphic view, to specify the elevation.
<b>Contour interval</b>	Type a value, or pick two points in a graphic view, for the distance between each contour line. Distance is measured vertically in project units. This setting affects the <b>Estimated contours</b> in the read-only <b>Surface Information</b> group.
<b>Index contour frequency</b>	Type a value for the frequency of major to minor contour lines. Entering 5 means that every fifth line will be an index contour line.
<b>Layer</b>	Select the layer on which the contour object will be placed, or select <b>&lt;New layer&gt;</b> to open the <b>New Layer</b> dialog, in which you can create another layer.
<b>Contour color</b>	Select a color for the minor contour lines, or select to have contours derive their colors by layer.
<b>Index contour color</b>	Select a color for the major contour lines, or select to have contours derive their colors by layer.
<b>Maximum elevation</b>	This displays the elevation at the highest point on the selected surface.
<b>Minimum elevation</b>	This displays the elevation at the lowest point on the selected surface.
<b>Estimated number of contours</b>	This displays the number of contour lines that will appear on the surface.

## Related topics

- ❑ [Create a Surface Contour at an Elevation](#) (on page 344)
- ❑ [Create Surface Contours at Intervals](#) (on page 343)

## Surface Cut/Fill Grid Options

Use these options to create cut/fill grids and elevation grids for surfaces. They are available in the **Create Cut/Fill Grid** command pane. These properties and others can be edited in the **Properties** pane.

## Options

<b>Grid type</b>	<p><b>Cut/Fill</b> - Select this to create a grid of values labeling the elevation differences between two surfaces for cut/fill operations.</p> <p><b>Elevation</b> - Select this to create a grid of values labeling the absolute elevations of a single surface.</p>
<b>Initial surface</b>	Select the surface that reflects the state of topography before the surface you select as the final surface below.
<b>Final surface</b>	Select the surface that reflects the state of topography after the surface you select as the initial surface above.
<b>Surface</b>	For an elevation grid, select the surface for which you want to see absolute elevations.
<b>Layer</b>	Select the layer on which you want the grid to reside.
<b>Text style</b>	Select a style that controls the text font, font style, justification, and size for the grid annotations.
<b>Grid spacing</b>	Type a value for the uniform interval at which the grid lines or tick marks for the measurements will be spaced.
<b>Grid style</b>	Select an option for how the location of each measurement will be denoted.
<b>Decimal precision</b>	Select the number of decimals to use in the measurement.

## Related topics

- ❑ [Create a Surface Cut/Fill Grid](#) (on page 346)

## Add Surface Materials

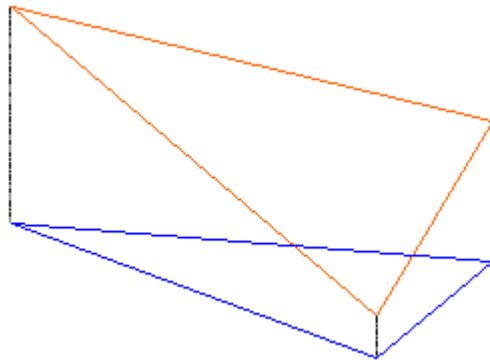
### Understanding Earthwork Volume Calculations

In the *Earthwork Report*, surface to surface (and surface to elevation) volumes are calculated by computing the isopach between the surfaces (or between the elevation and surface) using the prismatic method. First, each point is projected onto the other surface. Then, the corresponding elevation is interpolated, and the elevation difference is stored with the generated isopach point. The breaklines of both surfaces are used to generate isopach breaklines in a similar manner. Additional points are inserted into the isopach breaklines where they cross the surface triangles.

A Triangulated Irregular Network (TIN) is generated to represent the surface created by the isopach points and breaklines. The isopach represents the thickness, or difference in elevation, between the two surfaces. The software generates the surface triangles for the isopach by linking the isopach points, while honoring all breaklines and points common to both surfaces.

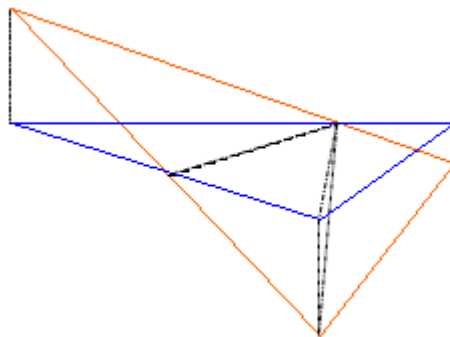
The triangulated isopach is used to determine the volumes by breaking the data for each triangle into the appropriate number of truncated vertical prisms. If the three points of a triangle all have positive or all have negative elevations (all fill or all cut) then a single triangular prism is present, as shown in case 1.

#### Case 1: All Excavation



If the isopach triangle has both positive and negative elevations (cut and fill), then the triangle is broken into additional triangular prisms such that each prism represents only cut or fill, as shown in case 2.

#### Case 2: Partial Excavation and Partial Fill



The volume of each prism is equal to the average height (i.e. isopach elevation) times the planimetric area. The total fill volume for the isopach is the sum of all positive prismatic volumes. The total cut volume is the sum of all negative prismatic volumes.

This method is considered more accurate than cross-section average end area or grid methods. The differences obtained between methods depend on the size of the grid or interval of cross-sections and the irregularity of the surfaces. Smooth surfaces may show differences of less than 1%. Some test cases have shown differences of as much as 10% when the cross-section interval is 15 meters (49.21 feet) and the terrain is rough. Mathematically, as the size of the grid or cross-section interval decreases to a very small value, the volumes of these methods agree with volumes computed from the isopach triangles.

**Note:** The accuracy of volume calculations is dependent on the accuracy of the data that is used and the surface that is created. If breaklines are not properly used to form a valid surface, good point data can produce bad volumes. If an insufficient number of points are used to describe a surface, then irregularities from the recorded data to the actual ground will produce volumes differing from field conditions.

**Note:** Duplicate points are re-layered to layer '0' during the linking process. Duplicate points are often created when computing the isopach surface, especially if a boundary line is used. In order to conserve memory, these points should be erased as required.


### Related topics

- ❑ [Define Materials for Earthwork Reports](#) (on page 351)
- ❑ [Run an Earthwork Report](#) (on page 353)

## Define Materials for Earthwork Reports

Define materials that you want to use in earthworks volume calculations and reports. You can enter two of three values for the shrinkage, haul bulkage, or haul compaction of materials to calculate the third value.

### To create a material:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Surface > Define Materials**.

The *Materials* dialog displays.
2. Click the option for the value that you want to calculate in the **Calculate** group. The value you select is unavailable in the calculation table.
3. Click the option for the format in which you want to view calculations in the **Display as** group.
4. Type the name of a material you want to add in the **Material** column.
5. Type percentages or factors in the **Shrinkage**, **Haul Bulkage**, and/or **Haul Compaction** boxes, as required.
6. Press **[Tab]** to add another material, if needed.
7. Click **OK**.

### Related topics

- ❑ [Calculate Volumes Using the Earthwork Report](#) (see "Run an Earthwork Report" on page 353)
- ❑ [Materials Options](#) (on page 352)

## Materials Options

Use these options to define materials to use in earthworks volume calculations and reports. They are available in the **Define Materials** dialog. For an example, see below the table.

### Options

#### Calculate

**Shrinkage** - Select to calculate the shrink or swell from cut or the natural bed to fill.

= (haul bulkage x haul compaction)

**Haul Bulkage** - Select this to calculate the swell from the natural bed to the loose condition.

= (shrinkage / haul compaction)

**Haul compaction** - Select this to calculate the shrink to compacted in place condition from the loose condition.

= (shrinkage / haul bulkage)

#### Display as

**Factor** - Select this to view the calculation as the number by which the volume is multiplied or divided.

**Percentage** - Select this to view the calculation as a percentage (proportion multiplied by 100).

---

**Note:** To understand the relationship between factor and percentage in relation to shrinkage and haul bulkage, experiment by entering various numbers. For example:

---

- Shrinkage or haul compaction of 25% = a multiplication factor of 0.75

whereas

- Haul bulkage of 25% = multiplication factor of 1.25

#### Material

Type a descriptive name for the material to cut or fill.

#### Shrinkage %

Type the percent, or factor by which, the material will shrink when it is cut and then used as fill.

#### Haul bulkage %

Type the percent that, or factor by which, the material will swell when it is cut and hauled in loose condition.

#### Haul compaction %

Type the percent that, or factor by which, the material will shrink when it is cut and hauled in loose condition.

When entering values for material shrinkage, bulkage, and compaction, consult a standard soils engineering reference. **As an example**, an industry manual might show these values:



Material	Swell	Shrink (in negative %)
Earth (loam) – dry	35%	12%
Earth (loam) – wet (mud)	0%	20%

According to the sample reference, if you dig wet earth out of the ground, and pile it, or load it on a truck, it won't expand (0%). It will remain the same volume as it was in the ground. If you dig dry, compacted dirt out of the ground, and pile it, or load it on a truck, its volume will expand about 35%.

If you dig wet earth out of the ground, use it as fill on the site, and compact it, it should shrink by 20%. The water will be squeezed out as it's compacted. If you take dry earth and do the same, the difference between its volume will be about 12% less.


### Related topics

- ❑ [Calculate Volumes Using the Earthwork Report](#) (see "Run an Earthwork Report" on page 353)
- ❑ [Define Materials for Earthwork Reports](#) (on page 351)

## Run an Earthwork Report

Use the **Earthwork Report** to calculate volumes based on a single surface, or the comparison of two surfaces.

### To generate a volume report:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Reports > Earthwork Report**.

The **Earthwork Report** command pane displays.
2. Select the option for the type of surface report you want to generate in the **Report type** group. Options in the steps below will vary based on the report type you select.
3. Select the surface you want to report on, or the first surface to compare, in the **Surface** list. The classification for the surface displays below the box.
4. Select the second surface to compare in the **Final** list, if applicable.
5. Pick a point in a graphic view, right-click for options, or type a value in the **Elevation** box, if applicable.
6. Pick lines to define the perimeter in the **Boundary** box, or click **Options** for more selection methods.

7. To account for materials in the calculations, check **Use in calculation** in the **Materials** group.


**Note:** Before using this option, you need to have defined surface materials in the **Define Materials** dialog.

8. Select the material at the site in the **Native** list.
9. Select the material that will be brought to the site in the **Borrow** list.
10. Select a type in the **Volume Breakdown** group, specifying a value by picking in the graphics window or typing a value, if necessary.
11. Click **OK**. The report displays in your default Internet browser.

### Related topics

- ❑ [Define Materials for Earthwork Reports](#) (on page 351)
- ❑ [Earthwork Report Options](#) (on page 354)
- ❑ [Understanding Earthwork Volume Calculations](#) (on page 349)

## Earthwork Report Options

Use these options to configure what your earthwork report shows. They are available on the **Earthwork Report** command pane. The  icon on the command pane's toolbar gives you quick access to the **Define Materials** dialog, where you can define materials to use in earthwork report calculations.

## Options

### Report Type

**Stockpile/Depression** - Select to generate a comparison of the volume formed by comparing one surface to a temporary surface formed by the edge points of the surface

A **Stockpile/Depression Report** uses only one material in the report.

**Surface to Surface** - Select to generate a comparison between two surfaces

**Surface to Elevation** - Select to generate a comparison of the volume formed by comparing one surface to a flat plane defined by the user giving its elevation

---

**Note:** You can set the name for the difference between surfaces in the **Computational Settings > Surface** section of the **Project Settings** dialog.

---

### Select surface

#### Surface/initial

Select the surface on which you want to report. If you are comparing a surface to another surface or elevation, select the surface from the earlier phase of construction.

#### Final

If you are comparing a surface to another surface, select the surface from the later phase of construction.

#### Elevation

Pick a point in a graphic view, or type a value for the elevation to be compared to the specified surface.

#### Boundary

Select the perimeter of the volume you want to use in computations.

### Materials

#### Use in calculation

Check this to open the **Native** and **Borrow** boxes, if you are using materials in calculations.

#### Native

Select the material being excavated from the site, and often being reused as fill material on the site.

#### Borrow

Select the material that is being brought in from off-site to be used as fill material on the site. Borrow material can be the same as native material.

**Volume breakdown**

**Volume totals only** - Select this to report only the total difference between volumes.

**By depth increment** - Select this to report the volumes in increments. Pick two points in a graphic view, or type a value to specify the depth of the increment.

**By elevation interval** - Select this to report volume differences between specific elevations. Pick two points in a graphic view, or type a value to specify the distance between elevations.

**Index elevation** - Pick a point in a graphic view, or type a value to specify the elevation.

**Related topics**

- ❑ [Calculate Volumes Using the Earthwork Report](#) (see "Run an Earthwork Report" on page 353)
- ❑ [Define Materials for Earthwork Reports](#) (on page 351)

**Shrink 3D Faces**

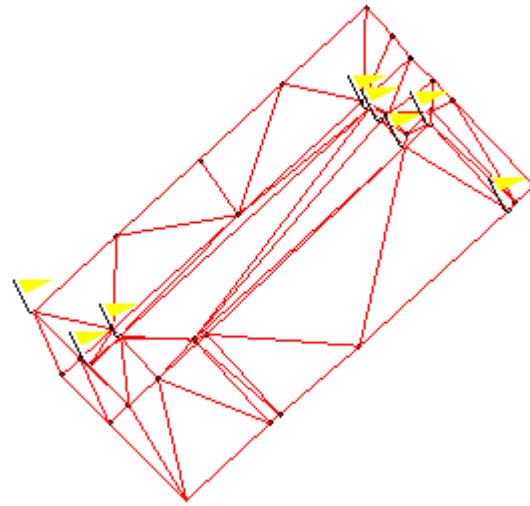
If you use a mesh of 3D faces to create a surface, any vertices that exist at the same horizontal location, but which have different elevations, will create vertical 3D faces in the resulting surface. You will know that you need to 'shrink 3D faces' when you see flags at surface vertices and messages in the **Flags** pane stating that "*The vertex has been ignored. It is directly above another vertex.*"

To fix this, 'shrink the 3D faces' to form closed, triangular breaklines slightly inset (.06 mm) from each selected face. These breaklines can then be used to create a surface that closely approximates the original mesh of 3D faces.

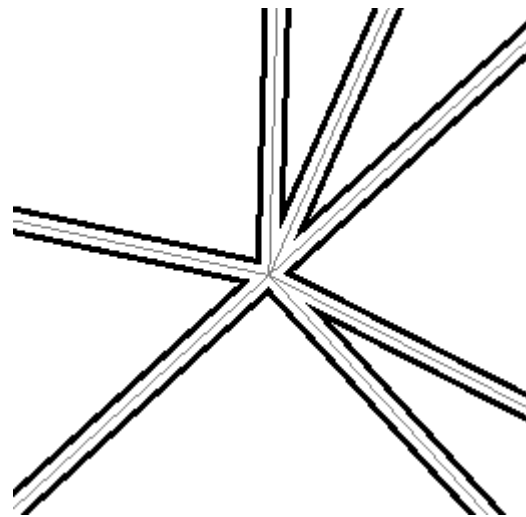
This can be helpful in forming a valid surface from data incorporating vertical faces or when coincidental 3D face vertices that have differing elevations at the same location.

**Note:** This command is primarily used to fix data from applications that allow points with different elevations at the same horizontal location.

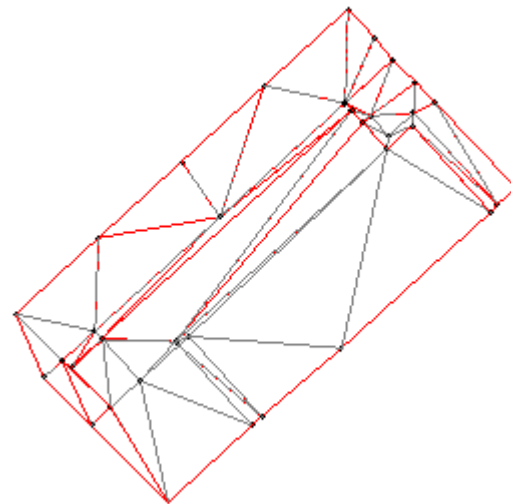
Initially, the surface is formed using vertical 3D faces with coincidental vertices, resulting in two elevations at these vertices. This data is not valid for forming a surface.



The non-vertical 3D faces are shrunk to create triangular breaklines (shown in black) with offset vertices which can be used to form the surface.



Using the new breaklines, the surface is properly formed with a single elevation at each vertex.



**To 'shrink 3D faces' for a surface:**


Follow these steps immediately after you receive one or more flagged messages that a vertex is directly above another vertex in a surface. If you cannot undo the surface creation because you have performed other commands in the interim, you will need to delete the surface and start with step 2 below.

1. Select **Edit > Undo Create Surface**. The surface disappears.
2. If the command pane is not already displayed, select **View > Command Pane**, or press **F12**. The **Command Pane** displays.
3. Click **Shrink 3D Faces** in the **All Commands** list, or type `shrink3dfaces` on the **Command** line at the top of the pane and press **[Enter]**. The **Shrink 3D Faces** command pane displays.
4. In the **Plan View**, pick the 3D faces that you want to use to form a surface
5. Select the layer on which you want the new breaklines to reside in the **Layer for breaklines** box.

**Note:** You will select the data on this layer to use when you create the surface. If necessary, create a new layer by selecting **<<New layer>>**.

6. Click **OK**. The breaklines are created within each selected triangular, 3D face.

**To create a surface from the newly created breaklines:**

1. Select the **View > View Filter Manager**. The **View Filter Manager** displays.
2. If necessary, click the  icon to collapse each group except **Layers**.
3. Uncheck the box for each group to hide the data.
4. In the **Layers** group, right-click the name of layer you created the 3D face breaklines on, and select **View Only This** from the context menu. All of the other layers in the group are unchecked so that only the breaklines are visible.
5. Select all of the breaklines in the graphic view, and select **Surface > Create Surface**.
6. Follow the instructions for [creating a surface](#) (see "Create a Surface" on page 323).

**Related topics**

- ❑ [Edit a Surface by Trimming Edge Triangles](#) (on page 336)
- ❑ [Edit a Surface by Adding and Removing Members](#) (on page 328)

## CHAPTER 14

# Work with Feature Data

## Understanding Feature Data

Features represent objects that surveyors might encounter as they collect survey data. Examples include trees, fences, gates, signs, utility poles, and buildings. After import and processing, the software can display symbols and line work that represent the real world objects. Features can be exported to other systems (for example, CAD packages) as necessary.

There are two basic types of features:

- A **point feature** is used to identify a single feature, such as tree or utility pole.
- A **line feature** is made up of two or more points that define a line, such as a fence or path.

A feature is identified in the software by a feature code string that is made up of one or more parts.

A feature code string for a point feature is made up of one or two parts:

- An alphanumeric code identifying the point feature itself (for example, "TREE")
- Optionally, a brief description of the point feature (for example, "West of house")

A feature code string for a line feature is made up of one to four parts:

- An alphanumeric code identifying the line feature itself (for example, "FENCE")
- Optionally, an alphanumeric instance identifier that is used for each point that makes up a single line (for example, "FENCE1" might define one fence; "FENCE2" might define a different fence)
- Optionally, an alphanumeric line control code that identifies the start or end of a line (for example, "START" might be used to identify the start of a fence)
- Optionally, a brief description of the line feature (for example, "Barbwire")

**Note:** If you delete a point used in a line feature, the position for the point is still used to define the line.

**Note:** You can [split](#) (see "Split Line Features" on page 365) line features in the software.

To make the feature code strings easier to read in the **Properties** pane and **Point Spreadsheet**, automatic formatting is used to differentiate the parts:

- The code itself is underlined (for example, "FENCE").
- The instance number and line control code are normal text.
- The brief description is in italics (for example "*Barbwire*").

So, a complete line feature code string might look like this: "FENCE1 START *Barbwire*"

Each feature code can also include one or more attributes that provide additional information about the feature. For example, a utility pole feature might have these attributes: utility type and owner, pole height, material, and condition.

Feature code types and attributes are defined and managed in the **Feature Definition Manager** software, which outputs a feature definition (.fxl) file that can be loaded on a field device. In the field, the surveyor can select from the feature library contained in the .fxl file the appropriate feature code to assign, add a brief description, and enter values for any attributes defined for the code. This same .fxl file is then used by the software for feature processing (see [Feature Code Processing Settings](#) (on page 157)) after the feature codes have been imported.

**Note:** If a surveyor manually enters a feature code not represented in an .fxl file, or the .fxl file used on the field device is not specified for the project, the feature code string will display in the software, but it will be ignored during feature code processing and reported in the **Feature Code Processing Report**.

Feature codes are displayed in the **Properties** pane when points to which they are assigned are selected. You can edit or delete feature codes as necessary, or add additional codes. You can also merge multiple line features into a single line feature or split a single line feature into multiple line features.

### Related topics

- [Workflow for Feature Data](#) (on page 361)



- ❑ [Feature Code Processing Settings](#) (on page 157)
- ❑ [Enter, Edit, and Delete Feature Code Strings](#) (on page 363)
- ❑ [Process Feature Codes](#) (on page 367)

## Workflow for Feature Data

To work with features data, it is recommended that you use the following steps:

1. Specify the appropriate feature definition (.fxl) file and other feature project settings in the **Project Settings** (see "Feature Code Processing Settings" on page 157) dialog.
2. [Import](#) (see "Import Data" on page 193) the data file containing features data into your project.

**Note:** The **Edit Point > Description** field used in Survey Pro and Field Surveyor 2.0 (not Field Surveyor 1.x) contains the feature code imported into the software and displayed in the **Feature code** field.

3. If feature codes were processed on data import, review the resulting **Feature Code Processing Report** to determine if all codes were processed correctly.

**Note:** You use the **Project Settings** (see "Feature Code Processing Settings" on page 157) dialog to specify whether or not to process feature codes on data import.

4. Review the **Plan View** and **Point Spreadsheet** to ensure the survey was correctly performed and feature codes were correctly entered.

**Note:** If feature codes are not displayed in the **Plan View**, select **View > View Filter Manager** and ensure the **Show feature code** option is selected in the **View Filter Manager** pane.

5. If appropriate, click **Surface > Create Surface** to ensure features are correctly included in the surface computation.
6. If necessary, make changes to any feature codes as described in [Enter, Edit, and Delete Feature Code Strings](#) (on page 363).
7. If necessary, [split](#) (see "Split Line Features" on page 365) any line feature into two line features by selecting the segment between the two points you want to separate.
8. If you made any changes to the feature codes in step 6, re-process the feature codes as described in [Process Feature Codes](#) (on page 367).
9. Review the resulting **Feature Code Processing Report** to determine if all codes were processed correctly.

### Related topics


- ❑ [Understanding Feature Data](#) (on page 359)

- ❑ [Feature Code Processing Settings](#) (on page 157)
- ❑ [Enter, Edit, and Delete Feature Code Strings](#) (on page 363)
- ❑ [Rules for Merging Feature Attributes](#) (on page 365)
- ❑ [Process Feature Codes](#) (on page 367)

## Feature Code Processing Settings

Use feature processing settings to configure how feature codes are processed.

### To specify feature processing settings:

1. Do one of the following:
  - Select **Project > Project Settings**.
  - Click the  icon on the toolbar.

The *Project Settings* dialog displays.
2. Select **Feature Code Processing** in the left pane.
3. Click each section and set the options as shown in the following table.
4. Click **OK**.

#### Options

##### General

**Process feature codes on import** **Prompt** - Select this option if you want to be prompted on whether or not to process feature codes during data import. If you select to process feature codes during import and a feature definition (.fxl) file has not been specified in the project settings, you will be prompted to specify it at the time of import.

**Yes** - Select this option if you want feature codes to be automatically processed during data import without prompting you. If a feature definition (.fxl) file has not been specified in the project settings, you will be prompted to specify it at the time of import.

**No** - Select this option if you never want feature codes to be processed during data import.

##### Decimal precision

Specify the number of decimals to display with a numeric attribute (real number) when no feature definition (.fxl) file is specified.

