

Creating and Editing Drawings Using SmartSketch[®]



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SECTION 1

SmartSketch Environment

Before beginning a design session in SmartSketch, the user should be familiar with its interface tools and working environment.

Topics

- SmartSketch and SmartSketch Drawing Editor
- Invoking SmartSketch
- SmartSketch Working Environment
- Toolbars and Ribbon Bars
- Symbol Explorer
- SmartSketch Document Windows
- Accessing SmartSketch Help

SmartSketch and SmartSketch Drawing Editor

What is SmartSketch?

SmartSketch is a 2D CAD application sold standalone by Intergraph. SmartSketch generates files with the .igr file extension; however, it is capable of reading/writing many different file types. The underlying graphics platform for SmartSketch is called RAD, and multiple different Intergraph applications are built using RAD. So, if you are using data from an application built with RAD, you can likely use SmartSketch to read/write the files.

Based on this capability, many people use SmartSketch as a generic CAD tool to view/edit different types of application files built with RAD. For example, SmartSketch supports opening the following file types:

- .igr — SmartSketch
- .sym — Symbol
- .sha — SmartSketch Drawing Editor
- .pid — SmartPlant P&ID
- .sma — SmartPlant Instrumentation
- .spe — SmartPlant Electrical
- .zyq — Zyqad

Another common use for SmartSketch is to read/write data from/to AutoCAD or MicroStation files. The underlying software supporting this functionality is referred to commonly as the *Translators* module, or just *Translators*, or *Translation*. Translation is a key functionality and concern for many Smart 3D customers who have requirements to deliver documents in AutoCAD or MicroStation format or work with existing