##### Processing

**Feature definition file** Specify the feature definition (.fxl) file you want to use to process feature codes in the project. This is required if you want to process feature codes.

### Related topics

- ❑ [Choose Project Settings](#) (on page 147)

- ❑ [Understanding Feature Data](#) (on page 359)
- ❑ [Workflow for Feature Data](#) (on page 361)
- ❑ [Enter, Edit, and Delete Feature Code Strings](#) (on page 363)
- ❑ [Process Feature Codes](#) (on page 367)

## Enter, Edit, and Delete Feature Code Strings

If a feature code string has been assigned to a point, it is displayed in the **Properties** pane when the point is selected.

**Note:** Feature codes are also visible and editable in the **Point Spreadsheet** and **Occupation Spreadsheet**. They can be viewed, but are not editable in the **Plan View**.

You can edit or delete feature code strings as necessary. You can also assign additional code strings.


### To enter, edit, or delete a feature code string:

1. Display in the **Properties** pane the point with which you want to work.

If a feature code string has been assigned to the point you selected, it is displayed in the **Feature Code** box.

If you select multiple points, the value in the **Feature Code** box is \*Varies\*, indicating that different codes may be assigned to the selected points. Any changes you make to the code affect all of the selected points.

**Note:** The **Edit Point > Description** field used in Survey Pro and Field Surveyor 2.0 (not Field Surveyor 1.x) contains the feature code imported into the software and displayed in the **Feature code** field.

2. Click the  icon in the **Feature code** box. The **Feature Code Editor** dialog displays.

**Note:** If you know the feature code string you want to assign to the point, you can manually enter it directly in the **Feature code** box without opening the **Feature Code Editor** dialog. However, if you need to enter attribute values, you must enter them in the **Feature Code Editor** dialog. In addition, you can manually delete a feature code string from the **Feature code** box.

3. To delete a feature code string assigned to the point, delete it from the **Feature code** box.
4. To assign or edit a feature code string for a point when a feature definition (.fxl) file has not been specified for the project, type the new string, or edit the existing string, in the **Feature code** box.

You can add additional feature code strings to the point as necessary.

5. To assign or edit a feature code string for a point when a feature definition (.fxl) file has been specified for the project, do the following:

- a. If necessary, delete any of the existing code string from the *Feature code* box
- b. Select a feature code (for example, "FENCE") from the *Codes* list and click **Add Code**. The feature code is displayed in the Feature code box.
- c. If the feature code represents a line feature, type the alphanumeric instance of the line feature immediately before or after the code (no spaces) in the *Feature code* box (for example, "FENCE1").
- d. If a line feature requires a control code (for example, "START"), select it in the *Codes* list and click **Add Code**. The control code is displayed in the *Feature code* box.
- e. If appropriate, enter a brief description of the feature in the *Feature code* box at the end of the string (for example, "Barbwire").

When you change focus from the *Feature code* box, the feature code string is reformatted as follows:

- The feature code is underlined.
- The instance for a line feature, if entered, appears in regular text.
- The control code for a line feature, if entered, appears in regular text.
- The description, if entered, appears in italics.

For example, if the feature code is for the line feature "FENCE", the instance is "1", the control code is "START", and the description is "barbwire", the string would appear as "FENCE1 START *Barbwire*".

- f. If the feature code supports attribute values, ensure the feature code is selected in the *Details* list, then select attribute values for the code in the *Attribute* list.

If a feature code is changed from the raw data, any attributes associated with the original feature code are merged with the new feature code per specific rules. For more information, see [Rules for Merging Feature Attributes](#) (on page 365).

6. To add more feature codes to the point, repeats step 4 or 5 as necessary.

Regardless of the order in which you add additional feature codes, the codes are automatically arranged in alphabetical order in the *Feature code* box (with the exception of line feature codes).

7. Click **OK**.

After you have completed all feature code changes, you must [reprocess the feature codes](#) (see "Process Feature Codes" on page 367).

### Related topics

- [Understanding Feature Data](#) (on page 359)

- ❑ [Workflow for Feature Data](#) (on page 361)
- ❑ [Feature Code Processing Settings](#) (on page 157)
- ❑ [Rules for Merging Feature Attributes](#) (on page 365)
- ❑ [Process Feature Codes](#) (on page 367)

## Rules for Merging Feature Attributes

If a feature code is changed from the imported data, the attributes of the imported data feature code and the attributes that are the same for the imported and new feature code are copied to the new feature code. For example, if the feature code is changed from "FENCE" to "MH", the raw attributes of "FENCE" and the attributes that are the same for "FENCE" and "MH" are copied to "MH".

In the case of multiple feature codes being assigned to a point, the order of the codes before and after editing determine where the attributes in the raw feature code are merged. For example:

- If "FENCE TREE" is changed to "MH TREE", the imported attributes of "FENCE" and attributes that are the same for "FENCE" and "MH" are copied to "MH".
- If "FENCE TREE" is changed to "MH", the attributes of "FENCE" merge to "MH".
- If "FENCE POST TREE" changes to "MH VALVE", the attributes of "FENCE" merge into "MH" and the attributes of "POST" merge to "VALVE".

When attributes are merged to the new feature code and the definition does not recognize it, the attributes become unknown attributes. You can delete these unknown attributes in the *Feature Code Editor* (see "Enter, Edit, and Delete Feature Code Strings" on page 363) dialog.

### Related topics

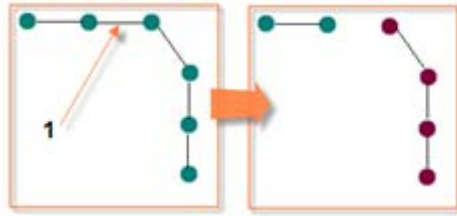
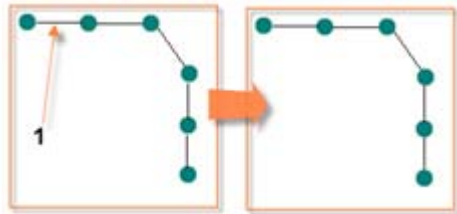
- ❑ [Understanding Feature Data](#) (on page 359)
- ❑ [Workflow for Feature Data](#) (on page 361)
- ❑ [Enter, Edit, and Delete Feature Code Strings](#) (on page 363)

## Split Line Features

You can split a single line feature into two line features by selecting the segment between the two points you want to separate, as shown in the following diagram (example 1). Note that you cannot select a beginning or ending line segment for the split (example 2).


Key to example diagrams:

- Green points ● represent the first line feature selected for the split.
- Maroon points ● represent the new line feature created after the split.
- 1 shows the line feature segment selected for the split.

**Example 1:****Example 2:**

**Note:** Before performing this procedure, ensure that features codes have been [processed](#) (see "Process Feature Codes" on page 367).

**To split a line feature:**

1. Do one of the following:
  - Select **Survey > Feature Coding > Split Line Feature**.
  - Click the  icon on the toolbar.

The *Split Line Feature* pane displays.
2. Click in the **Select** box.
3. In the *Plan View*, select the segment between the two line feature points you want to separate. The segment is removed.
 

You cannot select a beginning or ending line segment for the split (see example 2).
4. If necessary, select additional line features in the *Plan View* to split them.
5. When you are done, click **Close** in the *Split Line Feature* pane.

**Related topics**


- ❑ [Understanding Features Data](#) (see "Understanding Feature Data" on page 359)
- ❑ [Workflow for Features Data](#) (see "Workflow for Feature Data" on page 361)
- ❑ [Enter, Edit, and Delete Feature Code Strings](#) (on page 363)

## Process Feature Codes

You must process feature codes to display features correctly in graphic views or to export. You can process feature codes during data import (depending on your *Feature Code Processing settings* (on page 157)), or you can process or reprocess them after import. For example, you might have processed feature codes during data import but you have since made changes to the codes in the software and need to reprocess them. Or, you might have added points to the project that include features you want to process.

### To process feature codes:



1. Do one of the following:

- Select **Survey > Feature Coding > Process Feature Codes**.
- Click the  icon on the toolbar.

The *Process Feature Codes* pane displays listing one or more point sources from which you can select to process feature codes. A point source is either an imported file or a *Keyed in Block* (points manually added to the project). If you select multiple sources, they are processed separately.

2. In the *Select point source(s) to process* list, check the box for each source containing points you want to process.

3. Optionally, do either of the following:

- Click the  icon to display the *Report Options* (on page 438) pane. This allows you to select report options for the *Feature Code Processing Report* before generating the report.
- Click the  icon to display the *Project Settings* dialog to change *Feature Code Processing settings* (on page 157).

**Note:** This button is enabled only when an imported file is selected as a point source in the list.

4. Click **Process** to process feature codes in the selected source(s).

If no feature definition (.fxl) file is specified in the project, you will be prompted to specify it at this time.

If you have previously processed the features codes in the project, reprocessing will delete the previous results and replace them with new results.

5. After processing, review the *Feature Code Processing Report*, which provides a summary of the process.

The report displays on the *Feature Code Processing Report* tab. Use the tool bar located above the report to navigate, print, or save the report.

### Related topics

- ❑ [Understanding Feature Data](#) (on page 359)
- ❑ [Workflow for Feature Data](#) (on page 361)
- ❑ [Feature Code Processing Settings](#) (on page 157)
- ❑ [Enter, Edit, and Delete Feature Code Strings](#) (on page 363)

## Export Geodatabase Files (.xml)

Export feature data contained in your project to a geodatabase XML file from which the data can be imported into an Environmental Systems Research Institute, Inc. (ESRI) geographic information system (GIS).

Click the **GIS** tab in the **Export** command pane to access the **Geodatabase XML** exporter.

**Note:** The geodatabase XML format represents ESRI's most current open mechanism for information interchange between geodatabases and other external systems and is a replacement for the earlier shapefile (.shp) spatial data format.

### Related topics

- ❑ [Export Data](#) (on page 441)
- ❑ [Export Data in a Custom Format](#) (on page 448)

## Feature Definition Manager Utility

The **Feature Definition Manager** is a standalone utility that gives you the ability to create and manage feature libraries (.fxl files) for feature code processing and GIS attribute data collection. A feature library is a collection of features with codes and attributes that describe them, as well as line control codes that modify how the features relate.

The **Feature Definition Manager** comes with a default library of features with predefined attributes. This library provides a good starting point for feature coding. As you create new features and edit existing ones, the library will become suited to the specific needs of your projects.

Feature coding in the editor enables you to:

- Make detailed data collection in the field more efficient and consistent by controlling how features and attributes can be captured. Setting parameters for what you can and must enter ensures data integrity and completeness.
- Add symbols and annotations to feature-coded field data so that the information can be presented in a more visual format.
- Connect points to define line features, such as pavement or building edges, or the centerlines of ditches or fences. Line control codes give you the power to add new points automatically, and add lines, curves, and arcs between points.

If you are working with surfaces, coded features also let you:



- Define the breaklines of a surface.
- Control how surfaces are formed by specifying which points should be used, and which lines should act as breaklines. Surfaces can be modified by moving points to specific layers based on their feature codes.

**Note:** The *Feature Definition Manager* has its own help system. While in the *Feature Definition Manager*, select **Help > Contents**, or press [F1].

### Related topics

- [Understanding Feature Data](#) (on page 359)
- [Workflow for Feature Data](#) (on page 361)

## CHAPTER 15

# Prepare Data for Site Positioning Systems

HEAVY CONSTRUCTION 

## CHAPTER

## Prepare Data for Site Positioning Systems

HEAVY CONSTRUCTION 

Trimble® Business Center - Heavy Construction Edition and Trimble Site Controller Software (SCS) are components within Trimble's site positioning systems (SPS). In Trimble Business Center, you can prepare site data, design data, and work orders for site controllers with SCS. Using these controllers, you can measure material volumes, monitor grades and laid material thickness, and perform other site-related tasks, such as point, line, roadway, and surface stakeout. For more information, see [Understanding SPS and SCS](#) (on page 372).

Office Synchronizer software manages the exchange of site data, design data, work orders, and work order results between your computer and site controllers. This exchange process is known as 'synchronization'. Data, work orders, and results that need to be synchronized are stored in a file structure on your hard drive, the location of which is defined in the **Synchronizer root folder** box in the **Options** dialog. For more information, see [Understanding Sites, Designs, Work Orders, and Synchronization](#) (on page 373)

**Note:** For simplicity, the synchronizer root folder's file structure is referred to as the '**data synchronization area**' throughout this documentation. .

The default path for the data synchronization area depends on your operating system:

- In Windows® XP or earlier: *C:\Documents and Settings\((user name))\My Documents\Trimble Sync*
- In Windows Vista™: *C:\Trimble Synchronizer Data*

You may, however, have placed your data synchronization area in a different location or named it differently.

**Note:** The data synchronization area is meant for data exchange only. You should not directly modify its structure or contents, but manage all your construction project files in a separate location.

Following is the general workflow for preparing data for SPS in this software.

### Workflow for preparing data for Trimble site positioning systems:

1. Create or [import](#) (see "Import Data" on page 193) the relevant roadway, surface, CAD, point, and other construction data into your project.
2. Create a [site](#) (see "Workflow for Creating a Site" on page 377) or add an [external site](#) (see "Workflow for Using an External Site" on page 378), and add the site data.
3. Create one or more [designs](#) (see "Create a Design" on page 393), or add one or more [external designs](#) (see "Add an External Design" on page 394), and add design data related to the site.

**Note:** The site and its related designs are created as objects within your project.

4. Create one or more [work orders](#) (see "Create a Work Order" on page 409) to be loaded onto site controllers, instructing your field crews what tasks need to be performed on site.

**Note:** Work orders are accessible from your project, but they are not truly objects within it. Work orders reside in the synchronization area and on the controllers to which they are assigned.

5. Graphically [view](#) (see "View a Work Order's Data Graphically" on page 416) the data associated with each work order.
6. Synchronize controllers running SCS, such as SCS900, sending your site data, design data, and work orders to the controller used by the designated construction crew, and retrieving data resulting from their prior operations.
7. While crews are actively using the controllers to complete work orders in the field, prepare new data and instructions for them. When the crews next synchronize their controllers, the additional information or revisions that you have prepared are automatically updated on their controllers, and any new data from the field is automatically updated on your computer.

8. Import the completed work orders and [results](#) (see "Import Work Order Results" on page 422) back into your project.
9. Check the data by running [alignment](#) (see "Run an Alignment Geometry Report" on page 298), [surface](#) (see "Run a Surface Report" on page 339), and [work order](#) (see "Run a Work Order Report or Summary" on page 423) reports.
10. [Compress and email](#) (see "Compress and E-mail SCS Files" on page 421) site data, design data, and work orders to colleagues or site controllers.

### Related topics

- ❑ [Manage a Site and Site Data](#) (on page 377)
- ❑ [Manage Designs and Design Data](#) (on page 393)
- ❑ [Manage Work Orders and Results](#) (on page 406)

## Understanding SPS and SCS

### HEAVY CONSTRUCTION

**Trimble®** site positioning systems (SPS) are comprehensive construction positioning solutions comprised of:

- positioning hardware (GPS base, rover, and radios or total station)
- site controllers (typically hand held or tablet computers)
- **site controller software (SCS)**

In general, SPS provides non-surveyors the ability to perform construction measurement and stakeout tasks associated with site preparation, heavy/highway construction, mining, and landfill operations, etc.. The overall system, comprised of the above components, allows contractors to track, report, validate, and control site measurement and positioning workflows from receipt of initial designs to project completion.

SPS also gives site foremen, supervisors, grade checkers, and site engineers the tools they need to help run machines on site, reducing downtime, and resolving issues as they arise, without office assistance.

Using SPS, contractors employ GPS antennas, GPS receivers, total stations, and hand held site controllers developed specifically for heavy/highway construction applications for measurement and stakeout activities on the job site.

SCS is software that runs on a site controller, which is typically a hand held device. Contractors can use software, such as SCS900, to complete a full range of construction tasks, including:

- Performing initial site measurement and verification of original ground levels
- Measuring site features
- Checking finished grade and laid material thickness
- Collecting topographic site data and computing material stockpile volumes
- Staking out points, lines, alignments, planes, surfaces, complex roadways, side slopes, and catch points
- Performing as-built site measurements

SCS helps contractors efficiently control and quantify site operations so they do not need to rely on a contract surveyor for their site measurement and stakeout requirements.

## Understanding Sites, Designs, Work Orders, and Synchronization

### HEAVY CONSTRUCTION

Before preparing data for Trimble® site positioning systems (SPS) within this software, it will be helpful to familiarize yourself with the concepts below. For additional information, see [Understanding SPS and SCS](#) (on page 372).

### Project

Work in Trimble Business Center is structured in projects, which are analogous to 'construction projects' in the real world, i.e. the work performed in fulfilling a construction contract. The data associated with a project is stored within a project file (\*.vce).

### Site

A site is an object in your project that contains site settings and site-specific data, such as a site calibration, the related control points, and a background map. A project's site represents what is commonly referred to as the 'job site', i.e., the property or physical location at which the improvements associated with a construction project will be made. In creating a site, you designate which portions of the data in your project need to be made available to field crews in the background map, and therefore sent to the site controllers that you expect to be employed on that site. The background map can be used by crews in the field to identify specific areas where construction work is to be done at the job site, and other area to be avoided.

**Note:** It is important to give each site a unique name. See the *Project and site naming* section in *Edit a Site's Name and Settings* (on page 383).

Each project can contain only one site. A site can reside in your project without being assigned to any controllers. If you choose to assign a site to one or more controllers, you must designate the path to the data synchronization area in advance. When you assign a site to one or more controllers, its settings and data are automatically copied to appropriate locations within the folder structure of the data synchronization area (the folder structure located beneath the synchronizer root folder), in the forms in which they will be used on the designated controllers in the field.

Each instance of the site, as it resides on every controller's representation within the data synchronization area, is associated with the project file within which it is defined.

### **External Site**

External sites are those that exist in the data synchronization area, but not in your current project. You can add an external site to your project to make its site data and related designs and design data available within the project. Then you can edit the site's settings or data.

When you add an external site to your project, its related designs are created as objects within the project and appear in the Project Explorer. If you add an external site that resides on multiple controllers to your project, all of the designs for the site on the various controllers appear in the Project Explorer.

### **Design**

A design is an object in your project that contains design data, such as a surface model, stakeout points, and a foreground map representing features of the design. In creating a design, you designate which portions of the data in your project need to be available to field crews in regard to the subject matter of that design, and therefore sent to designated site controllers. Design data is used by crews in the field to stakeout or check designed features, alignments, and surfaces.

After creating a site or adding an external site, one or more designs can be created in or added to the site. These designs reside in the association with single site that resides in the project. Designs can be assigned to any of the controllers to which the site has been previously assigned.

### **External Design**

External designs are those that exist in the data synchronization area, but not your current project. You can add external designs to make their design data part of your project. Then you can edit the design data.

### Work order

A work order contains instructions for a field crew about the tasks they need to perform at a site, as well as tolerances and other site controller settings associated with those tasks. A work order that requires design data in order to be carried out, will contain a reference to the applicable design. Work orders are used to direct and track the progress of construction improvements at the job site.

Multiple work orders can be created in association with a site. Even though they appear in the Project Explorer, work orders, unlike a site and designs, are not actually objects within your project. They simply reside in the data synchronization area and on the site controller to which they are assigned. A work order can be created without being assigned to a particular construction crew's site controller. It can later be assigned to one and only one site controller. Each of those work orders appears in the Project Explorer beneath the site listing.

### Site Controller

A site controller is a field device, also known simply as a 'controller', that runs site controller software (SCS) within a Trimble site positioning system (SPS). Site controllers used by construction crews are typically hand held computers. The site controller software can also be operated on a tablet or notebook computer in the field. Those devices may commonly be used by construction supervisors, foremen, etc., who may operate them from within a truck cab, as they drive the site. Site data, design data, and work orders are transferred back and forth between office computers and site controllers. After using a controller to collect data in the field, the measurements are transferred from the controller to an office computer.

### Data Synchronization Area/Synchronizer Root Folder


Synchronization is the process of making files on a site controller match the files in the data synchronization area (the folder structure located beneath the synchronizer root folder) used by Trimble Business Center - Heavy Construction Edition, and SCS Data Manager. To be in a state of synchronization, a file with the same creation/modification date must appear on both the controller and in the synchronization area. Files are synchronized so that office staff using this software and field crews using the SCS are always using the same (or most current) data.

The data synchronization area is a folder structure (installed by the Office Synchronizer program on the office computer's hard drive) that contains site settings and data, design data, and work orders in a file structure that the Trimble Site Controller Software can use. Office Synchronizer uses the data synchronization area to exchange files between the office software and the site controller software so that they are synchronized.

**Note:** The data synchronization area is meant for data exchange only. You should not directly modify its structure or contents, but manage all your construction project files in a separate location.

### Conflict and Conflict Resolution

A conflict is triggered by a difference between the site or design data in your project and data for the same site or design in the data synchronization area. In addition, a conflict could arise from a difference between data for the same site or design on multiple controllers, or the introduction of new data on a controller.

A conflict is indicated by an active **Resolve** button with an  icon in the command pane. The icon is a reminder that you need to resolve the inconsistency by choosing which data to use in both your project and on site controllers. Depending on your choice, either your project data or the data in the data synchronization area is updated, thereby resynchronizing them.

## SCS Data Manager

### HEAVY CONSTRUCTION

The SCS Data Manager is a program that lets you create sites, organize site and design data, write work orders, and assign the data and work orders to controllers in preparation for Trimble® site controller software (SCS).

SCS enables you to measure material volumes, monitor grades and laid material thickness, and perform other site-related tasks, such as roadway point, line, and surface stakeout.

**Note:** Most of the SCS Data Manager's functions have been embedded in the **SCS Site Manager** in this software.

#### To open the SCS Data Manager:

Do one of the following:

- Select **SCS > SCS Data Manager**.
- Click the  icon on the toolbar.

#### Related topics

- ❑ [Manage a Site and Site Data](#) (on page 377)
- ❑ [Workflow for Creating a Site](#) (on page 377)
- ❑ [Workflow for Using an External Site](#) (on page 378)



## Manage a Site and Site Data

HEAVY CONSTRUCTION 

Use the *SCS Site Manager* to create a site to be used by a Trimble® site positioning system. Then, prepare site data, and assign the data to the specific site controllers to be used on the job site. Use the *SCS Design Manager* to prepare designs, and use the *Work Order Editor* to create work orders.

For additional information, see [Understanding Sites, Designs, Work Orders, and Synchronization](#) (on page 373).

### Related topics

- ❑ [Workflow for Creating a Site](#) (on page 377)
- ❑ [Workflow for Using an External Site](#) (on page 378)
- ❑ [Create a Site](#) (on page 379)
- ❑ [Manage Designs and Design Data](#) (on page 393)
- ❑ [Manage Work Orders and Results](#) (on page 406)

## Workflow for Creating a Site

HEAVY CONSTRUCTION 

Prepare site data for field crews. Then, assign the site and its data to specific site controllers so that when the crews synchronize their controllers, the correct data goes to the designated controllers. Here is the general workflow:

1. Open or create a project within which to manage the data associated with a construction project and its job site.
2. [Import](#) (see "Import Data" on page 193) any CAD or survey data that you want field crews to access at the site or from which you will derive design data.
3. [Create a site](#) (on page 379) in the *SCS Site Manager*, and (optionally) assign it to one or more specific site controllers.

The site, which will contain specific site-level project data associated with the use of a Trimble site positioning system, will be created in your project. As that site is subsequently assigned to a particular controller, an external folder and its data will be created within the data synchronization area, ready to be placed on that controller the next time it is synchronized.

**Note:** Each project can contain only one site.

4. Review settings for the site.
5. Select and [add control points](#) (see "Add or Remove Control Points" on page 386) from your project data to the control point file included in the site data, as needed.

6. Select and [add objects](#) (see "Add or Remove Background Map Objects" on page 387) in your project data to the background map. The site, site data, and settings are written to the data synchronization area so they can be transferred to assigned controllers when those devices are synchronized.

**Note:** To synchronize a controller using the Office Synchronizer utility, see [Office Synchronizer](#) (on page 219) and [Prepare to Connect a Field Device](#) (on page 217).

### Related topics

- ❑ [Create a Site](#) (on page 379)
- ❑ [Workflow for Using an External Site](#) (on page 378)

## Workflow for Using an External Site

### HEAVY CONSTRUCTION

If you have already created an external site (that resides in within the data synchronization area) using SCS Data Manager, Terramodel, or some other means, you can use that site data within a project as a site within this software. Then, you can edit the site data by adding points and objects from your project data, if needed. Here is the general workflow:

1. Open or create a project within which to manage the site.
2. [Add the external site](#) (see "Add an External Site" on page 380) using the **SCS Site Manager**. This makes the site part of your current project.

**Note:** Each project can contain only one site.

3. Select and [add control points](#) (see "Add or Remove Control Points" on page 386) in your project data to the control points included in your site data, if needed.
4. Select and [add objects](#) (see "Add or Remove Background Map Objects" on page 387) in your project data to the background map included in your site data, if desired.
5. [Resolve any conflicts](#) (see "Resolve a Conflict in Site Settings" on page 388) between site settings and site data in your project and those in the data synchronization area.
6. Check the controller assignment, and [assign the site](#) (see "Reassign a Site to Controllers" on page 388) to other controllers as needed. The site, site data, and settings are written to the synchronization area so the assigned controllers will be updated when they are synchronized.


**Note:** To synchronize a controller using the Office Synchronizer utility, see [Office Synchronizer](#) (on page 219) and [Prepare to Connect a Field Device](#) (on page 217).

## Related topics

- ❑ [Add an External Site](#) (on page 380)
- ❑ [Workflow for Creating a Site](#) (on page 377)


## Create a Site

### HEAVY CONSTRUCTION

If your project does not already contain a site, create one so you can assign site data to one or more controllers for use in the field. When you create a site, it is stored in the current project, and appears in the *Project Explorer*, indicated by the  icon. A site is essentially a container in which you can designate portions of project data related to the site to be available to field crews employing a Trimble site positioning system.

**Note:** Each project can contain only one site.

#### To create a site:

1. Do one of the following:
  - Select **SCS > SCS Site Manager**.
  - Click the  icon on the toolbar.

The *SCS Site Manager* displays.

2. Select **Create new site**, and type a unique name in the *New site name* box (See the *Project and site naming* section in *Edit a Site's Name and Settings* (on page 383)).
3. Check the box for each controller that you want to assign the site to in the *Controllers* group.

**Note:** You can create a site without assigning any controllers, but the site and site data cannot be saved to, or used on, controllers until you do so. The site and site data are saved in your project until controllers are assigned in the future.

4. Click **OK**. The *SCS Site Manager* command pane displays, and the site appears in the *Project Explorer*. Start at step 3 in [Edit a Site's Name and Settings](#) (on page 383).

**Note:** Settings for new sites are inherited from *Project Settings*. If these settings are not recognized by the assigned controllers, the *Unsupported Project Settings* dialog displays. You must [resolve unsupported site settings](#) (see "Resolve a Conflict in Site Settings" on page 388) before creating a site.

#### To delete a site:

1. Select the site in the *Project Explorer*.

2. Press **[Ctrl] + [D]**. The site is removed from the project and disappears from the *Project Explorer*.

**Note:** If you delete a site in this software, it is removed from the project, but not from the data synchronization area. If you delete a site from the data synchronization area using SCS Data Manager or Windows® Explorer, it remains in the project, but assignments to any controllers are lost.

### Related topics

- ❑ [Add an External Site](#) (on page 380)
- ❑ [Edit a Site's Name and Settings](#) (on page 383)
- ❑ [Site Options](#) (on page 381)
- ❑ [Workflow for Creating a Site](#) (on page 377)


## Add an External Site

### HEAVY CONSTRUCTION

External sites are those that exist in the data synchronization area, but not in your current project. Add external sites to make their site data and associated designs and design data part of your project. Adding an external site makes it, and its designs and work orders, appear in the *Project Explorer*. Then you can edit the site's settings or data.

**Note:** Each project can only contain one site.

#### To add an external site:

1. Do one of the following:
  - Select **SCS > SCS Site Manager**.
  - Click the  icon on the toolbar.

The *SCS Site Manager* displays.

2. Select **Add external site**, and select a site in the *External site* list.

**Note:** The *External site* list shows all of the sites found in the data synchronization area on your hard drive.

3. Click **OK**. The *SCS Site Manager* command pane displays, and the site appears in the *Project Explorer*. See [Edit a Site's Name and Settings](#) (on page 383).

### Related topics

- ❑ [Create a Site](#) (on page 379)
- ❑ [Edit a Site's Name and Settings](#) (on page 383)
- ❑ [Site Options](#) (on page 381)
- ❑ [Workflow for Creating a Site](#) (on page 377)

## Site Options

### HEAVY CONSTRUCTION

Use these options to edit a site's name and settings, and to add and edit site data, such as a calibration file, control points, and a background map. In addition, you can change the site controllers to which the site is assigned. The options are available in the **SCS Site Manager** command pane.

## Options



### Site settings

Click this to launch the [SCS Data Manager](#) (on page 376).


**Rename** - Click this to display the *Rename Site* dialog, where you can change the site name.

---

**Note:** Before you rename a site, read the *Project and site naming* section in *Edit a Site's Name and Settings* (on page 383).

---

**Units** - Click this to display the *Site Units* dialog where you can review project and controller-based settings for the site. To change any of the settings, click **Project Settings**.

**Resolve** - If there are settings conflicts, click this to display the [conflict resolution](#) (see "Resolve a Conflict in Site Settings" on page 388) dialog. The  icon indicates that there is a conflict.

### Site calibration

This shows either the date that an imported site calibration (.dc) was created or the date a site calibration was performed in this software.

**Resolve** - If there are conflicts, click this to display the [conflict resolution](#) (see "Resolve a Conflict in Site Calibrations" on page 390) dialog.

### Control points

**Select** - Click this to select/highlight the control points currently included in the site data.



- Click this to display the *Add/Remove Control Points* dialog, in which you can change the control points included in the site data.

**Resolve** - If there is a conflict with the control points, click this to display the [conflict resolution](#) (see "Resolve a Conflict in Control Points" on page 391) dialog.

### Background maps

**Select** - Click this to select/highlight the map objects currently included in the site data.



- Click this to display the *Add/Remove Background Map Objects* dialog, in which you can change the background map objects included in the site data.

**Resolve** - If there is a conflict with the background map, click this to display the [conflict resolution](#) (see "Resolve a Conflict in Background Maps" on page 392) dialog.

### Controllers

**Reassign** - Click this to display the *Reassign Site* dialog, in which you can change the controllers to which the site is assigned.

## Related topics

- ❑ [Edit a Site's Name and Settings](#) (on page 383)
- ❑ [Edit Site Data](#) (on page 384)

- ❑ [Reassign a Site to Controllers](#) (on page 388)
- ❑ [Workflow for Creating a Site](#) (on page 377)
- ❑ [Workflow for Using an External Site](#) (on page 378)
- ❑ [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)

## Edit a Site's Name and Settings

### HEAVY CONSTRUCTION

Review and change a site's name and units in preparation for use on controllers in the field. These are the common settings found in Trimble's site controller software (SCS) and Trimble Business Center - Heavy Construction Edition projects: Distance units, angular units, station format, and coordinate order.

### Project and site naming

You should give each of your project files a unique name so they uniquely pertain to a particular construction project and reliably avoid confusion with other projects. In addition, you must name each site uniquely because site controllers identify the project and its site by the site name alone; the project name is not available on the site controller. If two construction projects contain sites of the same name, and if those sites are each assigned to a number of controllers, there is no way for the user of the site controller in the field to determine which project the site data pertains to, other than through his familiarity with the data. In general, the name of a project and its site should be the same. If that is not feasible, try to derive a site name shortened from the project name. Shortening the site name may be warranted in light of the small screen size available on some site controllers.

### To edit a site's name and settings:


If you have just created or added a site, the **SCS Site Manager** command pane is displayed. Otherwise, do one of the following:

- Select **SCS > SCS Site Manager**.
- Click the  icon on the toolbar.

The **SCS Site Manager** command pane displays.

1. To change the site name, click **Rename** in the **Site Settings** group. The **Rename Site** dialog displays.
2. Type a unique name in the **New site name** box, and click **OK**.
3. Click **Units** to review the settings for the site. The **Site Units** dialog displays.

**Note:** Trimble Business Center supports a number of units of measure that are not supported by SCS. When using this software with SCS900, the units must be compatible.

**Note:** Changing a site's name or settings can cause it to become out of sync with the data synchronization area. A desynchronized site is indicated by an  icon in the Project Explorer. To resolve conflicts after you edit a site, click each **Resolve** button that is available.

To change the site calibration, control points, or background map objects included in your site data, see [Edit Site Data](#) (on page 384).

To change any of the controllers the site is assigned to, click **Reassign** in the **Controllers** group.

1. Click **Close**. The site name and settings are saved to the data synchronization area.

### Related topics

- ❑ [Reassign a Site to Controllers](#) (on page 388)
- ❑ [Site Options](#) (on page 381)
- ❑ [Resolve a Conflict in Site Settings](#) (on page 388)
- ❑ [Resolve a Conflict in Site Calibrations](#) (on page 390)
- ❑ [Resolve a Conflict in Control Points](#) (on page 391)
- ❑ [Resolve a Conflict in Background Maps](#) (on page 392)


## Edit Site Data

### HEAVY CONSTRUCTION

Review and edit your site data so it can be used on controllers at the job site. Prior to editing the site data, make sure that you have imported any point and CAD data that you want to make available to construction crews on site so you can add additional points and objects to the site data as needed. If you want to include a site calibration in your site data, perform the calibration in your project or import a .dc calibration file.

#### To edit site data:

If you have just created or added a site, the **SCS Site Manager** command pane is displayed. Otherwise, do one of the following:


- Right-click the site in the *Project Explorer*, and select **SCS Site Manager**.
- Select **SCS > SCS Site Manager**.
- Click the  icon on the toolbar.

The **SCS Site Manager** command pane displays.





1. If you have imported or performed a site calibration in your project, it is automatically added to the site data, as indicated by the **Last update** file date in the **Site Calibration** group (see [Include a Site Calibration](#) (on page 385)).

To highlight the control points that are included in the site data in graphic views, click **Select** in the **Control Points** group.

To change the control points that are included in the site data, click  (see [Add or Remove Control Points](#) (on page 386)). The **Add/Remove Control Points** dialog displays.

To highlight the background map objects that are included in the site data, click **Select** in the **Background Map** group.

To change the background map objects that are included in the site data, click  (see [Add or Remove SCS Background Map Objects](#) (see "Add or Remove Background Map Objects" on page 387)). The **Add/Remove Background Map Objects** dialog displays.

**Note:** Changing a site's data can cause it to become unsynchronized with the data synchronization area, creating conflicts. A desynchronized site is indicated by the  icon in the **Project Explorer**. To resolve conflicts after you edit a site, click each **Resolve** button that is available.

1. Click **Close**. The site data is saved to the data synchronization area.

### Related topics

- ❑ [Site Options](#) (on page 381)
- ❑ [Resolve a Conflict in Site Settings](#) (on page 388)
- ❑ [Resolve a Conflict in Site Calibrations](#) (on page 390)
- ❑ [Resolve a Conflict in Control Points](#) (on page 391)
- ❑ [Resolve a Conflict in Background Maps](#) (on page 392)

## Include a Site Calibration

### HEAVY CONSTRUCTION

To include a site calibration in your site data, simply [perform a site calibration](#) (see "Calibrate a Site" on page 184) in your project or [import a .dc calibration file](#) (see "Import Data Collector Files (.dc)" on page 196). Either method automatically adds a .dc file to your site data in the data synchronization area. The presence of a site calibration is indicated by the date in the **Site Manager's Site Calibration** group. If you add a site that you created and calibrated in SCS to your project, that calibration will also be reflected in the **Site Calibration** group.

Even if you do not perform or import a site calibration, the coordinate system and any datum, geoid, projection, and shift grid information that you specify in the **Coordinate System Settings** (on page 148) section of the **Project Settings** are automatically saved in the data synchronization area so they will be available when you open your site in SCS. If you calibrate your site or make other coordinate system changes in SCS, you will be prompted to [resolve the site calibration conflict](#) (see "Resolve a Conflict in Site Calibrations" on page 390) once you synchronize and reopen the site in this software.

## Add or Remove Control Points

### HEAVY CONSTRUCTION

Add control points to your site data to make them available on designated controllers. Control points are used by crews in the field to establish the positional accuracy of a site survey.


**Note:** You cannot add or remove control points if there is a conflict in site units. See [Resolve a Conflict in Site Settings](#) (on page 388).

#### To add/remove site control points:

If you have just created or added a site, the **SCS Site Manager** command pane is displayed. Otherwise, do one of the following:

- Select **SCS > SCS Site Manager**.
- Click the  icon on the toolbar.

The **SCS Site Manager** command pane displays.

1. Click  in the **Control Points** group. The **Add/Remove Control Points** command pane displays.
2. To add additional control points to the file, pick them in a graphic view, or click **Options** for selection options.
3. Click **Add**. The selected control points are added to the control point file.
4. To remove excess control points, pick them or click **Options**.
5. Click **Remove**. The points are removed from the site data.
6. Click **OK**. The control points are saved as a .csv file in the data synchronization area.

**Note:** If you are also using Omnistar® control points, they will reside with the site data in a file called *reference.csv* in the data synchronization area. This will not cause a conflict with your control points file.

- To check the control points included in the site data, click **Select** in the *SCS Site Manager*.

### Related topics

- [Edit a Site Name and Settings](#) (see "Edit a Site's Name and Settings" on page 383)
- [Resolve a Conflict in Control Points](#) (on page 391)

## Add or Remove Background Map Objects

### HEAVY CONSTRUCTION

Add a background map to your site data to make it available on designated controllers. Background maps are referenced by crews in the field to determine the location of features and boundaries at a site.


**Note:** You cannot add or remove background map objects if there is a conflict in site units. See [Resolve a Conflict in Site Settings](#) (on page 388).

### To add/remove background map objects:

If you have just created or added a site, the *SCS Site Manager* command pane is displayed. Otherwise, do one of the following:

- Select **SCS > SCS Site Manager**.
- Click the  icon on the toolbar.

The *SCS Site Manager* command pane displays.

- Click  in the *Background Map* group. The *Add/Remove Background Map Objects* dialog displays.
- To add additional objects to the map, pick them in a graphic view, or click **Options** for selection options.
- Click **Add**. The selected objects are added to the background map.
- To remove excess objects, pick them or click **Options**.
- Click **Remove**. The objects are removed from the map.
- Click **OK**. The background map is saved as a .dxf file in the data synchronization area.
- To check the objects included in the background map, click **Select** in the *SCS Site Manager*.

### Related topics


- [Edit a Site's Name and Settings](#) (on page 383)
- [Resolve a Conflict in Background Maps](#) (on page 392)

## Reassign a Site to Controllers

HEAVY CONSTRUCTION 

Assign your site to specific controllers so they can be accessed by the designated crews in the field.

### To change a site's assigned controllers:

1. Do one of the following:
  - Select **SCS > SCS Site Manager**.
  - Click the  icon on the toolbar.

The *SCS Site Manager* command pane displays.
2. Click **Reassign** in the *Controllers* group. The *Reassign Site* dialog displays.
3. Check boxes for any controllers to which you want to assign the site. To remove the site from any controllers, uncheck the box.

**Warning:** If you unassign a site from a controller, all data and work orders associated with the site are removed from the controller as well.

4. Click **OK**.


### Related topics

- ❑ [Edit a Site's Name and Settings](#) (on page 383)
- ❑ [Site Options](#) (on page 381)

## Resolve a Conflict in Site Settings

HEAVY CONSTRUCTION 

Change unsupported project settings so that they can be recognized by assigned controllers. Unsupported settings are those that are valid in this software, but not in site controller software (SCS). For example, the angular unit "Mils" is unsupported in SCS, so it must be changed to a value that can be recognized.

Resolve conflicts between site-related settings in your project, and those on assigned controllers by changing and resynchronizing the settings. Conflicting settings are those that are valid in, but inconsistent between, this software and SCS. A conflict is indicated by the active **Resolve** button with the  icon in the *SCS Site Manager*.

Causes of a site settings conflict include differences in distance or angular units, station format, or coordinate order.

**Note:** If there is a conflict in site units, you cannot edit or resolve conflicts in control points or background maps.

**Note:** The *Undo* and *Redo* commands do not affect any of the functions that synchronize files. *Undo* and *Redo* affect only edits made to database objects within your project; this does not include files in the data synchronization area (which SCS uses). If you try to undo a conflict resolution, the **Resolve** button remains disabled because the conflict is still resolved.

#### To resolve unsupported project settings:

1. If applicable, click **Resolve** in the *SCS Site Manager*. The *Unsupported Project Settings* dialog displays.
2. Review the settings that are unsupported, and click **Project Settings**. The *Project Settings* dialog displays.
3. Change the unsupported settings to values that are supported in SCS, and click **OK**. If you selected a supported setting that is in conflict with the settings on the assigned controllers, the *Resolve Site Units Conflict* dialog displays (see the next section). Otherwise, the conflict is resolved.

#### To resolve a conflict in site settings:

1. Click **Resolve** in the *SCS Site Manager*. The *Resolve Site Units Conflict* dialog displays.
2. Review the settings that are in conflict.
3. To use the units from your *Project Settings*, click **OK**. The site settings are updated in the data synchronization area, and the conflict is resolved. Otherwise, to use the settings on your controllers, click **Project Settings**. The *Project Settings* dialog displays.

**Warning:** Resolving a conflict by selecting the project-based units overwrites the units on the assigned controllers when they are synchronized.


4. Edit the settings in your project that are in conflict with those on the controllers so they match, and click **OK**. The conflict is resolved.

#### Related topics

- ❑ [Create a Site](#) (on page 379)
- ❑ [Add an External Site](#) (on page 380)
- ❑ [Edit a Site's Name and Settings](#) (on page 383)
- ❑ [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)

## Resolve a Conflict in Site Calibrations

### HEAVY CONSTRUCTION

Resolve inconsistencies between a site calibration you have included in your site data and calibrations for the same site on assigned controllers by choosing which calibration to use for all. A conflict is indicated by the active **Resolve** button with the  icon in the **SCS Site Manager** command pane.

Causes of a site calibration conflict include differences in site calibration names or dates. In addition, any differences between the coordinate system, datum, geoid, projection, or shift grid you specified in your project and those on assigned controllers will need to be resolved.

**Note:** The **Undo** and **Redo** commands do not affect any of the functions that synchronize files. **Undo** and **Redo** affect only edits made to database objects within your project; this does not include files in the data synchronization area (which SCS uses). If you try to undo a conflict resolution, the **Resolve** button remains disabled because the conflict is still resolved.

#### To resolve a site calibration conflict:

1. Click **Resolve** in the **SCS Site Manager**.
2. Click **Yes** if asked if you want to resolve conflicts. The **Resolve Site Calibration Conflict** dialog displays, showing any calibration in your project and all .dc calibration files for the site on the assigned controllers.
3. Select either a project-based or controller-based calibration (or coordinate system file) in the list.

**Warning:** Resolving a conflict by selecting the project-based calibration overwrites the calibration on assigned controllers when they are synchronized.


4. Click **OK**. The conflict is resolved.

#### Related topics

- ❑ [Edit a Site's Name and Settings](#) (on page 383)
- ❑ [Site Options](#) (on page 381)
- ❑ [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)

## Resolve a Conflict in Control Points

### HEAVY CONSTRUCTION

Resolve inconsistencies between control points you have included in your site data and control points for the same site on assigned controllers by choosing which points to use for all. A conflict is indicated by the active **Resolve** button with the  icon in the **SCS Site Manager** command pane.

Causes of a control point conflict include differences in coordinates, feature codes, point file names or dates, or the points they contain.

**Note:** You cannot resolve a conflict in control points if there is a conflict in site units.

**Note:** Certain characters are invalid in control point IDs:

/ \ : \* ? < > | ; , ~ ` ^ " \$ [ ] { } ^ ( )

**Note:** The **Undo** and **Redo** commands do not affect any of the functions that synchronize files. **Undo** and **Redo** affect only edits made to database objects within your project; this does not include files in the data synchronization area (which SCS uses). If you try to undo a conflict resolution, the **Resolve** button remains disabled because the conflict is still resolved.

#### To resolve a multiple control point file conflict:

1. Click **Resolve** in the **SCS Site Manager**.
2. Click **Yes** if asked if you want to resolve conflicts. The **Resolve Control Points File Conflict** dialog displays, showing the files that exist on a single controller.
3. Select a control point file in the list, and click **OK**. All other control point files are removed, and the conflict is resolved.
4. If there are multiple control point files on another controller, repeat the process and click **OK**. If you selected a control point file that is in conflict with others on assigned controllers, the **Resolve Control Points Conflict** dialog displays (see the next section). Otherwise, the conflict is resolved.

#### To resolve a control point conflict:

1. Click **Resolve** in the **SCS Site Manager**.
2. Click **Yes** if asked if you want to resolve conflicts. The **Resolve Control Points Conflict** dialog displays, showing the control points in your project and on assigned controllers that are in conflict.
3. Check one box for the control point you want to retain in each set of conflicting points.
4. Click **OK**. The conflict is resolved.


**Warning:** Resolving a conflict by selecting the project-based control points overwrites the control points on assigned controllers when they are synchronized.

### Related topics

- ❑ [Add or Remove Control Points](#) (on page 386)
- ❑ [Edit a Site's Name and Settings](#) (on page 383)
- ❑ [Site Options](#) (on page 381)
- ❑ [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)

## Resolve a Conflict in Background Maps

### HEAVY CONSTRUCTION

Resolve inconsistencies between a background map you have included in your site data and background maps for the same site on assigned controllers by choosing which file to use for all. A conflict is indicated by the active **Resolve** button with the  icon in the **SCS Site Manager** command pane.

Causes of a background map conflict include differences in background map file names, dates, or the objects they contain.

**Note:** You cannot resolve a conflict in background maps if there is a conflict in site units.

**Note:** The **Undo** and **Redo** commands do not affect any of the functions that synchronize files. **Undo** and **Redo** affect only edits made to database objects within your project; this does not include files in the data synchronization area (which SCS uses). If you try to undo a conflict resolution, the **Resolve** button remains disabled because the conflict is still resolved.

### To resolve a background map conflict:

1. Click **Resolve** in the **SCS Site Manager**.
2. Click **Yes** if asked if you want to resolve conflicts by using the project-based map. Otherwise, click **No**. The **Resolve Background Map Conflict** dialog displays, showing background maps in your project and on assigned controllers that are in conflict.
3. Select a background map file in the list, and click **OK**. The conflict is resolved.

**Warning:** Resolving a conflict by selecting the project-based background map overwrites the background map on assigned controllers when they are synchronized.

### Related topics

- ❑ [Add or Remove Background Map Objects](#) (on page 387)



- ❑ [Edit an Site's Name and Settings](#) (see "Edit a Site's Name and Settings" on page 383)
- ❑ [Site Options](#) (on page 381)
- ❑ [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)

## Manage Designs and Design Data

### HEAVY CONSTRUCTION

After creating or adding a site to your project, use the **SCS Design Manager** to create designs, add design data, and assign the design data to specific controllers with site controller software. Then, use the **Work Order Editor** to create work instructions for your field crews.

For additional information, see [Understanding Sites, Designs, Work Orders, and Synchronization](#) (on page 373).

#### Related topics

- ❑ [Create a Design](#) (on page 393)
- ❑ [Manage a Site and Site Data](#) (on page 377)
- ❑ [Manage Work Orders and Results](#) (on page 406)


### Create a Design

### HEAVY CONSTRUCTION

Create designs and add design data, such as a surface model, stakeout points, and foreground map objects, so that they can be sent to specific controllers to be used on the job site. Multiple designs can be associated with a site.

Designs that you add or create become objects within your project. A design is essentially a container in which you can designate portions of project data as design data related to the site, making it available to field crews using site controller software.

#### To create a design:

1. Do one of the following:
  - Select **SCS > Create SCS Design**.
  - Click the  icon on the toolbar.

The **Create SCS Design** command pane displays.
2. Type a unique name in the **Design name** box.
3. Check the box for each controller you want to assign the design to in the **Controllers** group.

**Note:** You can create a design without assigning it to any controllers, but the design cannot be saved to, or used on, controllers until you do so. The design and design data are saved in your project until the design is assigned to specific controllers.

4. Click **OK**. The *SCS Design Manager* command pane displays, and the design appears in the *Project Explorer*. See [Edit Design Data](#) (on page 395).

#### To delete a design:

1. Select the design in the *Project Explorer*.
2. Do one of the following:
  - Right-click and select **Delete** from the context menu.
  - Press **[Ctrl] + [D]**.

**Note:** If you delete a design that is assigned to controllers, it becomes an external design, is moved into *External Designs* in the *Project Explorer*, and remains in the data synchronization area. If you delete a design that is not assigned to any controllers, it is removed from your project and the data synchronization area, and disappears from the *Project Explorer*.

If you delete a design using SCS Data Manager or Windows® Explorer, it remains in your project, but assignments to any controllers are lost.

#### Related topics

- [Design Options](#) (on page 396)
- [Edit Design Data](#) (on page 395)
- [Add an External Design](#) (on page 394)

### Add an External Design




Add external designs to your site to make them part of your project so you can edit their design data. External designs are those that exist in the data synchronization area, but not your current project.

#### To add an external design:

1. Do one of the following:
  - Right-click an external design in the *Project Explorer*, and select **Add SCS External Design** from the context menu. The design is added to your site and project, and moves from *External Designs* to *Designs* in the *Project Explorer* (step 2 in not needed).

or

- Select **SCS > Add SCS External Design**.
- Click the  icon on toolbar.

The *Add SCS External Design* dialog displays.

2. Select an external design in the list and click **OK**. The design is added to your site and project, and moves from *External Designs* to *Designs* in the *Project Explorer*.

**Note:** If an external design includes a Terramodel .pro file, it cannot be added to your site. Terramodel files are not supported.

#### To delete an external design:

1. Right-click an external design in the *Project Explorer*, and select **Reassign SCS External Design** from the context menu. The *Reassign SCS External Design* dialog displays.
2. Uncheck the box for each controller.
3. Click **OK**. A confirmation message displays.
4. Click **Yes**. The design is removed from the *Project Explorer* and the data synchronization area.

#### Related topics

- ❑ [Create a Design](#) (on page 393)
- ❑ [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)


## Edit Design Data

### HEAVY CONSTRUCTION



Review and edit your design data so it can be used on controllers at the job site. Prior to editing the data, make sure that you have imported any point and CAD data that you want to make available to construction crews on site. Then you can add additional points and objects to the site data as needed. If you want to use a road or surface model in your design, import any surface and alignment data that you need.


#### To edit site data:

1. If you have just created a design, the *SCS Design Manager* command pane is displayed. Otherwise, do one of the following:
  - Right-click a design in the *Project Explorer*, and select **SCS Design Manager** from the context menu.
  - Select **SCS > SCS Design Manager**.

- Click the  icon on the toolbar.

The **Select Design** dialog displays. Select the design you want to edit, and click **OK**. The **SCS Design Manager** command pane displays.

2. To add a surface model to your design, click **Edit** in the **Model** group. The **Edit SCS Design Model** dialog displays (see [Add and Edit a Design Model](#) (on page 398)).
3. To highlight the stakeout points that are included in the design data in graphic views, click **Select** in the **Stakeout Points** group.
4. To change the stakeout points included in the design data, click . The **Add/Remove Stakeout Points** dialog displays (see [Add or Remove Stakeout Points](#) (on page 399)).
5. To highlight the foreground map objects that are included in the design data, click **Select** in the **Foreground Map** group.
6. To change the foreground map objects included in the design data, click . The **Add/Remove Foreground Map Objects** dialog displays (see [Add or Remove Foreground Map Objects](#) (on page 401)).

**Note:** Changing a design's data can cause it to become unsynchronized with the data synchronization area, creating conflicts. An unsynchronized design is indicated by the  icon in the **Project Explorer**. To resolve conflicts after you edit a design, click each **Resolve** button that is available.

7. Click **Close**. The design data is saved to the data synchronization area.

### Related topics

- [Design Options](#) (on page 396)
- [Resolve a Conflict in an SCS Model](#) (see "Resolve a Conflict in Design Models" on page 402)
- [Resolve a Conflict in SCS Stakeout Points](#) (see "Resolve a Conflict in Stakeout Points" on page 403)
- [Resolve a Conflict in SCS Foreground Maps](#) (see "Resolve a Conflict in Foreground Maps" on page 404)

## Design Options

### HEAVY CONSTRUCTION

Use these options to add and edit design data, such as a surface model, stakeout points, and a foreground map. In addition, you can change the site controllers to which the design is assigned. They are available in the **SCS Design Manager**.

## Options



### Info

Click this to launch the **SCS Data Manager** (on page 376).


### Model

**Design name** - This shows the name of the current design.

**Type** - This shows whether the design model is a surface or road surface.

**Last update** - This displays the date the surface model was added or last modified.

**Edit** - Click this to display the **Edit SCS Design Model** dialog, where you can add a model to the design, or edit an existing model.

**Resolve** - If there is a conflict with the model, click this to display the [conflict resolution](#) (see "Resolve a Conflict in Design Models" on page 402) dialog. The  icon indicates that there is a conflict.

### Stakeout points

**Select** - Click this to highlight the stakeout points currently included in the design data.



- Click this to display the **Add/Remove Stakeout Points** dialog, in which you can change the stakeout points included in the design data.

**Resolve** - If there is a conflict with the stakeout points, click this to display the [conflict resolution](#) (see "Resolve a Conflict in Stakeout Points" on page 403) dialog.

### Foreground map

**Select** - Click this to highlight the map objects currently included in the design data.



- Click this to display the **Add/Remove Foreground Map Objects** dialog, in which you can change the map objects included in the design data.

**Resolve** - If there is a conflict with the foreground map, click this to display the [conflict resolution](#) (see "Resolve a Conflict in Foreground Maps" on page 404) dialog.

### Controllers

**Reassign** - Click this to display the **Reassign Design** dialog, in which you can change the controllers to which the design is assigned.

## Related topics

- ❑ [Edit Design Data](#) (on page 395)

- ❑ [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)

## Add and Edit a Design Model

### HEAVY CONSTRUCTION

Add a model to your design to make it available on assigned controllers. Models, which can be defined by a surface, a surface and an alignment, or simply an alignment, are used by crews in the field to perform grading, checks, and other operations.

#### To add or edit a design model:

1. If you have just created a design, the **SCS Design Manager** command pane is displayed. Otherwise, do one of the following:
  - Select **SCS > SCS Design Manager**.
  - Click the  icon on the toolbar.

The **Select Design** dialog displays.

2. Select the design you want to edit, and click **OK**. If no designs exist, you will need to [Create a Design](#) (on page 393). The **SCS Design Manager** command pane displays.
3. Click **Edit** in the **Model** group. The **Edit SCS Design Model** command pane displays.
4. Select **Surface**, **Alignment**, or **Road Surface** as the model type, depending on what kind of data you have. The pane updates, indicating the types of data you need.

**Note:** If the surface used in your design model contains a boundary, all breaklines in the surface will appear as unnamed features when the surface is viewed in SCS. If the surface in your design model does not contain a boundary, the breakline names will appear in SCS.

5. If you are creating a surface or road surface model, select the surface you want to use for the model in the **Surface** list.
6. If you only want to use a portion of the surface, select the boundary you want to use to clip it in the **Surface clipping boundary** list.
7. If you are creating a road surface or alignment model, check the boxes for one or more alignments in the **Alignments** box.

**Note:** The *Alignment* model can be made from 3D alignments defined by a HAL and a VAL, and 2D alignments defined by a HAL alone (if either no VAL exists or more than one VAL exists). This will enable field crews to stake the alignments, or others generated on-the-fly in relation to them, based on a combination of horizontal offset, vertical offset, and cross-slope in SCS.

**Note:** If you do not have the surface, boundary, or alignment in your project yet, you can create them by selecting **New**.

**Note:** For a road surface model, the alignment and surface you select must be coincident.

8. Click **OK**. The model is added to your design. If you created a surface model, it is saved as a .ttm file in the data synchronization area. If you created a road surface or alignment model, it is saved as a .pro file.

### Related topics


- ❑ [Resolve a Conflict in an Model](#) (see "Resolve a Conflict in Design Models" on page 402)
- ❑ [Create a Surface](#) (see "Understanding Surfaces" on page 320)
- ❑ [Create a Boundary](#) (on page 340)
- ❑ [Create an Alignment](#) (on page 288)


## Add or Remove Stakeout Points

### HEAVY CONSTRUCTION

Add stakeout points to your design to make them available on designated controllers. Stakeout points are used by crews in the field to stakeout features at the job site. Both survey points and CAD points can be selected as stakeout points, but you must convert the CAD points to survey points before they can be included.

#### To add/remove SCS stakeout points:

1. Do one of the following:
  - Select **SCS > SCS Design Manager**.
  - Click the  icon on the toolbar.

The *Select Design* dialog displays.
1. Select the design you want to edit, and click **OK**. The *SCS Design Manager* command pane displays.
2. Click  in the *Stakeout Points* group. The *Add/Remove Stakeout Points* command pane displays.
3. To add additional points to the file, pick them in a graphic view, or click **Options** for selection options.

4. Click **Add**. If any CAD points are selected, the **Convert CAD Points** dialog displays (see [Convert CAD Points](#) (see "Convert CAD Points to Survey Points" on page 400)). Otherwise, the selected points are added to the stakeout point file.
5. To remove excess stakeout points, pick them or click **Options**.
6. Click **Remove**. The points are removed from the file.
7. Click **OK**. The stakeout points are saved as a .csv file in the data synchronization area.
8. To check the stakeout points included in the design data, click **Select** in the **SCS Design Manager**.

### Related topics

- [Resolve a Conflict in Stakeout Points](#) (on page 403)
- [Convert CAD Points to Survey Points](#) (on page 400)

## Convert CAD Points to Survey Points

### HEAVY CONSTRUCTION

When you select points to add to your design data as stakeout points, any CAD points you have selected must be converted to survey points to be included.

#### To convert CAD points to survey points:

1. Uncheck the **Delete CAD points when done** box if you want to retain both the CAD and survey points at the same location after the conversion. Otherwise, only the new survey points will remain.

**Note:** Checking this box is recommended to prevent creating duplicate survey points at the same location by selecting the CAD points again in the future.

2. Select a quality for the new survey points in the **Point quality** list (see [Point Options](#) (on page 271) for descriptions of the qualities).
3. Type a feature code for the points if they will be used in feature code processing.
4. Type a name to use as the base name and starting point for all of the new point IDs.



**Note:** By default, the name of your design is used as the starting point ID. The new point IDs will be numbered and increment upwards. If the starting point ID includes a number, the new points will increment upwards from that number. For example, if the name of your site is "Landfill", and you are converting five CAD points, the new survey points will be named *Landfill1*, *Landfill2*...*Landfill5*. If you change the starting point ID to "Converted point 210", the five new survey points will be *Converted point 211*, *Converted point 212*...*Converted point 215*. If any of the incremented names already exists, they will be skipped, and the naming will start again at the next available increment.

**Note:** Certain characters are invalid in point IDs: / \ : \* ? < > | ; , ~ ' ` " \$ [ ] { } ^ ( ) When points are imported, these characters are replaced by an underscore in the point ID. If you type one of these characters in a point ID, it will not be accepted, and the program will beep.

5. Click **Yes** to convert the points, or click **No** to ignore the selected CAD points and only add the selected survey points as stakeout points. The CAD points are converted to survey points and added to your stakeout points. Unlike CAD points, the new survey points can now also appear in the *Point Spreadsheet*.

#### Related topics

- [Add or Remove Control Points](#) (on page 386)
- [Add or Remove Stakeout Points](#) (on page 399)

## Add or Remove Foreground Map Objects


### HEAVY CONSTRUCTION

Add a foreground map to your design to make it available on designated controllers. Foreground map objects are selectable on site controllers in the field. They are used by crews to stake out features, alignments, and surfaces at the job site.

#### To add/remove foreground map objects:

1. Do one of the following:
  - Select **SCS > SCS Design Manager**.
  - Click the  icon on the toolbar.

The *Select Design* dialog displays.

1. Select the design you want to edit, and click **OK**. The *SCS Design Manager* command pane displays.
2. Click  in the *Foreground Map* group. The *Add/Remove Foreground Map Objects* dialog displays.


3. To add additional objects to the map, pick them in a graphic view, or click **Options** for selection options.
4. Click **Add**. The selected objects are added to the foreground map.
5. To remove excess objects, pick them or click **Options**.
6. Click **Remove**. The objects are removed from the foreground map.
7. Click **OK**. The foreground map is saved as a .dxf file in the data synchronization area.
8. To check the objects included in the foreground map, click **Select** in the *SCS Design Manager*.

### Related topics

- ❑ [Resolve a Conflict in Foreground Maps](#) (on page 404)

## Resolve a Conflict in Design Models

### HEAVY CONSTRUCTION

Resolve inconsistencies between a surface or road surface model in your design and models for the same design on assigned controllers by choosing which model to use for all. A conflict is indicated by the active **Resolve** button with the  icon in the *SCS Design Manager* command pane.

Causes of a model conflict include differences in model names, dates, or data.

**Note:** The *Undo* and *Redo* commands do not affect any of the functions that synchronize files. *Undo* and *Redo* affect only edits made to database objects within your project; this does not include files in the data synchronization area (which SCS uses). If you try to undo a conflict resolution, the **Resolve** button remains disabled because the conflict is still resolved.

#### To resolve a model conflict:

1. Click **Resolve** in the *SCS Design Manager*.
2. Click **Yes** if asked if you want to resolve conflicts by using the project-based model. Otherwise, click **No** to display the *Resolve Model Conflict* dialog, showing the model in your project and all .ttm files on the controllers to which the design is assigned.
3. Select either the project-based or a controller-based model in the list.

**Warning:** Resolving a conflict by selecting the project-based model overwrites the model on assigned controllers when they are synchronized.


4. Click **OK**. The conflict is resolved, and the model is updated in your project or the data synchronization area, depending on your choice.

## Related topics

- ❑ [Design Options](#) (on page 396)
- ❑ [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)

## Resolve a Conflict in Stakeout Points

### HEAVY CONSTRUCTION

Resolve inconsistencies between stakeout points you have included in your design and stakeout point for the same design on assigned controllers by choosing which points to use on all. A conflict is indicated by the active **Resolve** button with the  icon in the **SCS Design Manager** command pane.

Causes of a stakeout point conflict include differences in coordinates, feature codes, point file names or dates, or the points they contain.

**Note:** Certain characters are invalid in stakeout point IDs:


/ \ : \* ? < > | ; , ~ ` ^ " \$ [ ] { } ^ ( )

**Note:** The **Undo** and **Redo** commands do not affect any of the functions that synchronize files. **Undo** and **Redo** affect only edits made to database objects within your project; this does not include files in the data synchronization area (which SCS uses). If you try to undo a conflict resolution, the **Resolve** button remains disabled because the conflict is still resolved.

### To resolve a multiple stakeout point file conflict:

1. Click **Resolve** in the **SCS Design Manager**.
2. Click **Yes** if asked if you want to resolve conflicts. The **Resolve Stakeout Points File Conflict** dialog displays, showing the files that exist on a single controller.
3. Select a stakeout point file in the list, and click **OK**. All other stakeout point files are removed, and the conflict is resolved.
4. If there are multiple stakeout point files on another controller, repeat the process and click **OK**. If you selected a stakeout point file that is in conflict with others on assigned controllers, the **Resolve Stakeout Points Conflict** dialog displays (see the next section). Otherwise, the conflict is resolved.

### To resolve a stakeout point conflict:

1. Click **Resolve** in the **SCS Design Manager**.
2. Click **Yes** if asked if you want to resolve conflicts. The **Resolve Stakeout Points Conflict** dialog displays, showing the stakeout points in your project and on assigned controllers that are in conflict.
3. For each point in the list, click the  icon to expand the set.

4. Check one box for the stakeout point you want to retain in each set of conflicting points.
5. Click **OK**. The conflict is resolved, and the stakeout points are updated in the data synchronization area.


**Warning:** Resolving a conflict by selecting the project-based stakeout points overwrites the stakeout points on assigned controllers when they are synchronized.

### Related topics

- ❑ [Design Options](#) (on page 396)
- ❑ [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)

## Resolve a Conflict in Foreground Maps

### HEAVY CONSTRUCTION

Resolve inconsistencies between a foreground map you have included in your design and foreground maps for the same design on assigned controllers by choosing which map to use on all. A conflict is indicated by the active **Resolve** button with the  icon in the **SCS Design Manager** command pane.

Causes of a foreground map conflict include differences in foreground map names, dates, or the objects they contain.

**Note:** The **Undo** and **Redo** commands do not affect any of the functions that synchronize files. **Undo** and **Redo** affect only edits made to database objects within your project; this does not include files in the data synchronization area (which SCS uses). If you try to undo a conflict resolution, the **Resolve** button remains disabled because the conflict is still resolved.

### To resolve a foreground map conflict:

1. Click **Resolve** in the **SCS Design Manager**.
2. Click **Yes** if asked if you want to resolve conflicts by using the project-based map. Otherwise, click **No**. The **Resolve Foreground Map Conflict** dialog displays, showing foreground maps in your project and on assigned controllers that are in conflict.
3. Select a foreground map file in the list, and click **OK**. The conflict is resolved, and the foreground map file is updated in the data synchronization area.

**Warning:** Resolving a conflict by selecting the project-based foreground map overwrites the foreground map on assigned controllers when they are synchronized.

## Related topics

- [Design Options](#) (on page 396)
- [Icons in the Project Explorer](#) (see "View Construction Data in Project Explorer" on page 427)


## Reassign a Design to Controllers

### HEAVY CONSTRUCTION

Assign your designs to specific controllers so they can be accessed by the designated crews in the field. External designs are those that exist in the data synchronization area, but not your current project.


**Note:** You cannot reassign a design if it has conflicts, or if the site the design is associated with has conflicts. Resolve the conflicts first.

#### To reassign a design:

1. Do one of the following:
  - Right-click a design in the *Project Explorer*, and select **SCS Design Manager** from the context menu.
  - Select **SCS > SCS Design Manager**.
  - Click the  icon on the toolbar.

The *Select Design* dialog displays.
2. Select the design you want to edit, and click **OK**. The *SCS Design Manager* command pane displays.
3. Click **Reassign** in the *Controllers* group. The *Reassign Design* dialog displays.
4. Check boxes for any controllers on which you want on access the design. To remove the design from any controller, uncheck the box.
5. Click **OK**.

#### To reassign an external design:

1. Do one of the following:
  - Right-click an external design in the *Project Explorer*, and select **Reassign SCS External Design** from the context menu.
  - Select **SCS > Reassign SCS External Design**.
  - Click the  icon on the toolbar.
2. Select the design in the *Select External Design* dialog, and click **OK**. The *Reassign SCS External Design* dialog displays.

3. Check the box for the controllers on which you want to access the design. To remove the design from any controllers, uncheck the box.
4. Click **OK**.

#### Related topics

- ❑ [Design Options](#) (on page 396)

## Manage Work Orders and Results

### HEAVY CONSTRUCTION

After creating a site and one or more designs, use the *Work Order Editor* to create work orders to send to your field crews, instructing them what tasks are to be performed at the job site, as well as specifying the design, tolerances, and other settings to use.

For additional information, see [Understanding Sites, Designs, Work Orders, and Synchronization](#) (on page 373).

#### Related topics

- ❑ [Workflow for Managing Work Orders](#) (on page 406)
- ❑ [Create a Work Order](#) (on page 409)
- ❑ [Manage a Site and site Data](#) (on page 377)
- ❑ [Manage Designs and Design Data](#) (on page 393)

## Workflow for Managing Work Orders

### HEAVY CONSTRUCTION

Prepare instructions for your field crews by creating work orders in which you outline the required task, tolerances, and other settings to be use. Here is the general workflow:

1. After you have added a site and design to your project, create a work order and assign it to a controller.
2. Select a related design, if appropriate.
3. Specify the primary task to be completed, and the tolerances and settings to use.
4. Graphically review the site and design data associated with the work order.
5. Save the work order. The work order is written to the data synchronization area so it can be transferred to the designated controller when it is synchronized.
6. If needed, compress and e-mail the work order to colleagues or field crews.
7. Once the task outlined in the work order has been completed and returned from the field, import it and any results into your project.

8. Review the completed work order and field measurement data.
9. Run work order reports.
10. Copy, update and reuse the work order, or archive it in case you need to retrieve it in the future.

**Types of work order tasks:**

- **General** - Use this type to instruct a field crew to perform a miscellaneous task, as specified in the *Instructions* box.
- **Check surface grade** - Use this type to instruct a field crew to measure a surface while work is in progress, so that the measured surface can be compared to a specified design surface. This will help determine how much fill or cut remains to be done at each measurement point. The resulting report also shows whether the measured surface elevation is within tolerance, and therefore whether the design intent has been achieved. The **Check surface grade** task monitors and documents progress towards, and final achievement of, design specifications. Data required for this task includes a design containing the design surface data and surface grade tolerances, if applicable.
- **Check material thickness** - Use this type to instruct a field crew to measure the upper surface of a particular material, such as a roadway base. The upper surface can then be compared to the previously measured upper surface of the underlying material to compute the material thickness. Data required for this task includes a design containing the previously measured underlying surface, the specified material thickness, and the allowable tolerances, if applicable.
- **Check grade & material thickness** - Use this type to instruct a field crew to measure the surface of a material in order to compare the achieved surface shape to the design surface and, at the same time, to calculate a material thickness. To calculate a material thickness, site control software (SCS) compares the measured surface to the previously measured upper surface of the underlying material. Data required for this task includes a design containing the design surface data, a design containing the previously measured underlying surface, the specified material thickness, and the surface grade and material thickness tolerances, if applicable.
- **Measure surface & site features** - Use this type to instruct a field crew to perform either a Topo surface or a Measure site features operation. Select this task if you want the crew to measure random points, breaklines, and surface boundaries to represent a surface. Also select this task if you want a crew to measure existing point-based, line-based, or enclosed boundary site features. The information collected for a Topo surface and site features task is typically used to map and model existing site features or topography.
- **Stakeout points & alignments** - Use this type to instruct a field crew to perform field staking of the individual points contained in a specified design, or to perform field staking of the points along, or horizontally and/or vertically offset from, a selected alignment. The alignment is contained in the foreground map that is included in a specified design.
- **Stakeout sideslopes** - Use this type to instruct a field crew to stakeout sideslope information.



- **Stakeout surface** - Use this type to instruct a field crew to perform field staking of a surface that is contained in a specified design.
- **Stakeout roadway** - Use this type to instruct a field crew to perform field staking of a roadway alignment that is contained in a specified design.
- **Multiple tasks** - Use this type to instruct a field crew to perform combinations of the above tasks within a single work order, as specified in the *Instructions* box.

**Note:** This task makes all of the possible work order options available.

### Related topics


- [Create a Work Order](#) (on page 409)

## Create a Work Order

### HEAVY CONSTRUCTION

Create work orders to send to your field crews, instructing them what tasks are to be performed at a site, as well as specifying the design, tolerances, and other settings to use. Multiple work orders can be created for a site.

#### To create a work order:

1. Do one of the following:
  - Select **SCS > Create Work Order**.
  - Click the  icon on the toolbar.

The *Create Work Order* command pane displays.

2. Type a unique name in the *Work order name* box.

**Note:** Different work orders assigned to different controllers can have the same name

3. Select a single controller on which to access the work order in the *Controllers* list.

**Note:** Each work order can only be assigned to one controller. This is an SCS requirement.

4. Click **OK**. The *Work Order Editor* displays. See [Edit a Work Order](#) (on page 410).

#### To delete a work order:

1. Right-click the work order in the *Project Explorer*, and select **Delete Work Order** from the context menu. A confirmation message appears.
2. Click **OK**. The work order is removed from the *Project Explorer* and the data synchronization area.


## Related topics

- [Edit a Work Order](#) (on page 410)
- [Work Order Options](#) (on page 411)

## Edit a Work Order

HEAVY CONSTRUCTION 

### To edit a work order:

1. If you have just created a work order, the *Work Order Editor* is already displayed. Otherwise, do one of the following:
  - Right-click a work order in the *Project Explorer*, and select **Work Order Editor** from the context menu.
  - Select **SCS > Work Order Editor**.
  - Click the  icon on the toolbar.

The *Work Order Editor* dialog displays.

2. Select a work order in the tree and click **OK**. The *Work Order Editor* displays.
3. Select the type of task the field crew needs to perform in the *Work order task* list. The options in the dialog change based on the type of task you select. For descriptions of work order tasks, see [Workflow for Managing Work Orders](#) (on page 406).

**Note:** If you change the work order task to one that does not use the values you have set, when you save and reopen the work order, those original values will not reappear if you change the task back to one that does use them.

4. Select the design with data related to the work order in the *Design* list, and specify the design surface offset, if applicable.

**Note:** If your work order is assigned to a different controller than the design, you cannot select the design until any conflicts have been resolved.

5. Set the current status, priority, and date that the work must be completed in the *Work order status* group.
6. Describe the tasks that need to be performed in the *Instructions* box.
7. Enter data and select settings in the other groups as needed. (See [Work Order Options](#) (on page 411)).
8. To see the site and design data that is associated with the work order, click **View**. The *Work Order Viewer* displays. When you're done reviewing the data, click **Close**.
9. Click **OK**.

### Related topics

- ❑ [Work Order Options](#) (on page 411)
- ❑ [View a Work Order's Data Graphically](#) (on page 416)

## Work Order Options

### HEAVY CONSTRUCTION

Use these options to create a work order, instructing field crews what tasks to perform at a job site, as well as specifying the design, tolerances, and other settings to use. They are available in the *Work Order Editor*.

Options with an \* in the dialog are optional for one of these reasons:

- There is a default setting in the site controller software (SCS). Specify a setting to override the default, and the field engineer will not be prompted to choose one.
- The selected task is *Multiple tasks*, so all options are displayed, but may not be needed for the actual task. If you do not specify a required setting, the field engineer will be prompted to do so.

**Tip:** To set tolerances, continuous measurement settings, and the coverage map grid size to a suggested default value, click in the option's box, and click **Set to default**.

## Options

### Work order

<b>Name</b>	This displays the name of the selected work order.
<b>Site name</b>	This shows the name of the site with which the work order is associated.
<b>Controller name</b>	This shows the controller to which the work order is assigned.
<b>Work order task</b>	See the <i>Types of work order tasks</i> section in <i>Workflow for Managing Work Orders</i> (on page 406).

---

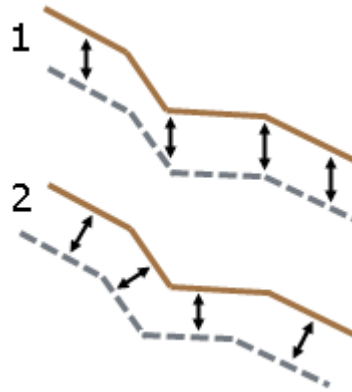
**Note:** The option *Multiple Tasks* shows all of the possible work order settings.

**Warning:** If you change the work order task to one that does not use the values you have set, when you save and reopen the work order, those original values will not reappear if you change the task back to one that does use them.

---

## Designs

<b>Design</b>	<b>Design</b> - Select the design that supplies the required data for this work order.
<b>Surface offset</b>	Enter the offsets that can be applied to a surface in the design, in order to create a different surface. If the field crew, in order to create a different surface, must measure a surface that is offset from the surface specified in the design (above), enter the offset measurement.  Once you have entered the surface offset, indicate whether the offset is applied by raising or lowering the surface by the specified amount:  <b>Above</b> - Select this if the offset surface should be above the surface specified in the design.  <b>Below</b> - Select this if the offset surface should be below that surface.
<b>Offset Type</b>	<b>Vertical</b> - Select this if the surface is to be vertically offset from the design surface (1 in the graphic below).  <b>Perpendicular</b> - Select this if the surface is to be offset perpendicular to the design surface (2 in the graphic below).



### **Underlying surface design**

For a material thickness check, select the design that contains the underlying surface. The underlying surface is the upper surface of the material on which the material of interest has been placed. In most cases, this will be a previously measured surface.

To compute the material thickness at each measured point, the site controller software compares the surface measured by the field crew against the underlying surface.

### **Work order status**

#### **Status**

Select the status that reflects the work order's current state of completion.

#### **Priority**

Select the priority that reflects the relative importance of the work order.

#### **Due date**

Click the arrow to display the calendar. Select the work order completion date.

You can also enter the date in any valid alphanumeric format. The date will automatically be converted to the date format set in your Windows® operating system.

#### **Instructions**

Enter any instructions you have for the field crew.

### **Surface grade tolerances**

Set these tolerances to check a surface grade against a design surface if the selected task requires or allows measurements to be taken. An example of this is grade checking.

### Above design by

Enter an amount (tolerance) that the finished surface can be above the elevation that is established by the design.

---

**Note:** The default setting in most boxes is "?", indicating that no tolerance has been specified. If you enter a tolerance outside the allowable range, you will need to correct the value.

---

### Below design by

Enter an amount (tolerance) that the finished surface can be below the elevation that is established by the design.

### Material thickness and allowable tolerances

Set these tolerances if the selected task requires or allows measurements to be made in order to check the thickness of a material.

<b>Thicker by</b>	Enter an amount (tolerance) that the measured thickness of a material can exceed the design thickness.
<b>Design thickness</b>	Enter the intended thickness of a material lying beneath the surface that is to be measured.
<b>Thinner by</b>	Enter an amount (tolerance) that the measured thickness of a material can be less than the design thickness.
<b>Measured</b>	<p><b>Vertically</b> - When material has been placed on a previously measured surface, select this to compute the material thickness as the difference in elevation between the measured surface and the new surface.</p> <p><b>Perpendicular</b> - When material has been placed on a previously measured surface, select this to compute the material thickness using a perpendicular projection from the measured surface to the new surface.</p>

### Reference line offset from source line

<b>Horizontal offset</b>	Select <b>Left</b> or <b>Right</b> for the direction of the offset from the source line. Then enter a horizontal distance in the box.
<b>Vertical offset</b>	Select <b>Above</b> or <b>Below</b> for the direction of the vertical offset from the source line. Then enter a vertical distance in the box.

### Coverage map

<b>Grid size</b>	Enter the size of the grid cell for the coverage map that is to be displayed on the controller. Enter the size in the units used at the site. For example, if a project is in meters, a value of "10" equates to a cell that is 10 meters square.
------------------	---

### Continuous measurement settings

Set these values if the selected task requires or allows continuous measurements to be taken, either from a vehicle or when walking. These settings determine how often measurements are taken when the site controller software is used in **Continuous** measurement mode.

- Distance interval** Enter a horizontal distance, in the units used at the site where the work order will be completed. Every time the receiver (or survey rod) detects that this change in distance has taken place, a point is recorded.
- Elevation change** Enter an elevation change, in the units used at the site where the work order will be completed. Every time the receiver (or survey rod) detects that this change in elevation has taken place, a point is recorded.

### Related topics


- ❑ [Create a Work Order](#) (on page 409)
- ❑ [Edit a Work Order](#) (on page 410)

## View a Work Order's Data Graphically

### HEAVY CONSTRUCTION

Use the *Work Order Viewer* to verify that the correct site and design data has been provided with the work order; it displays the same graphical view of the data that site controller software (SCS) will display in the field.

#### To view a work order's data graphically:

1. If you have just created a work order, the *Work Order Editor* is already displayed. Otherwise, do one of the following:
  - Select a work order in the *Project Explorer*, right-click and select **Work Order Editor** from the context menu.
  - Select **SCS > Work Order Editor**.
  - Click the  icon on the toolbar.
2. The *Work Order Editor* dialog displays. Select a work order in the tree and click **OK**. The *Work Order Editor* displays.
3. Click **View**. The *Work Order Viewer* displays.
4. Review the site and design data in the *Map view* to confirm that it is complete and appropriate for the work order.
5. Check boxes to show and hide different types of data in the *Display options* group, and use the zoom buttons to scale the view. For coverage map options, see [Work Order Viewer Options](#) (on page 417).
6. To control the visibility of layers both in the *Work Order Viewer* and in the foreground map in SCS, click **Layer Filter**. In the *Foreground Map Layer Filter* dialog, check boxes for the layers you want to show, and uncheck boxes for the layers to hide. Click **OK** to return to the *Work Order Viewer*.



7. Click **Close** to return to the *Work Order Editor*.

### Related topics

- [Work Order Viewer Options](#) (on page 417)

## Work Order Viewer Options

### HEAVY CONSTRUCTION

Use these options to graphically check the site and design data associated with your work order. They are available in the *Work Order Viewer*.

### Options

#### Map view

This shows the site and design data that is associated with the work order you are viewing.

#### Zoom buttons

Click these to scale the view.

#### Display options

Check and uncheck the boxes to show and hide different types of data. If needed, click **Select All** or **Unselect All** to quickly change the layers that are displayed.

#### Coverage map

See *Using coverage maps* at the bottom of this topic.

**Grade cut/fill** - Select this to show the coverage map resulting from a **Check Surface Grade** operation. The color codes show whether the measured elevation indicates the need for additional cut, additional fill, or if the grade is within tolerance. The shade of the red and blue colors provides an indication of the magnitude of additional cut or fill needed:

- **Reds** indicate the need for additional cut to achieve the design elevation
- **Blues** indicate the need for additional fill to achieve the design elevation
- **Green** indicates that the measured elevation is within tolerance of the design elevation

**Elevation** - Select this to show the coverage map resulting from a **Measure Surface** operation. The color codes indicate the relative magnitude of the measured elevation as follows:

- **Darker blue** suggests higher elevations, based on the elevation range formed by the lowest and highest measured elevation, with the darkest blue representing the highest measured elevation
- **Lighter blue** suggests lower elevations, based on the elevation range formed by the lowest and highest measured elevation. Within the lightest blue representing the lowest measured elevation

**Thickness cut/fill** - Select this to show the coverage map resulting from a **Check Material Thickness** operation. The color codes show whether the measured elevation indicates the need for additional cut, additional fill, or if the material thickness is within tolerance of the design thickness. The shade of the red and blue colors provides an indication of the magnitude of additional cut or fill needed:

- **Reds** indicate the need for additional cut to achieve the design thickness
- **Blues** indicate the need for additional fill to achieve the design thickness
- **Green** indicates that the measured elevation demonstrates that the computed material thickness is within tolerance of the design thickness

### Layer filter

Click this to display the **Layer Filter** dialog, and check boxes to control the visibility of foreground map layers both in the **Work Order Viewer** and in the foreground map in SCS.

These layer settings are stored with the work order.

### Zooming and panning options

Once you have selected the display options, you can zoom and pan. The scale bar in the map view updates when you zoom.

To zoom in or out:

- Click **Zoom In** or **Zoom Out**. Then, you can also roll the mouse wheel up and down to zoom.

To zoom to a window:

- Click-and-drag to zoom to a window.

To zoom to the extents of the data:

- Click **Zoom Extents**.

To pan:

- Click-and-drag using the mouse wheel.

### Using coverage maps:

Coverage maps show how much of a site's area has been covered in performing specific surface measurements, as well as the status of certain earthmoving operations based on those measurements. The maps show the area divided into grid squares. The size of the squares is set in the **Coverage map grid size** box in the **Work Order Editor**. For more detailed coverage maps, set the size of the grid squares small enough that only one or two measurements occupy each square. The coverage map settings let you display coverage for measurements made in performing surface measurements (topo surveys), surface grade checks, and material thickness checks.

For any of the settings, if multiple measurements exist in one square, the worst-case measurement (not the average of the measurements) is used. This helps you locate bad measurements that might need to be deleted. As you zoom out, and the scale of your view gets smaller, each coverage map square might actually represent multiple squares. In this case, the same worst-case measurement rule applies. If there are squares that have no measurements within them, the squares turn grey, indicating that the area still needs to be measured. To determine the area that may need to be measured, zoom in to the grey squares to identify the squares without measurements.

### Related topics


- ❑ [Edit a Work Order](#) (on page 410)
- ❑ [View a Work Order's Data Graphically](#) (on page 416)

## Reassign or Copy a Work Order

### HEAVY CONSTRUCTION

Save work by copying work orders that you want to reuse, or changing the controller to which a work order was assigned. Each work order can only be assigned to one controller at a time.

#### To copy a work order:

1. Do one of the following:
  - Right-click a work order in the *Project Explorer*, and select **Copy Work Order** from the context menu.
  - Select **SCS > Copy Work Order**.
  - Click the  icon on the toolbar.

The *Copy Work Order* dialog displays.


2. Select the original work order in the *Work order* tree.
3. Type a name for the new work order in the *New name* box.
4. Select a controller on which to access the new work order in the *New Controller* list.

**Note:** You can only assign copied work orders to controllers to which the site is assigned.

5. Click **OK**. The copy of the work order appears in the *Project Explorer*.

**Note:** When you copy a work order, the new work order's state is set to *Not started* and no work order results are copied.

#### To reassign a work order to a different controller:

1. Do one of the following:
  - Right-click a work order in the *Project Explorer*, and select **Reassign Work Order** from the context menu.
  - Select **SCS > Reassign Work Order**.
  - Click the  icon on the toolbar.

The *Reassign Work Order* dialog displays.


2. Select the work order in the *Work order* tree.
3. Select a different controller on which to access the work order in the *New Controller* list.
4. Click **OK**.

## Compress and E-mail SCS Files


### HEAVY CONSTRUCTION

Compress your site data, design data, and work orders to archive or e-mail them to colleagues or field crews. Files are compressed in a .zip folder.

#### To compress and e-mail site, design, or work order files:

1. Select a site, design, or work order in the *Project Explorer*, and do one of the following:
  - Right-click and select **Compress/E-mail SCS Files** from the context menu.
  - Select **SCS > Compress/E-mail SCS Files**.
  - Click the  icon on the toolbar.

The *Compress /E-mail SCS Files* dialog displays.

2. Select a controller in the *Controller* box. The files assigned to that controller for the selected site, design, or work order appear.
3. Type a path and name for the output file in the *Compressed File Name* box. To browse for a file path, click .
4. Check the box for each file to include in the compressed folder in the *Files to compress* list.
5. To filter the list to include only files that have been created or changed since you last synchronized the selected controller, check the *Only show files modified since last synchronization* box.
6. To send the compressed folder to an e-mail address, check the *E-mail file* box.
7. Type an e-mail address to send the file to in the *E-mail address* box, if applicable.
8. Click **OK**. The selected files are saved in the compressed folder and emailed as specified.

#### Related topics

- [SCS File Compression and Email Options](#) (on page 421)

## SCS File Compression and Email Options

### HEAVY CONSTRUCTION


Use these options to compress and email site, design, and work order files to colleagues or field crews. They are available in the *Compress /E-mail SCS Files* dialog.

## Options

### Controller

Select the controller to which the files you want to compress are assigned.

### Compressed file name

Type a path and name for the output file. To browse for a file path, click .

The default path saves to a temporary folder.

### Files to compress

Check the box for each file to include in the compressed folder. Uncheck boxes for files you do not want to include.

For the assigned controller, all of the files beneath the selected site, design, or work order are listed. By default, all files are selected for inclusion in the .zip file.

### Only show files modified since last synchronization

Check this to filter the list to include only files that have been created or changed since you last synchronized with the selected controller.

### E-mail file

Check this to send the compressed folder to an email address.

---

**Note:** Some Trimble® controllers, such as the TSC2™, can have a dedicated e-mail account. When the controller connects to a network and accesses the e-mail server, the .zip file appears in the e-mail Inbox.

---

### E-mail address

Type the e-mail address for the colleague or controller.

---

**Note:** This feature requires Microsoft® Outlook® to work. Other email programs may work, but are not guaranteed. Web-based email programs, such as Microsoft Hotmail, Yahoo®Mail or Google®Gmail™, will not work.

**Tip:** If your email program is not supported, you can still create the compressed file, open your email program, attach the file to a message, and send it.



---

## Related topics

- ❑ [Compress and E-mail SCS Files](#) (on page 421)

## Import Work Order Results


### HEAVY CONSTRUCTION


When crews return from the field, and synchronize their controllers, new data from the field is transferred to the data synchronization area, making it available to your project. In addition, any site, design, or work order updates that you have prepared and assigned to the controllers are automatically updated on the controllers. When results are available for your work orders, the work order icons in the *Project Explorer* are appended with an  icon like this , indicating that the results can be imported.

These types of files are brought into your project when you import work order results:

- **<work order name>-Report** (.txt) - This file is the work order operations log, the historical record of all field operations performed for the work order.
- **<work order name>-Record** (.txt) - This file is the measurement record for the work order.
- **<work order name>-measure** (.dxf) - This file is the CAD output containing graphical entities resulting from the field measurements. The field crew may not always create one of these.

### To import work order results:

Select a work order that has an  icon in the *Project Explorer*, and do one of the following:

- Right-click and select **Import Work Order Results** from the context menu.
- Select **SCS > Import Work Order Results**.
- Click the  icon on the toolbar.

The results are imported into your project.

### Related topics


- ❑ [Work Order Results Options](#) (see "Edit a View Filter" on page 75)
- ❑ [Run a Work Order Report or Summary](#) (on page 423)

## Run a Work Order Report or Summary

### HEAVY CONSTRUCTION

Run reports to check the status of all your work orders and individual work order results. The **Work Order Summary** lists basics on each of the work orders in your project, including the assigned controller, task, status, priority, and due date. The **Work Order Report** provides details on base, design, difference, and alignment data from the field, as well as tolerances in the original work order.

### To run a summary of work orders:

1. Do one of the following:
  - Select **SCS > Work Orders Summary**.
  - Click the  icon on the toolbar.
  - Right-click **Work Orders** in the *Project Explorer*, and select **Work Orders Summary** from the context menu.


The **Work Orders Summary** displays.

- Review the work orders in your project, clicking any column heading to sort the list based on that column.

**Note:** "?" indicates that no value was specified.


- Click **Close**.


### To filter the Work Orders Summary:

Click the  icon at the top of a column to select a filter from a list. The icon changes color when a filter is active. The options are:

- **All** – This displays all items of the column's type and removes any other filter applied.
- **Blanks** – This displays all rows with a blank in the column.
- **Custom** – Use this to create a custom filter, (e.g. to show all work orders due within the next 3 days).
- **NonBlanks** – This displays all rows that do not have a blank in the column.
- <specific item> - This displays all rows with the specific name in the column. This filter choice allows you to display all rows pertaining to a specific work order, site, design, task, etc.

### To run a report on a specific work order:

Select a work order that has an  icon in the *Project Explorer*, and do one of the following:

- Right-click and select **Work Order Report** from the context menu.
- Select **SCS > Work Order Report**.
- Click the  icon on the toolbar.

The *Work Order Report* displays in your spreadsheet application.

**Note:** If the spreadsheet does not display, the required template is missing or in the wrong location. The template is installed during the site controller software (SCS) installation. If the report template cannot be found, you can set the path to it by selecting **Tools > Options**. Then click **SCS** and **File locations**.

**Note:** The *Work Order Report* does not show work order results.

**Tip:** In the report spreadsheet, click the **Report** tab to view the Report file. For more information about these files, see the documentation for SCS.

### Related topics

- ❑ [Import Work Order Results](#) (on page 422)
- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)



## Troubleshoot an SCS Site or File Problem




Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
<p>The program cannot find the SCS900 report utility .xls file.</p> <p>An error message states that it cannot find the file or the full file extension and if you browse to it the file is correct. Best to keep on the C drive.</p>	<p>Paths to Microsoft® Excel files are limited to 219 characters. In Trimble® Business Center- Heavy Construction Edition, this means that if the path to the SCS report utility file (named <b>SCS900 Reporting Utility.xls</b> or <b>SCS Report Utility.xls</b>) exceeds the limit, the file will not open. In most cases, this will only happen if the .xls file has been placed in the 'My Documents' directory.</p>	<p>Move Excel (.xls) files to locations with shorter paths (fewer subfolders), or rename files with shorter names.</p> <p>Unzipping a folder to a location within several folders could exceed maximum path character limits, preventing deeper files from being unzipped. There may be a warning that the files cannot be unzipped or the windows may simply close, making it look like they were all unzipped. It may be best to unzip files directly to the C drive with full administrative rights.</p>
<p>The SCS900 report utility .xls file does not work.</p> <p>An error message says that there is a mismatch in data when the report is opened.</p> <p><b>Note:</b> If the Import Record.xls is opened directly in Excel, there is a no warning until you click <b>Import Record</b>.</p>	<p><b>Note:</b> The number of the cause corresponds to the number of the solution.</p> <ol style="list-style-type: none"> <li>1. The <b>-Report.txt</b> and <b>-Record.txt</b> files are not both present or in the proper format.</li> <li>2. Language settings do not match in the operating system's regional settings, Trimble Business Center, Microsoft Office, and the SCS Report Utility file.</li> <li>3. The synchronizer root folder is not set to the correct folder in the <b>Options</b> dialog.</li> <li>4. The user does not have privileges to the folder.</li> <li>5. The Report Utility file is a different version from</li> </ol>	<ol style="list-style-type: none"> <li>1. Ensure that both the <b>-Report.txt</b> and <b>-Record.txt</b> are present and in the proper format in the data synchronization area's <b>Output</b> folder.</li> <li>2. Ensure that language settings match in the operating system's regional settings, Trimble Business Center, Microsoft Office (apply a language pack if necessary), and the SCS report utility file.</li> <li>3. In this program, select <b>Tools &gt; Options</b>. Click <b>SCS</b> and then <b>File Locations</b> in the left pane. Click the  icon and navigate to the folder in which SCS installed the <b>SCS Report Utility.xls</b> file.</li> <li>4. If the security level is changed in Excel, then a new Excel program needs to be started for the macro to work. To change the security level in Excel, open <b>Tools &gt; Options &gt;</b></li> </ol>





















	<p>the one used on the site controller.</p>	<p><b>Security &gt; Macro Security</b>, and set the <b>Security Level</b> to <b>Low</b> (or add to the <b>Trusted Publishers</b> list). The macro can then be disabled when the report utility is opened.</p> <p>5. Find and use, or create, a report utility file of the same version as the one used on the site controller. The utility maker is on the SCS installation CD.</p>
<p>The program appears to freeze when trying to e-mail SCS files.</p>	<p>If you are running the program on a secondary monitor, and you attempt to e-mail SCS files using the <b>Compress/E-mail SCS Files</b> command, your e-mail program may open a dialog confirming the operation out of either monitor's visible range. The dialog will be located off of the primary monitor, in the space opposite the secondary monitor, causing the program to appear frozen.</p>	<p>To reach the dialog and confirm the e-mail operation, right-click the e-mail application's name on the Windows taskbar and select <b>Move</b>. Then, press the appropriate arrow key to move the dialog into your primary's monitor's visible range.</p>
<p>Label text in the <b>Work Order Viewer's</b> coverage map is blurry.</p>	<p>In Windows display properties, effects are set to <b>Clear Type</b>.</p>	<p>Smooth the edges of the screen fonts in the viewer. Right-click on your Windows desktop and select <b>Properties</b> from the context menu. In the <b>Display Properties</b> dialog, click the <b>Appearance</b> tab and click <b>Effects</b>. In the <b>Effects</b> dialog, select <b>Standard</b> in the second drop-down list.</p>
<p>You receive a message that your e-mail program is not supported when you try to e-mail SCS files.</p>	<p><b>Note:</b> The number of the cause corresponds to the number of the solution.</p> <p>1. This command requires Microsoft® Outlook® to work. Other email programs may work, but are not guaranteed. Web-based email programs, such as Microsoft Hotmail, Yahoo®Mail or</p>	<p>1. If your email program is not supported, create the compressed file, save it, open your email program, attach the file to a message, and send it.</p> <p>2. Download the Outlook CDO (Collaboration Data Objects) file to fix the problem:  <a href="http://www.microsoft.com/downloads/details.aspx?FamilyId=2714320D-C997-4DE1-986F-24F081725D36&amp;displaylang=en#QuickInfoContainer">http://www.microsoft.com/downloads/details.aspx?FamilyId=2714320D-C997-4DE1-986F-24F081725D36&amp;displaylang=en#QuickInfoContainer</a></p>

Google®Gmail™, will not work.	http://www.microsoft.com/downloads/details.aspx?FamilyId=2714320D-C997-4DE1-986F-24F081725D36&displaylang=en#QuickInfoContainer
2. If you do have Outlook, it may be missing a component file.	For Microsoft Outlook 2003, install the CDO file from the original program CD.

## View Construction Data in Project Explorer

Icons in the *Project Explorer* show the type of each object in the hierarchy. Many of the construction icons also indicate the state of the object. Unsynchronized objects are those in conflict with data in the data synchronization area. They are denoted by the  icon.

### Icons in the Project Explorer

	A project
	A point
	A coordinate of a point
	A surface
	A surface contour
	An alignment
	A synchronized site
	An unsynchronized site
	A site with an unsynchronized design
	A design
	An unsynchronized design
	An external design
	An unsynchronized external design
	An unstarted work order
	An in-progress work order
	A work order with field operations complete
	A completed work order
	An archived work order
	A work order (any state) with imported results
	A boundary

### Related topics

- [Project Explorer](#) (on page 7)

- ❑ [Resolve a Conflict in a Model](#) (see "Resolve a Conflict in Design Models" on page 402)
- ❑ [Resolve a Conflict in Site Calibrations](#) (on page 390)
- ❑ [Resolve a Conflict in Background Maps](#) (on page 392)
- ❑ [Resolve a Conflict in Control Points](#) (on page 391)
- ❑ [Resolve a Conflict in Foreground Maps](#) (on page 404)
- ❑ [Resolve a Conflict in Site Settings](#) (on page 388)
- ❑ [Resolve a Conflict in Stakeout Points](#) (on page 403)

## CHAPTER 16

# Run Reports


## Run an Alignment Geometry Report

Generate an *Alignment Geometry Report* to see a simple summary or detailed listing of the geometry of an alignment in your project. You can choose to report on just the horizontal component or both the horizontal and vertical components of alignments. If you have specified station equations, they will be reported as well.

### To run an Alignment Geometry Report:

1. Select **Reports > Alignment Geometry Report**. The *Alignment Geometry Report* command pane displays.
2. Select an alignment in the *Alignment* list.
3. Click **OK**. The *Alignment Geometry Report* displays in your default Web browser.

### To customize the report:

1. Do one of the following:
  - Click the  icon on the *Alignment Geometry Report* command pane's toolbar.
  - Select **Reports > Report Options**.The *Report Options* command pane displays.
2. Select **Alignment Geometry Report** in the list.
3. Expand sections and specify output settings in the **Settings** group as needed.
4. Click **Apply** if you want to customize additional reports, or **OK** to close the command pane.

**Tip:** You can also change the abbreviations used for horizontal and vertical alignment classifications in the *Alignment Geometry Report*. Select **Project > Project Settings**, and click **Abbreviations** in the left pane. Edit any of the abbreviations in the right column and click **OK**. Rerun the report to see your changes.


### Related topics

- ❑ [Create an Alignment](#) (see "Understanding Alignments" on page 286)
- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Run an Earthwork Report

Use the *Earthwork Report* to calculate volumes based on a single surface, or the comparison of two surfaces.

### To generate a volume report:

1. Do one of the following:
  - Click the  icon on the toolbar.
  - Select **Reports > Earthwork Report**.

The *Earthwork Report* command pane displays.
2. Select the option for the type of surface report you want to generate in the *Report type* group. Options in the steps below will vary based on the report type you select.
3. Select the surface you want to report on, or the first surface to compare, in the *Surface* list. The classification for the surface displays below the box.
4. Select the second surface to compare in the *Final* list, if applicable.
5. Pick a point in a graphic view, right-click for options, or type a value in the *Elevation* box, if applicable.
6. Pick lines to define the perimeter in the *Boundary* box, or click **Options** for more selection methods.
7. To account for materials in the calculations, check **Use in calculation** in the *Materials* group.

**Note:** Before using this option, you need to have defined surface materials in the *Define Materials* dialog.

8. Select the material at the site in the *Native* list.
9. Select the material that will be brought to the site in the *Borrow* list.
10. Select a type in the *Volume Breakdown* group, specifying a value by picking in the graphics window or typing a value, if necessary.
11. Click **OK**. The report displays in your default Internet browser.

### Related topics

- ❑ [Define Materials for Earthwork Reports](#) (on page 351)
- ❑ [Earthwork Report Options](#) (on page 354)

- [Understanding Earthwork Volume Calculations](#) (on page 349)

## Run an Import Report

Generate an *Import Report* to see a project summary, details on imported files, and any associated errors or warning messages.

### To run an Import Report:

- Select **Reports > Import Report**.
- Select **Reports > Report Options**. Select **Import Report** in the command pane, and click **OK**.

The *Import Report* displays in your default Web browser.

**Tip:** Click a file name in the report to jump to the creation and import dates and times.

### Related topics


- [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Run a Mean Angle Report



Run a *Mean Angle Report* to view details of how each mean angle was computed.

### To run a Mean Angle Report:

- Select **Reports > Mean Angle Report**.
- Right-click a mean angle node icon  in *Project Explorer* and select **Mean Angle Report** from the context menu.
- Click the **Report** button in the *Mean Angle Residuals* dialog.

The *Mean Angle Report* displays in your default Web browser.

**Note:** In the *Mean Angle Report*, all angles are normalized to the range of 0 to 360 degrees expressed in project units. All distances are displayed in project units.

The *Mean Angle Report* includes a separate table for each mean angle in the project. At the top of each table, the point ID, station ID, and backsight ID are displayed. Beneath that, the table includes a row of information for each enabled observation used to compute the mean angle. Information includes observed readings and residual values for the horizontal angle, vertical angle, and slope distance. The last row in the table displays the computed horizontal angle, vertical angle, and slope distance for the mean angle point.

**Tip:** Click any point in the report to select it in the *Project Explorer* and graphical and spreadsheet views, and display its properties in the *Properties* pane.

### Related topics

- ❑ [Understanding Total Station Data](#) (on page 233)
- ❑ [Workflow for Total Station Data](#) (on page 238)
- ❑ [View Total Station Data in Project Explorer](#) (on page 240)
- ❑ [View and Edit Mean Angle Residuals](#) (on page 248)

## Run a Point Derivation Report

Generate a *Point Derivation Report* to see details on the survey data used to calculate the final coordinates of points in your project.

### To run a Point Derivation Report:

- Select **Reports > Point Derivation Report**.
- Select **Reports > Report Options**. Select *Point Derivation Report* in the command pane, and click **OK**.

The *Point Derivation Report* displays in your default Web browser.

**Tip:** Click a point ID or coordinate in the report to select the point in graphic views and the *Project Explorer*.

### Related topics

- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Run a Point List Report

Generate a *Point List* to see a simple summary of the coordinates for each point in your project.

### To run a Point List report:

- Select **Reports > Point List**.

The *Point List* displays in your default Web browser.



**To modify the report:**

- Select **Reports > Report Options**. Select **Point List** in the command pane, and click **OK**.
- In the **Settings** group at the bottom of the command pane, you can specify the type of coordinates (grid, local, or global), and the type of data to display. The data options include quality control information such as scale factors and convergence angle.

**Tip:** Click a point ID in the report to select the point in graphic views and the **Project Explorer**.

**Related topics**

- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Run a Project Computation Report

Generate a computation report to see a summary of the errors and warnings that occurred during the last computation of your project data.

**To run a Project Computation Report:**

- Select **Reports > Project Computation Report**. The **Project Computation Report** displays in your default Web browser.

**To customize the report:**

1. Select **Reports > Report Options**. The **Report Options** command pane displays.
2. Select **Project Computation Report** in the list.
3. Expand sections and specify output settings in the **Settings** group as needed.
4. Click **Apply** if you want to customize additional reports, or **OK** to close the command pane.


**Related topics**

- ❑ [Project Computations](#) (on page 157)
- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Run a Renamed Point List

Generate a **Renamed Point List** to see a simple summary of the original and new names of points that you have renamed in your project.

1. Select **Reports > Report Options**. The **Report Options** command pane displays.
2. Select **Renamed Point List** in the **Reports** list.
3. Edit options in the **Settings** group as needed.

4. Click the  icon on the pane's toolbar. The *Renamed Point List* displays in your default web browser.

### Related topics

- [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Run a Site Calibration Report

After you calibrate a site, generate a *Site Calibration Report* to see details on the local site settings, horizontal and vertical calibration parameters, and residual differences between GNSS and grid points in your project.

### To run a Site Calibration Report:

- Select **Reports > Site Calibration Report**.

The *Site Calibration Report* displays in your default Web browser.

### To modify the report:

- Select **Reports > Report Options**. Select *Site Calibration Report* in the command pane, and click **OK**.

In the **Settings** group at the bottom of the command pane, you can specify the header and footer data to display.


### Related topics

- [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Run a Surface Report

Run a surface report to see surface measurements and limits, as well as the number of triangles, vertices, and other items in a surface in your project.

### To generate a surface report:

1. Do one of the following:
  - Select **Reports > Surface Information Report**.
  - Click the  icon.

The *Surface Information Report* command pane displays.
2. Select the surface you want to generate a report for in the **Surface** list.
3. Click **OK**. The report displays in your default Web browser.

### Related topics


- [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Run a Work Order Report or Summary

### HEAVY CONSTRUCTION

Run reports to check the status of all your work orders and individual work order results. The **Work Order Summary** lists basics on each of the work orders in your project, including the assigned controller, task, status, priority, and due date. The **Work Order Report** provides details on base, design, difference, and alignment data from the field, as well as tolerances in the original work order.

#### To run a summary of work orders:

1. Do one of the following:
  - Select **SCS > Work Orders Summary**.
  - Click the  icon on the toolbar.
  - Right-click **Work Orders** in the *Project Explorer*, and select **Work Orders Summary** from the context menu.


The **Work Orders Summary** displays.

2. Review the work orders in your project, clicking any column heading to sort the list based on that column.

**Note:** "?" indicates that no value was specified.


3. Click **Close**.


#### To filter the Work Orders Summary:

Click the  icon at the top of a column to select a filter from a list. The icon changes color when a filter is active. The options are:

- **All** – This displays all items of the column's type and removes any other filter applied.
- **Blanks** – This displays all rows with a blank in the column.
- **Custom** – Use this to create a custom filter, (e.g. to show all work orders due within the next 3 days).
- **NonBlanks** – This displays all rows that do not have a blank in the column.
- <specific item> - This displays all rows with the specific name in the column. This filter choice allows you to display all rows pertaining to a specific work order, site, design, task, etc.

### To run a report on a specific work order:

Select a work order that has an  icon in the *Project Explorer*, and do one of the following:

- Right-click and select **Work Order Report** from the context menu.
- Select **SCS > Work Order Report**.
- Click the  icon on the toolbar.

The **Work Order Report** displays in your spreadsheet application.

**Note:** If the spreadsheet does not display, the required template is missing or in the wrong location. The template is installed during the site controller software (SCS) installation. If the report template cannot be found, you can set the path to it by selecting **Tools > Options**. Then click **SCS** and **File locations**.

**Note:** The **Work Order Report** does not show work order results.

**Tip:** In the report spreadsheet, click the **Report** tab to view the Report file. For more information about these files, see the documentation for SCS.

### Related topics


- ❑ [Import Work Order Results](#) (on page 422)
- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)

## Customize a Report

Use report options to customize your reports. The settings you specify are saved so that output remains consistent each time you run a certain report. The **Report Options** command pane lists all the available reports. For some software modules, the **Earthwork Report** is available from the Reports menu. All reports are displayed in your default Web browser.

**To customize a report:**


You can customize the report layout as desired, selecting what to show in the header, footer, report settings and report sections.

1. Select **Reports > Report Options**. The *Report Options* command pane displays.
2. Select the report in the *Reports* list.
3. To view the settings for a report, click the  icons to expand sections in the *Settings* group. Click in boxes, and change options as needed.

**Tip:** To access commonly-used reports from the Reports menu, set the *Show on Reports menu* option to *Yes*.

4. Click **OK**.

**To run a report:**

1. Do one of the following:
  - Select **Reports > (report name)**. The report displays in your default Web browser.
  - Select **Reports > Report Options** if you want to review the report settings. The *Report Options* dialog displays.
2. Select the report in the *Reports* list.
3. Click the  icon on the command pane toolbar. The report displays in your default Web browser.
4. Click **OK** to close the dialog.

**Note:** Reports are not saved automatically. To save a report, select **File > Save as** in your Web browser.

**Related topics**

- [Report Options](#) (on page 438)
- Run a Job File Report

## Report View

The *Report View* displays when you run certain reports. Along with the content of the report, the view includes a toolbar located along the top of the tab that allows you to:

- Navigate to a specific page in the report
- View and change the print setup information
- View the print layout and print the report
- Export the report to a spreadsheet or PDF document
- Select a magnification to view the report
- Search for text in the report

#### Related topics

- [Tabbed View Arrangement](#) (on page 34)

## Report Options

Use these common options to customize report output. They are available in the **Report Options** command pane. Specific reports have additional options as well. For a description of any setting, click the name of the setting. The description appears in the information box at the bottom of the command pane.

## Settings

<b>Column display / Column selection</b>	Select which data types to show in the report by setting individual columns to <b>Show</b> or <b>Hide</b> .
<b>Coordinate type</b>	Select whether to show global, local, or grid coordinates in the report.  This is for the <b>Point List</b> .
<b>Footer</b>	Select whether to show the date, project name, and application name by selecting <b>Show</b> or <b>Hide</b> for each.
<b>Format options</b>	Select from summary or detailed formats.
<b>Header</b>	Select whether to show company, project, user, and coordinate system data by selecting <b>Show</b> or <b>Hide</b> for each.
<b>Report sections / Sections</b>	Select which data types to show in the report by setting individual sections to <b>Show</b> or <b>Hide</b> .
<b>Report setting</b>	Set the number of legs in loops, as well as PPM and delta criteria for <b>Loop Closure Results</b> .
<b>Residual plot</b>	<b>Separate systems</b> - Select this to combine all satellite residual data into a single graph.  <b>Individual satellites</b> - Select this to plot separate graphs of each satellite's residuals.  This is for the detailed <b>Baseline Processing Report</b> .
<b>Run-time display</b>	<b>Never</b> - Select this to prevent the report from displaying when it is generated (for example, during import); display it only when selected from the menu or the <b>Report Options</b> command pane.  <b>Show when warnings or errors are present</b> - Select this to display the report when it is generated if errors or warnings occur (for example, during import).  <b>Prompt</b> - Select this to display a prompt when the report is generated to ask if you want it to display.  <b>Prompt on warnings or errors</b> - Select this to display a prompt if errors or warnings are present when the report is generated.  <b>Always</b> - Select this to always display the report when it is generated.

**Save intermediate data** Select whether to include intermediate data in the report.

**Show on Reports menu** **No** - Select this to remove the report from the Reports menu  
**Yes** - Select this to re-add the report to the menu.

### Related topics

- ❑ [Customize and Run a Report](#) (see "Customize a Report" on page 436)
- ❑ Run a Job File Report



## CHAPTER 17

# Export Data

## Export Data

Export data from your project in a variety of formats. See the individual file format topics for details.

### To export data:


1. Do one of the following:
  - Select **File > Export**.
  - Click the  icon on the toolbar.


The **Export** command pane displays.

2. Click an export type (*Survey*, *CAD*, *Custom*, or *Construction*) in the **File Format** group. A list of available exporters displays.
3. Select an export format in the list. If one with the desired format is not listed, create a custom exporter.

**Caution:** If you have a field device connected, only file types compatible with the device appear.

4. If needed, use the [View Filter](#) (see "Filter a View" on page 76) command to filter the selectable data in the plan view.
5. Select the data to export using one of the [Selection Options](#) (see "Selection Methods and Options" on page 43).

6. Select a folder in the **File Name** list, or click the  icon to browse for a folder.

When you click the  icon, the **Save As** dialog displays with the export folder specified in **Tools > Options > File Location** selected. However, you can browse to and select a different folder.

7. Type a new file name in the **File Name** box if you do not want to overwrite an existing file.

8. If export settings appear in the **Settings** group, specify them as needed.
9. Click **OK** to export the data.

**Tip:** You can select data before you begin the **Export** command.

**Tip:** To customize the format of the exported data, select **File > Export Format Editor**.

### Related topics

- Export Format Editor
- Export Data Formats

## Export and Upload Data Formats

### HEAVY CONSTRUCTION

The **Export** command and data synchronization functions for sites, designs and work orders enable you to send the following types of data out from your project. See file-specific topics for details.

File format	Data type
<b>.csv, .pts</b>	<a href="#">ASCII</a> (see "Export ASCII Files" on page 443) point files
<b>.dc</b>	<a href="#">Trimble data collector</a> (see "Export Trimble Data Collector Files (.dc)" on page 448) files
<b>.dxf, .dwg</b>	<a href="#">CAD</a> (see "Export CAD Files (.dxf/.dwg)" on page 444) files
<b>.ini</b>	site settings files
<b>.pro</b>	Terramodel project files
<b>.ttm</b>	<a href="#">Trimble terrain model</a> (see "Export Trimble Surface Files (.ttm)" on page 448) surface files
<b>.txt</b>	SCS work order report and record files
<b>.xml</b>	<a href="#">LandXML</a> (see "Export LandXML Files (.xml)" on page 445) files

### Related topics

- [Export Data](#) (on page 441)
- [Export Data in a Custom Format](#) (on page 448)


## Export Related Files

You can specify which additional related files (for example, geoid files or datum grid files) to automatically export with a JOB file when you are exporting to a field device either directly (the device is connected) or indirectly (the files will be stored for upload at a later date).

**To specify related files to export to a field device:**

1. Select **View > Command Pane**. The *Command Pane* displays.
2. In the *Command Pane*, select **Related Files**. The *Related Files* command pane displays.

If there are files that are required for the project (for example, a coordinate system file specified in the *Project Settings* dialog), they are displayed in the *File Name* list with a gray background.

3. Click in the first empty *File Name* box, and then either type the path and name of the related file or click  to select the file.
4. Click in the *Application* box and select the associated device application (or <All> or <None>) from the drop-down list.
5. Repeat steps 3 and 4 for each additional related file and its associated device you want to specify.
6. In the *Upload with export* list, select to upload all files, just the files required for the project, or no files.
7. Select the *Copy files to project folder* box if you want to copy the files specified in the list to the project folder at the time of export.

The file referenced in the list is not changed. If the file already exists in the project folder, it is overwritten so that the project folder contains the same version that was uploaded to the device.

8. If there is a currently active device, it is displayed in the *Active Device* box. Click the **Upload Files Now** button to upload the selected related files to the device.
9. When you are done, click **OK**.

The related files you specified will be automatically uploaded with the JOB file when you perform an upload to a device.

**Related topics**

- ❑ [Export Data](#) (on page 441)
- ❑ [Export Data in a Custom Format](#) (on page 448)

**Export ASCII Files**

Export ASCII files (.asc, .csv, .txt,) that can be used in a variety of other applications and field devices, including:

- Spectra Precision® Field Surveyor
- TDS Survey Pro™
- Nikon NEH
- Trimble® Survey Manager™
- Trimble® Survey Controller™
- Trimble® Digital Fieldbook™ (v2,v3, and v5)

Click the **Custom** tab in the **Export** command pane to access the ASCII exporters. Click the **Survey** tab to export a trajectory as a .csv file. You can set various unit and format options for trajectory export in the **Settings** group.

### Related topics

- ❑ [Export Data](#) (on page 441)
- ❑ [Export Data in a Custom Format](#) (on page 448)

## Export CAD Files (.dxf/.dwg)

Select and export some or all of the data in your project to a CAD file. This can be used as a background map in field devices.

**Note:** You can also upload .bmp, .gif, .jpg, .png, and .tif formats to use as background maps in TDS Survey Pro, Trimble Survey Manager, and Spectra Precision® Field Surveyor.

Click the **CAD** tab in the **Export** command pane to access the .dxf and .dwg exporters. You can set the file version and an explode option in the **Settings** group.

Exported CAD files can be used in:

- Spectra Precision® Field Surveyor
- Trimble® Survey Controller™
- Trimble® Digital Fieldbook™ (v2,v3, and v5)
- TDS Survey Pro™ (as base maps)
- Trimble® Survey Manager™

**Note:** By default, a point ID exports as an attribute of the point.

**Note:** The .dwg format does not support all alignment information, such as stationing. If you export an alignment as a .dwg and then import it into another application as a .dxf/.dwg, vertical alignments in the file may not appear either.

### Related topics

- ❑ [Export Data](#) (on page 441)
- ❑ [Export Data in a Custom Format](#) (on page 448)

## Export Geodatabase Files (.xml)

Export feature data contained in your project to a geodatabase XML file from which the data can be imported into an Environmental Systems Research Institute, Inc. (ESRI) geographic information system (GIS).

Click the **GIS** tab in the **Export** command pane to access the **Geodatabase XML** exporter.

**Note:** The geodatabase XML format represents ESRI's most current open mechanism for information interchange between geodatabases and other external systems and is a replacement for the earlier shapefile (.shp) spatial data format.


### Related topics

- ❑ [Export Data](#) (on page 441)
- ❑ [Export Data in a Custom Format](#) (on page 448)

## Export LandXML Files (.xml)

Export alignments, cross-sections, and surfaces using the LandXML file format.

### To export LandXML data:

1. Select **File > Export**. The **Export** dialog displays.
2. Click the **Construction** tab and then select **LandXML exporter** in the **File Format** group.
3. Click in the **Selected entities** box.
4. In a graphic view, pick the objects you want to include in the export, or click **Options** and choose a selection option in the list.
5. If you want to clip a surface that you are exporting, select a boundary in the **Surface clipping boundary** list, or select **<New>** to create one.
6. Type a path and file name for the exported file in the **File Name** box, or click the  icon to browse for a location and specify a file name.
7. In the **Settings** group, set export properties.
8. Click **Export**.

### Related topics

- ❑ [LandXML Export Options](#) (on page 445)
- ❑ [Results of Exporting LandXML Files](#) (on page 446)

## LandXML Export Options

Use these options to specify how to handle surfaces definitions and duplicate points when you export LandXML files. They are available in the **LandXML Export** command pane.

## Options

### Surface description

**Optimize for data** - Select this to let the program choose the best export method (of the two below).

If the surface you have selected for export contains any internal data (surface was imported in a TTM or LandXML file), it will export as triangles.

If the surface only references external data (surface was created entirely in this program), it will export as points and breaklines.

---

**Note:** If you are exporting a portion of a surface that is clipped by a boundary, all data in the clipped portion is made internal.

---

**Points and breaklines** - Select this to export the surface, as well as the points, breaklines, contours, and boundaries used to create the surface.

**Triangles** - Select this when you only want to export the triangles defining the surface. This option also handles holes and islands in the data.

---

**Note:** This setting has no effect if the file does not contain surfaces.

---

### Duplicate point IDs

**Always ask** - Select to automatically check for points with the same Point ID. If any are found, you are prompted to import all duplicates, ignore all duplicates, or cancel the import.

**Export all** - Select to have all points export, including those with duplicate Point IDs.

**Ignore all** - Select to have only points without duplicate Point IDs export.

---

**Note:** This setting has no effect if the file does not contain points.

---

## Related topics

- ❑ [Export LandXML Files \(.xml\)](#) (on page 445)
- ❑ [Results of Exporting LandXML Files](#) (on page 446)

## Results of Exporting LandXML Files

When you export LandXML files, the points, alignments, and surfaces are handled in specific ways.

## Exported

### Points

## Results

When points are exported:

- The point name or number is used in the "name" field.
- Any valid feature codes are used in the "desc" fields.
- For invalid feature codes, "\*" is used.
- The "LandXML.desc" attribute of the first "LandXML feature" is used for the "desc" attribute, and the "LandXML.code" attribute of the first "LandXML feature" is used for the "code" attribute if they exist. Otherwise no "desc" or "code" attributes are written.

### Alignments

When alignments are exported:

- The data defining the alignment is written as an alignment in the LandXML file.
- Both horizontal and vertical components are retained. Multiple vertical alignments can be exported with each horizontal.

### Surfaces

When surfaces are exported:

- Either the source data or the definition is used, but not both.
- Points influencing the surface are saved as points.
- 3D polylines, breaklines, and sloping lines influencing the surface are saved as breakline or contour point lists, depending whether they are pure 3D or 2D+elevation.
- Internal breaklines are retained.

## Related topics

- ❑ [Export LandXML Files \(.xml\)](#) (on page 445)
- ❑ [LandXML Export Options](#) (on page 445)

## Export Trimble Data Collector Files (.dc)

Export points and coordinate system only data as .dc files. Click the **Survey** tab in the **Export** command pane to access the .dc exporter. You can set the file version, units, and output format in the **Settings** group.

### Related topics

- ❑ [Export Data](#) (on page 441)
- ❑ [Export Data in a Custom Format](#) (on page 448)

## Export Trimble Surface Files (.ttm)

Export triangulated terrain models (.ttm) to use in:

- Trimble® Survey Controller™
- Trimble® Digital Fieldbook™ (v2,v3, and v5)

Click the **Construction** tab in the **Export** command pane to access the .ttm exporter.


### Related topics

- ❑ [Export Data](#) (on page 441)
- ❑ [Export Data in a Custom Format](#) (on page 448)

## Export Data in a Custom Format

Use the **Export Format Editor** to create a custom converter to export your custom format. The converters created with this editor are used within the [Export](#) (see "Export Data" on page 441) command to export ASCII files with a non-standard format.

### To export a custom format:

1. Do one of the following:
  - Select **File > Export Format Editor**.
  - In the **Export** dialog, click the  icon.

The **Export Format Editor** opens and displays the [Select Definition](#) (see "Definition Options" on page 449) dialog.
2. Select a custom format in the definition list.
3. Click **Next** and select options in the [Description and Search Type](#) (see "Description and Search Type Options" on page 450) dialog.
4. Click **Next** and select options in the [General Properties](#) (see "General Properties Options" on page 212) dialog.
5. Click **Next** and select options in the [Fields](#) (see "Fields Options" on page 451) dialog.




6. Click **Finish** to create the exporter file.

You can create a custom converter to export any of the following:

- Delimited files - data is separated by a specific character.
- Fixed-width files - data is separated into defined columns.
- Files where data location is defined by a beginning and/or ending string of text.

#### To test a custom format exporter:

1. Select a custom format in the definition list.
2. Click **Test** in any of the four *Export Format Editor* dialogs. The dialog expands.
3. Click **Read File** and select the number of lines you want the exporter to read. If you select **View File**, it will open in a text editor.
4. Click the  icon and navigate to the type of file you want to export. The exporter will read the file and highlight any values that it is unable to convert.

**Note:** The file must have the same file extension as the exporter you chose.

5. Select a different exporter or edit the file to accommodate the reported errors.

#### Related commands

- ❑ [Definition Options](#) (on page 449)
- ❑ [Description and Search Type Options](#) (on page 450)
- ❑ [General Properties Options](#) (on page 451)
- ❑ [Fields Options](#) (on page 451)
- ❑ Export Data Formats
- ❑ [Selection Methods and Options](#) (on page 43)

### Definition Options

Use these options to create new export format definitions. These buttons appear to the right of the list of definitions on the first dialog of the *Export Format Editor*.

## Options

- New** Click this to enter a new definition name in the list. A unique name is required; a descriptive name is recommended. Click any other definition row to finish.
- Click **Next** to enter a [description and search type](#) (see "Description and Search Type Options" on page 450).
- Copy** After you click on a description (listing on left), click **Copy** to enter a definition name. A unique name is required; a descriptive name is recommended. Click **OK** to return.
- Click **Next** to enter a [description and search type](#) (see "Description and Search Type Options" on page 450).
- Rename** Select the name of one of the custom formats you have created, and click this to edit the name.
- Delete** After you click on a description (listing on left), click this to remove the definition from the list.

**Note:** To remove a description as an export option, you can click the **Enable** checkbox until no green check appears. These descriptions will not appear as export options. If you do not want to display these disabled descriptions on this page, enable **Only show enabled definitions** at the bottom left of the dialog box.

## Related topics

- ❑ [Description and Search Type Options](#) (on page 450)
- ❑ [General Properties Options](#) (on page 451)
- ❑ [Fields Options](#) (on page 451)
- ❑ [Export Data in a Custom Format](#) (on page 448)

## Description and Search Type Options

Use these options to define the type of custom exporter you want to create, and add a description. They are available in the second dialog of the **Export Format Editor**.

### Options

- Description** Enter a descriptive string to describe this exporter (optional).
- Type** Select one of the following options:
- Delimited** - this creates a file of data that is separated by a specific character.
  - Fixed Width** - this creates a file of data that is in pre-defined columns.
  - Search for Text** - this creates a file of data that begins and ends with a text string.

### Related topics

- ❑ [Definition Options](#) (on page 449)
- ❑ [General Properties Options](#) (on page 451)

- ❑ [Fields Options](#) (on page 451)
- ❑ [Export Data in a Custom Format](#) (on page 448)

## General Properties Options

Use these options to define how you want the file delimited and saved, and the data stored. They are available in the third dialog of the *Export Format Editor*.

### Options

<b>Delimiter</b>	From the drop-down list, select the character that is to separate the fields. If you select <other>, you must specify the required character. This can be one of the following: _ ) ( * & ^ % \$ # @ ! ~ `
<b>Default extension (recommended)</b>	Enter the default extension for the export format. The export analyzer uses this extension to help it decide what conversion options to offer the user in the context menu. This field is optional. If left blank, a default extension of ".txt" is assumed.
<b>Text qualifier (optional)</b>	Enter a special character to identify the beginning and ending of the string.
<b>Decimal separator</b>	Select a decimal separator if necessary. USA standard uses a point to separate the fractional number from the whole number; some areas in Europe use a comma as standard.

### Related topics

- ❑ [Definition Options](#) (on page 449)
- ❑ [Description and Search Type Options](#) (on page 450)
- ❑ [Fields Options](#) (on page 451)
- ❑ [Export Data in a Custom Format](#) (on page 448)

## Fields Options

Use these options to define the fields that you want to export, and their field order, and units. The options vary slightly based on the type of converter you are creating. They are available in the fourth dialog of the *Export Format Editor*.

## Options

<b>Fields</b>	<p>Click <b>Fields</b> to display a drop-down list of data properties. Select one and a tag appears as a field in the row of data. Continue to select all the fields that you want to export.</p> <hr/> <p><b>Note:</b> If you select the properties out of order, you can click and drag them into the proper order.</p> <hr/>
<b>Units Apply to all</b>	<p>To select the distance units for all data, select the units and enable the <b>Apply to all</b> check box. You can also disable the <b>Apply to all</b> check box, and select a unit for each exported field.</p>
<b>For Fixed Width (only)</b>	<p>Click on each field, and enter a <b>Start</b> and <b>End</b> value or a <b>Start</b> and <b>Width</b> value - the third value will be filled in automatically.</p>
<b>For Add Text (only)</b>	<p>Click on each field, and enter values with which to <b>Start</b> and <b>End</b> the field.</p> <hr/> <p><b>Note:</b> Spaces will not be visible in the <b>Start</b> and <b>End</b> fields, but you can see them in the <b>Preview</b> area.</p> <hr/>
<b>Test Preview</b>	<p>Click <b>Test</b> to open the testing display area. If there is selected point data, click <b>Preview</b> to see the format that the exporter would create. You can continue to modify the format setting and preview the results until you are satisfied.</p> <hr/> <p><b>Note:</b> If you have no data selected, click <b>Finish</b> to exit the <b>Export Format Editor</b> command, and save the exporter that you are creating. Then select some points and start the <b>Export Format Editor</b> again.</p> <hr/>

## Related topics

- ❑ [Definition Options](#) (on page 449)
- ❑ [Description and Search Type Options](#) (on page 450)
- ❑ [General Properties Options](#) (on page 451)
- ❑ [Export Data in a Custom Format](#) (on page 448)

# CHAPTER 18

## Troubleshoot Issues

### Troubleshoot a Coordinate System Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
You cannot create or edit coordinate systems, or save sites.	You are running as a Limited User (non-administrator). Limited users do not have “write” permissions for the <i>current.csd</i> file, which means that you cannot create or edit coordinate systems, or save sites.	<p>You must be granted “write” permissions for the <i>current.csd</i> file by an administrator. The location of that file depends on your operating system:</p> <ul style="list-style-type: none"> <li>▪ <b>In Windows® XP or earlier:</b> C:\Documents and Settings\All Users\Application Data\Trimble\GeoData or C:\Program Files\Common Files\Trimble\GeoData.</li> <li>▪ <b>In Windows Vista™:</b> C:\ProgramData\Trimble\GeoData or C:\Program Files\Common Files\Trimble\GeoData.</li> </ul> <hr/> <p><b>Note:</b> This may be a non-issue if other Trimble software has been previously installed, and access rights have been resolved.</p> <p><b>Note:</b> If you do not see the <i>Application Data</i> folder at the path listed above, it may be hidden. To show hidden folders, in Windows® Explorer, select <b>Tools &gt; Folder Options</b>. Click the <b>View</b> tab and select <b>Show hidden files and folders</b> in <i>Advanced Settings</i>. Then click <b>OK</b>.</p>


### Troubleshoot a Data Transfer/Synchronization Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
Active Sync 4.5 will not run.	You are running as a limited user. Active Sync 4.5 is not compatible with limited user accounts.	Change you permissions to the administrator level, or download and use Active Sync 4.0. <b>Start &gt; Control Panel &gt; User Accounts &gt; User Accounts.</b> In the <i>User Accounts</i> dialog, select your user name in the list and click <b>Properties</b> . In the <i>Properties</i> dialog, click the <b>Group Membership</b> tab. Select <b>Other</b> , and <b>Administrators</b> in the list. Click <b>OK</b> twice to close the dialogs.

## Troubleshoot a Layer or View Filter Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
The graphic view redraws slowly when you make changes to view filters.	Your project contains a lot of data, and your computer's graphics memory is running at capacity.	Click the  icon on the <i>View Filter Manager's</i> toolbar to open the <i>Advanced View Filter Settings</i> dialog. When you make changes to view filters in this dialog, the graphic view does not automatically redraw. You can click <b>Apply</b> at any time to have the view redraw.  or  Select <b>Project &gt; Project Settings</b> . Click <b>View</b> and then <b>View Filters</b> in the left pane, and select a default view filter other than <i>All</i> so that graphic views refresh more quickly.
The points on a layer are not visible, even though you have the layer's box checked in the <i>View Filter Manager</i> to make it visible.	The <i>Point</i> box in the <i>Raw Data</i> group is not checked.	Make sure that the boxes for both the points and the layer that the points are on are checked, making them visible in the view.
Some of the selection sets you created do not appear in the <i>View Filter Manager's</i> selection sets list.	The missing selection sets that do not contain any visible objects.	None. Selection sets can contain objects that have no visible display, such as coordinates. In the <i>Selection Explorer</i> , all selection sets are available. In the <i>View Filter Manager</i> , however, only selection sets that contain at least one visible object are available.

### Related topics

- ❑ [Create a View Filter](#) (on page 73)
- ❑ [Edit a View Filter](#) (on page 75)
- ❑ [Filter a View](#) (on page 76)
- ❑ [View Filter Options](#) (see "View Filter Manager Options" on page 79)
- ❑ [Advanced View Filter Options](#) (on page 81)

## Troubleshoot an Import Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
Duplicate points were created for points in an imported text file and points already in the project that have the same ID (that is, points were not merged as expected).	If you import a text file with "Unknown" or "Mapping" coordinate quality into a project that already contains point data, duplicate points will be created for points in the text file ( <a href="#">lightweight points</a> (see "Understanding Point Types" on page 269)) and points already in the project ( <a href="#">normal points</a> (see "Understanding Point Types" on page 269)) that have the same ID.	Import the text file into the project first to create the lightweight points, then import the other point data. The lightweight points from the text file will merge with the normal points from the other point data to create normal non-duplicated points.

## Troubleshoot a Program Freeze

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
Program freezes when selecting many objects.	The <b>Properties</b> pane is open. When you select many objects with the <b>Properties</b> pane open, it looks for the properties common to all of the objects, slowing the program down and making it look frozen.	Close the program. If you have trouble reopening the project, check the directory where the .vce file is stored. If there is a lock (*.lk) file with the same name as the project, delete it and reopen the project. Close the <b>Properties</b> pane before reselecting the objects.

<p>Program doesn't respond in the expected way; nothing seems to work.</p>	<p>The mouse may be set to a mode other than <b>Select</b>.</p>	<p>Check your mouse mode on the <b>Mouse</b> toolbar. If needed, reset it to <b>Select</b>.</p>
<p>The program freezes.</p>	<p>Toolbars are corrupted.</p>	<p>Consider contacting Technical Support.          Otherwise, remove the application data folder located at <b>C:\Documents and Settings\<user name="">\Application Data\Trimble\Trimble Business Center\<version&gt;< b=""></version&gt;<></user></b></p>
<p><b>Note:</b> If you do not see the <b>Application Data</b> folder at the path listed above, it may be hidden. To show hidden folders, in Windows® Explorer, select <b>Tools &gt; Folder Options</b>. Click the <b>View</b> tab and select <b>Show hidden files and folders</b> in <b>Advanced Settings</b>. Then click <b>OK</b>.</p>		
<p>The program appears to freeze when you float a pane or try to open a dialog.</p>	<p>If you are running the program on a secondary monitor, and you float a pane or use a command that launches a dialog, the pane or dialog might appear out of either monitor's visible range. It will be located off of the primary monitor, in the space opposite the secondary monitor, causing Trimble Business Center to appear 'frozen'.</p>	<p>To reach the dialog or pane, right-click the application's name on the Windows Taskbar and select <b>Move</b>. Then, press the appropriate arrow key to move the dialog into your primary's monitor's visible range.</p>
<p>Program appears to freeze when you are trying to e-mail SCS files.</p>	<p>If you are running the program on a secondary monitor, and you attempt to e-mail SCS files using the <b>Compress/E-mail SCS Files</b> command, your e-mail program may open a dialog confirming the operation out of either monitor's visible range. The dialog will be located off of the primary monitor, in the space opposite the secondary monitor, causing the program to appear 'frozen'.</p>	<p>To reach the dialog and confirm the e-mail operation, right-click the e-mail application's name on the Windows Taskbar and select <b>Move</b>. Then, press the appropriate arrow key to move the dialog into your primary's monitor's visible range.</p>



## Troubleshoot a Project Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
You cannot reopen a .vce project file.	<p>The project file is locked, possibly due to a crash or an improper program shutdown.</p> <p>Improper shutdown includes when:</p> <ul style="list-style-type: none"> <li>▪ A project is open and the power to the computer is interrupted.</li> <li>▪ A project is open and the process is ended from Windows® Task Manager.</li> </ul>	<p>Delete the lock (project name.lk) from the file project folder.</p> <p>The project will lose all changes made since the last save.</p>


## Troubleshoot a Toolbar or Menu Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
Toolbars are not in the same language as the installation.	The program was run in one language, and then reinstalled in a different language.	<p>Reset the toolbars to the reinstalled language by loading the default layout.</p> <p>Select <b>Tools &gt; Customize</b>. In the <b>Customize</b> dialog, click <b>Save/Load</b>. In <b>Default Layout</b>, select <b>Default Layout</b> and click <b>Load</b>. Click <b>OK</b> in the <b>New layout</b> dialog.</p>
<p>Some text in the user interface (the units on the Status bar for instance) is in a different language.</p> <p>This only occurs when opening project files that were created in a different language.</p>	The user interface text is being stored in the project file.	<p>There is no solution at this time.</p> <p>Newly created projects will not have the problem, but it will remain in the original project file.</p>

## Troubleshoot a View or Selection Problem

Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
The graphic view redraws slowly when you make changes to view filters, or when you change the size of panes.	Your project contains a lot of data, and your computer's graphics memory is running at capacity.	Click the  icon on the <b>View Filter Manager's</b> toolbar to open the <b>Advanced Settings</b> dialog. When you make changes to view filters in this dialog, the graphic view does not automatically redraw. You can click <b>Apply</b> at any time to have the view redraw.  or Select <b>Project &gt; Project Settings</b> . Click <b>View</b> and then <b>View Filters</b> in the left pane, and select a default view filter other than <b>All</b> so that graphic views refresh more quickly.
The 3D view is replaced by a red X and this message: "The system has run out of graphics memory. Close any unnecessary windows and retry."	The system has run out of graphics memory. (no screen saver interruption)	1. Close any unneeded programs that are running, especially ones that are graphics intensive. 2. Close all 3D views, including those with a red X. 3. Reopen the minimum number of 3D views you need.  Long-term: Consider upgrading your graphics card.
The 3D view is replaced by a red X and this message: "The system has run out of graphics memory. Close any unnecessary windows and retry."	The screen saver came on and interrupted Microsoft® DirectX.	<ul style="list-style-type: none"> <li>▪ Close the 3D view and reopen it.</li> <li>▪ Close and reopen the program and the project.</li> <li>▪ Update to the latest version of DirectX.</li> </ul>
Some of the selection sets I created do not appear in the View Filter Manager's selection sets list.	The missing selection sets that do not contain any visible objects.	None. Selection sets can contain objects that have no visible display, such as coordinates. In the Selection Explorer, all selection sets are available. In the View Filter Manager, however, only selection sets that contain at least one visible object are available.

Your graphic views are pixilated or contain artifacts when you pan or rotate them.	You are not using the optimal advanced display setting.	Select <b>Tools &gt; Options</b> . In the <b>Options</b> dialog, click <b>Startup and Display</b> in the left pane, and then click <b>Advanced</b> . Check the <b>Override automatic detection</b> box, and select the appropriate option for your operating system.
Your mouse movements are delayed or track intermittently, even though you are using an advanced display setting (see above).	Your graphics card is integrated into the motherboard, or is not sufficient for advanced display settings.	Select <b>Tools &gt; Options</b> . In the <b>Options</b> dialog, click <b>Startup and Display</b> in the left pane, and then click <b>Advanced</b> . Check the <b>Override automatic detection</b> box, and try each of these graphics display packages (in order, restarting the program between each):  <ol style="list-style-type: none"> <li>1. DirectX</li> <li>2. OpenGL</li> <li>3. GDI</li> </ol>
Your system suffers from generally poor graphics performance.	Two system settings are not set to optimize the graphics display.	Try the solution directly above first. If it does not improve your graphics performance, right-click on your Windows desktop, and select <b>Properties</b> from the context menu. In the <b>Display Properties</b> dialog, click the <b>Settings</b> tab, and select <b>Medium (16 bit)</b> in the <b>Color quality</b> list. Second, click the <b>Advanced</b> button in the same dialog. In the <b>Plug and Play</b> dialog, click the <b>Troubleshoot</b> tab, and set <b>Hardware acceleration</b> to <b>Full</b> .

## Troubleshoot an Import Problem

Before calling Support, use any applicable solutions to known issues below.

<b>Symptom</b>	<b>Possible Cause</b>	<b>Solution</b>
LandXML data imports in the wrong location or configuration.	Project units are not set correctly.	Check the units shown on the status bar. If they are not correct, undo the import. Then, click the units name to open the <b>Project Settings</b> dialog. Change to the correct type of units, and reimport the file.
A LandXML file will not import. The program's importer says it is a LandXML file, but when	The file is valid XML, but not valid LandXML.	Open the file in your default Web browser. If it is corrupt, you will get an error message. If you do not get a message, it may be valid


<p>you try to import it, a message says the file is invalid.</p>	<p>XML, but not valid LandXML, in which case the file needs to be recreated in a valid LandXML format.</p>
<p>Custom point data (.csv) imports at the wrong location.</p>	<p>1. Your project units are not set correctly. 2. The wrong custom import definition was selected; the northing and easting order are reversed.</p> <p>1. Make sure that your project units are set correctly. 2. Check the custom import definition you used to import the points. If you accidentally selected the definition with the northing and easting reversed, undo the import and reimport with the correct definition.</p>
<p>Duplicate points were created for points in an imported text file and points already in the project that have the same ID (that is, points were not merged as expected).</p>	<p>If you import a text file with "Unknown" or "Mapping" coordinate quality into a project that already contains point data, duplicate points will be created for points in the text file (<a href="#">lightweight points</a> (see "Understanding Point Types" on page 269)) and points already in the project (<a href="#">normal points</a> (see "Understanding Point Types" on page 269)) that have the same ID.</p> <p>Import the text file into the project first to create the lightweight points, then import the other point data. The lightweight points from the text file will merge with the normal points from the other point data to create normal non-duplicated points.</p>

## Troubleshoot an SCS Site or File Problem



Before calling Support, use any applicable solutions to known issues below.

Symptom	Possible Cause	Solution
<p>The program cannot find the SCS900 report utility .xls file. An error message states that it cannot find the file or the full file extension and if you browse to it the file is correct. Best to keep on</p>	<p>Paths to Microsoft® Excel files are limited to 219 characters. In Trimble® Business Center- Heavy Construction Edition, this means that if the path to the SCS report utility file (named <b>SCS900 Reporting</b></p>	<p>Move Excel (.xls) files to locations with shorter paths (fewer subfolders), or rename files with shorter names. Unzipping a folder to a location within several folders could exceed maximum path character limits, preventing deeper files from being unzipped. There may be a warning that the files cannot be unzipped or</p>

<p>the C drive.</p>	<p><b>Utility.xls</b> or <b>SCS Report Utility.xls</b>) exceeds the limit, the file will not open. In most cases, this will only happen if the .xls file has been placed in the '<b>My Documents</b>' directory.</p>	<p>the windows may simply close, making it look like they were all unzipped. It may be best to unzip files directly to the C drive with full administrative rights.</p>
<p>The SCS900 report utility .xls file does not work.</p> <p>An error message says that there is a mismatch in data when the report is opened.</p> <hr/> <p><b>Note:</b> If the Import Record.xls is opened directly in Excel, there is a no warning until you click <b>Import Record</b>.</p>	<p><b>Note:</b> The number of the cause corresponds to the number of the solution.</p> <ol style="list-style-type: none"> <li>1. The <b>-Report.txt</b> and <b>-Record.txt</b> files are not both present or in the proper format.</li> <li>2. Language settings do not match in the operating system's regional settings, Trimble Business Center, Microsoft Office, and the SCS Report Utility file.</li> <li>3. The synchronizer root folder is not set to the correct folder in the <b>Options</b> dialog.</li> <li>4. The user does not have privileges to the folder.</li> <li>5. The Report Utility file is a different version from the one used on the site controller.</li> </ol>	<ol style="list-style-type: none"> <li>1. Ensure that both the <b>-Report.txt</b> and <b>-Record.txt</b> are present and in the proper format in the data synchronization area's <b>Output</b> folder.</li> <li>2. Ensure that language settings match in the operating system's regional settings, Trimble Business Center, Microsoft Office (apply a language pack if necessary), and the SCS report utility file.</li> <li>3. In this program, select <b>Tools &gt; Options</b>. Click <b>SCS</b> and then <b>File Locations</b> in the left pane. Click the  icon and navigate to the folder in which SCS installed the <b>SCS Report Utility.xls</b> file.</li> <li>4. If the security level is changed in Excel, then a new Excel program needs to be started for the macro to work. To change the security level in Excel, open <b>Tools &gt; Options &gt; Security &gt; Macro Security</b>, and set the <b>Security Level</b> to <b>Low</b> (or add to the <b>Trusted Publishers</b> list). The macro can then be disabled when the report utility is opened.</li> <li>5. Find and use, or create, a report utility file of the same version as the one used on the site controller. The utility maker is on the SCS installation CD.</li> </ol>

<p>The program appears to freeze when trying to e-mail SCS files.</p>	<p>If you are running the program on a secondary monitor, and you attempt to e-mail SCS files using the <b>Compress/E-mail SCS Files</b> command, your e-mail program may open a dialog confirming the operation out of either monitor's visible range. The dialog will be located off of the primary monitor, in the space opposite the secondary monitor, causing the program to appear frozen.</p>	<p>To reach the dialog and confirm the e-mail operation, right-click the e-mail application's name on the Windows taskbar and select <b>Move</b>. Then, press the appropriate arrow key to move the dialog into your primary's monitor's visible range.</p>
<p>Label text in the <b>Work Order Viewer's</b> coverage map is blurry.</p>	<p>In Windows display properties, effects are set to <b>Clear Type</b>.</p>	<p>Smooth the edges of the screen fonts in the viewer. Right-click on your Windows desktop and select <b>Properties</b> from the context menu. In the <b>Display Properties</b> dialog, click the <b>Appearance</b> tab and click <b>Effects</b>. In the <b>Effects</b> dialog, select <b>Standard</b> in the second drop-down list.</p>
<p>You receive a message that your e-mail program is not supported when you try to e-mail SCS files.</p>	<p><b>Note:</b> The number of the cause corresponds to the number of the solution.</p> <ol style="list-style-type: none"> <li>1. This command requires Microsoft® Outlook® to work. Other email programs may work, but are not guaranteed. Web-based email programs, such as Microsoft Hotmail, Yahoo®Mail or Google®Gmail™, will not work.</li> <li>2. If you do have Outlook, it may be missing a component file.</li> </ol>	<ol style="list-style-type: none"> <li>1. If your email program is not supported, create the compressed file, save it, open your email program, attach the file to a message, and send it.</li> <li>2. Download the Outlook CDO (Collaboration Data Objects) file to fix the problem:  <a href="http://www.microsoft.com/downloads/details.aspx?FamilyId=2714320D-C997-4DE1-986F-24F081725D36&amp;displaylang=en#QuickInfoContainer">http://www.microsoft.com/downloads/details.aspx?FamilyId=2714320D-C997-4DE1-986F-24F081725D36&amp;displaylang=en#QuickInfoContainer</a>  <a href="http://www.microsoft.com/downloads/details.aspx?FamilyId=2714320D-C997-4DE1-986F-24F081725D36&amp;displaylang=en#QuickInfoContainer">http://www.microsoft.com/downloads/details.aspx?FamilyId=2714320D-C997-4DE1-986F-24F081725D36&amp;displaylang=en#QuickInfoContainer</a>                      For Microsoft Outlook 2003, install the CDO file from the original program CD.</li> </ol>

# CHAPTER 19

## Use Related Utilities

### Coordinate System Manager

The *Coordinate System Manager* is a standalone utility that gives you access to your coordinate system database (Current.csd). Use the manager to create coordinate systems, or to determine which coordinate systems, geoid models, and sites are available for use in your project.

#### To open the Coordinate System Manager:

- Select **Tools > Coordinate System Manager**.

**Note:** The *Coordinate System Manager* has its own help system. Open the utility and select **Help > Help Topics**, or press **[F1]** within the software.

#### Related topics

- ❑ [Change the Coordinate System](#) (on page 149)
- ❑ [Coordinate Systems](#) (see "Understanding Geodetic Reference Data" on page 170)
- ❑ [Define a New Coordinate System](#) (on page 149)
- ❑ [Restore the Original Coordinate System File](#) (on page 150)

### Feature Definition Manager Utility

The *Feature Definition Manager* is a standalone utility that gives you the ability to create and manage feature libraries (.fxl files) for feature code processing and GIS attribute data collection. A feature library is a collection of features with codes and attributes that describe them, as well as line control codes that modify how the features relate.

The *Feature Definition Manager* comes with a default library of features with predefined attributes. This library provides a good starting point for feature coding. As you create new features and edit existing ones, the library will become suited to the specific needs of your projects.

Feature coding in the editor enables you to:

- Make detailed data collection in the field more efficient and consistent by controlling how features and attributes can be captured. Setting parameters for what you can and must enter ensures data integrity and completeness.
- Add symbols and annotations to feature-coded field data so that the information can be presented in a more visual format.
- Connect points to define line features, such as pavement or building edges, or the centerlines of ditches or fences. Line control codes give you the power to add new points automatically, and add lines, curves, and arcs between points.

If you are working with surfaces, coded features also let you:

- Define the breaklines of a surface.
- Control how surfaces are formed by specifying which points should be used, and which lines should act as breaklines. Surfaces can be modified by moving points to specific layers based on their feature codes.

**Note:** The *Feature Definition Manager* has its own help system. While in the *Feature Definition Manager*, select **Help > Contents**, or press [F1].

#### Related topics

- [Understanding Feature Data](#) (on page 359)
- [Workflow for Feature Data](#) (on page 361)

## Planning Utility

Use the *Planning* utility to plan and schedule a GPS project based on good and bad satellite coverage information.

#### To access the utility:

- Select **Tools > Planning**.

**Note:** The *Planning* software has its own help system. Open the utility and select **Help > Index** from the *Planning* menu or press [F1] within the software.

## External Tools Manager

Use the *External Tools Manager* to add menu items for external application and utilities that you might want to use within this software. For example, if you want to have quick access to the Microsoft® Windows® Calculator, you can add it as a menu item. To add external tools to the *Tools* menu, see [Customize the Menu](#) (on page 16).



## SCS Data Manager

### HEAVY CONSTRUCTION

The SCS Data Manager is a program that lets you create sites, organize site and design data, write work orders, and assign the data and work orders to controllers in preparation for Trimble® site controller software (SCS).

SCS enables you to measure material volumes, monitor grades and laid material thickness, and perform other site-related tasks, such as roadway point, line, and surface stakeout.

**Note:** Most of the SCS Data Manager's functions have been embedded in the **SCS Site Manager** in this software.

#### To open the SCS Data Manager:

Do one of the following:

- Select **SCS > SCS Data Manager**.
- Click the  icon on the toolbar.

#### Related topics

- ❑ [Manage a Site and Site Data](#) (on page 377)
- ❑ [Workflow for Creating a Site](#) (on page 377)
- ❑ [Workflow for Using an External Site](#) (on page 378)

## Trimble Configuration Utility

Use the *Trimble Configuration Utility* to update your computer with the latest files for:

- GPS antennas
- GPS receivers
- GPS antenna model files

After running the utility, your Trimble software will support the latest GPS hardware.

#### To access the utility:

- Select **Tools > Configuration Web Page**.

## Trimble Data Transfer Utility

Use the *Trimble Data Transfer Utility* to transfer data from a range of devices to your computer. You can then import the data into Trimble Geomatics Office™, Trimble Total Control™, Terramodel®, Trimble Business Center, GPS Pathfinder® Office, Trimble Link™, or the GPS Analyst™ extension for ESRI ArcGIS.

**To access the utility:**

- Select **Tools > Data Transfer Web Page**.

# Index

## 2

2D View Navigation • 35

## 3

3D View • 28

3D View Navigation • 37

3D View Settings • 38

## A

Abbreviation Settings • 158

Add a Coordinate to a Point • 272

Add an External Design • 394

Add an External Site • 380

Add and Edit a Design Model • 398

Add and Edit Points and Coordinates • 270

Add or Remove Background Map Objects • 387

Add or Remove Control Points • 386

Add or Remove Foreground Map Objects • 401

Add or Remove Stakeout Points • 399

Add Surface Materials • 349

Adjust Level Runs • 265

Advanced Display Options • 141

Advanced View Filter Options • 81

Alignment Editor • 30, 299

Alignment Properties • 296

Alignment Stationing Options • 295

Along Line Snap • 128

Archive a Project • 166

At Point Snap • 127

## B

Bearing + Angle Snap • 96

Bearing Bearing Snap • 99

Bearing Distance Snap • 100

Boundary Options • 341

Break a Line • 314

Breakline Options • 314, 333

## C

Calculate and Enter Values • 86

Calculate the Inverse Between Points • 230, 280

Calibrate a Site • 181, 184

Center of Arc Snap • 101

Center of Gravity Snap • 102

Change Project Units • 152

Change the Coordinate System • 149, 171

Change the Default Template • 164

Change the Gridline Display • 153

Change the Template Folder • 144

Check GNSS Data • 229

Check Imported Surfaces • 321

Choose Application Options • 139

Choose Local Site Settings • 159

Choose Project Settings • 147

COGO Expressions, Units, and Entry Formats • 87

Command Pane • 13

Compress and E-mail SCS Files • 421

Computational Settings • 156

Contour Options • 347

Convert CAD Points to Survey Points • 400

Coordinate Options • 274

Coordinate System Manager • 173, 463

Coordinate System Settings • 148

Coordinates Scroll • 15

Copyright and Trademarks • ii

Corporate Office • ii

Create a Boundary • 340

Create a Datum Grid File • 174

Create a Design • 393

Create a Linestring • 300

Create a New Project • 161

- Create a Point • 270
  - Create a Project Template • 163
  - Create a Site • 379
  - Create a Surface • 323
  - Create a Surface Boundary and Contours • 339
  - Create a Surface Contour at an Elevation • 344
  - Create a Surface Cut/Fill Grid • 346
  - Create a View Filter • 73
  - Create a Work Order • 409
  - Create an Alignment • 288
  - Create an Alignment from a GENIO String • 297
  - Create and Edit a Layer • 68
  - Create and Edit a Linestring • 300
  - Create and Edit a Simple Breakline • 312, 331
  - Create and Edit a Surface • 320
  - Create and Edit an Alignment • 286
  - Create and Use Selection Sets • 57
  - Create and View a Surface Cross-Section • 326
  - Create and View a Surface Profile • 325
  - Create Surface Contours at Intervals • 343
  - Cross-Section View • 26
  - Customization Options and Tools • 21
  - Customize a Report • 436
  - Customize the Keyboard • 20
  - Customize the Menu • 16
  - Customize the Toolbar • 18
- D**
- Data Selection • 42
  - Data Synchronization • 223
  - Data View Display Formats • 32
  - Data Views • 16
  - Datum Grid Options • 176
  - Define a Geoid Sub-Grid • 177
  - Define a New Coordinate System • 149, 171
  - Define a Projection • 176
  - Define Materials for Earthwork Reports • 351
  - Define the Coordinate System • 171
  - Definition Options • 210, 449
  - Deflection Angle Snap • 92
  - Delete a Line Segment • 316
  - Delete an Object • 65
  - Delta X Delta Y Snap • 103
  - Description and Search Type Options • 211, 450
  - Design Options • 396
  - Device Pane • 11, 219
  - Direct Connection • 221
  - Distance + Distance Snap (distance) • 121
  - Distance + Distance Snap (offset) • 129
  - Distance Distance Snap • 103
  - Distance Snap • 130
  - Download Files (via data synchronization) • 223
  - Download Files (via Direct Connection) • 221
  - Drag and Drop to Import • 194
- E**
- Earthwork Report Options • 354
  - Edit a Linestring's Horizontal Segments • 301
  - Edit a Linestring's Vertical Control Points • 309
  - Edit a Site's Name and Settings • 383
  - Edit a Surface by Adding and Removing Members • 328
  - Edit a Surface by Changing a Point Coordinate • 329
  - Edit a Surface by Changing Its Properties • 338
  - Edit a Surface by Creating a Breakline • 334
  - Edit a Surface by Trimming Edge Triangles • 336
  - Edit a View Filter • 75
  - Edit a Work Order • 410
  - Edit an Alignment • 290
  - Edit an Alignment's Properties • 295
  - Edit an Object • 66
  - Edit Design Data • 395
  - Edit Site Data • 384

Edit the Point ID for a Station Setup and/or Backsight • 278

End Snap • 104

Enter a Bearing • 94

Enter a Coordinate • 97

Enter a Distance • 120

Enter a Station • 133

Enter an Angle • 92

Enter an Elevation • 125

Enter an Offset • 128

Enter, Edit, and Delete Feature Code Strings • 363

Explode a Block • 319

Explore an Object • 318

Export and Upload Data Formats • 442

Export ASCII Files • 443

Export CAD Files (.dxf/.dwg) • 444

Export Data • 441

Export Data in a Custom Format • 448

Export Geodatabase Files (.xml) • 368, 445

Export LandXML Files (.xml) • 445

Export Related Files • 442

Export Trimble Data Collector Files (.dc) • 448

Export Trimble Surface Files (.ttm) • 448

External Tools Manager • 464

## F

Factor of Distance Snap (distance) • 122

Factor of Distance Snap (offset) • 131

Factor of Line Snap • 105

Factor of Segment Snap • 106

Feature Code Processing Settings • 157, 362

Feature Definition Manager Utility • 368, 463

Fields Options • 213, 451

File Location Options • 142

Filter a View • 76

Find Help Topics • 22

Flags Pane • 14

Free Snap • 107

From Surface Snap • 126

## G

General Information Settings • 147

General Properties Options • 212, 451

Geoid Options • 177

Geoid Sub-Grid Options • 179

Get Familiar with the Interface • 6

Get Started • 3

Graphic Selection Methods • 43

Graphic Views • 25

## H

Help Options • 23

Horizontal Alignment Options • 291

Horizontal Linestring Segment Options • 305

## I

Import ASCII Files • 195

Import CAD Files (.dxf/.dwg) • 196

Import Data • 193

Import Data Collector Files (.dc) • 196

Import Data in a Custom Format • 209

Import DiNi Digital Level Files (.dat) • 207, 253

Import GENIO Files • 198

Import GNSS Job Files (.job) • 199

Import LandXML Files (.xml) • 200

Import MicroStation Files (.dgn) • 204

Import REB Files (.reb) • 206

Import Trimble Surface Files (.ttm) • 207

Import Wirth YXZ Files (.yxz) • 207

Import Work Order Results • 422

Importable Data Formats • 195

Include a Site Calibration • 385

Intersection of Lines Snap • 108

Intersection of Offset Lines Snap • 109

Intersection of Offset Segments Snap • 109

Intersection of Segments Snap • 110

Inverse Options • 231, 281

Isolate or Exclude a Layer • 70

## J

Join Lines • 315

## K

Keyboard Navigation • 40

## L

LandXML Conflict Resolution Options • 203

LandXML Export Options • 445

Layer Options • 71

Level Data Errors • 264

Local Site Setting Options • 160

## M

Manage a Site and Site Data • 377

Manage Designs and Design Data • 393

Manage the Data in Your Views • 67

Manage Work Orders and Results • 406

Materials Options • 352

Measure Angle Options • 138

Measure Angles • 137

Measure Options • 136, 283

Measure Values Between Points • 135, 282

Merge Duplicate Points • 278

Merge Level Runs • 266

Merge Points • 279

Middle of Point to Point Snap • 111

Middle of Segment Snap • 112

Modify Selection Sets • 58

Mouse Modes • 42

## N

Nearest to Line Snap • 113

Note on Level Runs Without Benchmarks • 267

## O

Office Synchronizer • 219

Offset at Point Snap • 132

Offset Line Snap • 113

Offset Segment Snap • 114

Open an Existing Project • 164

## P

Pane and Data View Positioning • 31

Perpendicular to Line Snap • 115

Perpendicular to Segment Snap • 115

Plan View • 25

Planning Utility • 229, 464

Point Coordinate Options • 275, 330

Point of Intersection Snap • 116

Point Options • 271

Point Snap • 117

Point to Point Bearing Snap • 97

Point to Point Snap (distance) • 123

Point to Point Snap (offset) • 132

Points Spreadsheet • 29, 284

Prepare Data for Site Positioning Systems • 370

Prepare to Connect a Field Device • 217

Print a View or Report • 166

Process Feature Codes • 367

Profile View • 26

Project Computations • 157

Project Explorer • 7

Properties Pane • 14

## R

Radius of Arc Snap (distance) • 124

Radius of Arc Snap (offset) • 133

Reassign a Design to Controllers • 405

Reassign a Site to Controllers • 388

Reassign or Copy a Work Order • 420

Reference

Regular Expressions • 214

Register This Software • 3

Release Notice • ii

- Rename Points • 276
  - Report Options • 438
  - Report View • 31, 437
  - Resolve a Conflict in Background Maps • 392
  - Resolve a Conflict in Control Points • 391
  - Resolve a Conflict in Design Models • 402
  - Resolve a Conflict in Foreground Maps • 404
  - Resolve a Conflict in Site Calibrations • 390
  - Resolve a Conflict in Site Settings • 388
  - Resolve a Conflict in Stakeout Points • 403
  - Resolve LandXML Conflicts • 202
  - Restore the Original Coordinate System File • 150, 172
  - Results of Default Folder Locations • 145
  - Results of Exporting LandXML Files • 446
  - Retaining User Settings When Upgrading • 5
  - Rules for Merging Feature Attributes • 365
  - Run a Mean Angle Report • 249, 431
  - Run a Point Derivation Report • 230, 432
  - Run a Point List Report • 229, 432
  - Run a Project Computation Report • 157, 433
  - Run a Renamed Point List • 277, 433
  - Run a Site Calibration Report • 192, 434
  - Run a Surface Report • 339, 434
  - Run a Work Order Report or Summary • 423, 435
  - Run an Alignment Geometry Report • 298, 429
  - Run an Earthwork Report • 353, 430
  - Run an Import Report • 215, 431
  - Run Reports • 429
  - Running Snap Mode Options • 90
- S**
- Save a Calibrated Site as a Coordinate System • 190
  - Save a Project • 165
  - Scale-Only Projection • 174
  - SCS Data Manager • 376, 465
  - SCS File Compression and Email Options • 421
  - SCS File Location Options • 146
  - Select by Elevation Range • 52
  - Select By Layer • 53
  - Select Duplicate Points • 52
  - Select from 2D Views • 44
  - Select from Spreadsheet Views • 46
  - Select from the 3D View • 45
  - Select from the Flags Pane • 48
  - Select from the Project Explorer • 48
  - Select Points • 49
  - Select Using Advanced Criteria • 53
  - Selection Explorer • 8, 56
  - Selection Explorer and Selection Sets • 54
  - Selection Explorer Options • 60
  - Selection Methods and Options • 43
  - Separate Global and Grid/Local Coordinate Points for Site Calibration • 191
  - Set a Line Elevation • 317
  - Set Default File Locations • 144
  - Set Running Snap Modes • 89
  - Set the Pick Aperture • 88
  - Set Up Geodetic Reference Data • 170
  - Set Up Projects • 139
  - Shrink 3D Faces • 356
  - Site Calibration Options • 187
  - Site Options • 381
  - Snaps Modes and Commands • 89
  - Split Line Features • 365
  - Spreadsheets and Other Views • 29
  - Startup and Display Options • 139
  - Station at Point Snap • 134
  - Station from Line Snap • 135
  - Status Bar • 10
  - Surface Cut/Fill Grid Options • 348
  - Surface Options • 324
  - Surface Properties • 338
  - Surface Slicer View • 27

## T

Tabbed View Arrangement • 34  
 Tangent Snap • 118  
 Three Point Snap • 93  
 Toggle Line Marking • 155  
 Total Station Data Errors • 247  
 Total Station Measurement Types • 236  
 Total Station Nodes Sequence in Project Explorer • 246  
 Total Station Setup Types • 234  
 Transfer/Synchronize Data • 217  
 Trimble Configuration Utility • 465  
 Trimble Data Transfer Utility • 466  
 Troubleshoot a Coordinate System Problem • 173, 453  
 Troubleshoot a Data Transfer/Synchronization Problem • 228, 453  
 Troubleshoot a Layer or View Filter Problem • 83, 454  
 Troubleshoot a Program Freeze • 168, 455  
 Troubleshoot a Project Problem • 167, 457  
 Troubleshoot a Toolbar or Menu Problem • 21, 457  
 Troubleshoot a View or Selection Problem • 84, 457  
 Troubleshoot an Import Problem • 215, 285, 455, 459  
 Troubleshoot an SCS Site or File Problem • 425, 460  
 Troubleshoot Issues • 453

## U

Undefined Snap • 126  
 Understanding Alignments • 286  
 Understanding Breaklines • 330  
 Understanding COGO Controls • 86  
 Understanding Earthwork Volume Calculations • 349  
 Understanding Feature Data • 359  
 Understanding Geodetic Reference Data • 170

Understanding Layers and View Filters • 67  
 Understanding Level Data • 251  
 Understanding Point Types • 269  
 Understanding Selection Sets • 54  
 Understanding Site Calibration • 181  
 Understanding Sites, Designs, Work Orders, and Synchronization • 373  
 Understanding SPS and SCS • 372  
 Understanding Surfaces • 320  
 Understanding Total Station Data • 233  
 Undo or Redo an Action • 67  
 Unit Settings • 151  
 Upload a Datum Grid File • 227  
 Upload a Geoid File • 228  
 Upload Files (via data synchronization) • 224  
 Upload Files (via Direct Connection) • 221  
 Upload Geodetic Reference Data • 227  
 Upload Tasks (via data synchronization) • 225  
 Upload Tasks (via Direct Connection) • 222  
 Use a Datum • 174  
 Use a Geoid • 177  
 Use a Project Template • 162  
 Use Related Utilities • 463  
 Use Snap Commands • 91  
 Use Terramodel (.pro) Data • 208  
 Use Valid Segment Order • 289  
 User Guide • i

## V

Vertical Alignment Options • 293  
 Vertical Linestring Options • 310  
 View a Slice of a Surface • 327  
 View a Work Order's Data Graphically • 416  
 View and Edit an Object's Properties • 64  
 View and Edit Level Data • 255  
 View and Edit Mean Angle Residuals • 248  
 View Construction Data in Project Explorer • 427  
 View Filter Manager • 9, 72



View Filter Manager Options • 79

View Imported Level Data in Project Explorer • 261

View Imported Total Station Data in Project Explorer • 241

View Level Data Associated with a Point in Project Explorer • 263

View Level Data in Project Explorer • 260

View Settings • 152

View Total Station Data Associated with a Point in Project Explorer • 244

View Total Station Data in Project Explorer • 240

View, Navigate, and Select • 25

## W

Welcome to Trimble Business Center - Heavy Construction Edition • 1

Work Order Options • 411

Work Order Viewer Options • 417

Work with Feature Data • 359

Work with GNSS Data • 229

Work with Level Data • 251

Work with Line Data • 286

Work with Point Data • 269

Work with Surface Data • 320

Work with Total Station Data • 233

Workflow for Creating a Site • 377

Workflow for Creating Alignments • 287

Workflow for Creating Surfaces • 322

Workflow for Feature Data • 361

Workflow for Level Data • 252

Workflow for Managing Work Orders • 406

Workflow for Total Station Data • 238

Workflow for Using an External Site • 378

Workflow for Using Imported Alignments • 287

Workflow for Using Imported Surfaces • 321

## X

XY Snap • 119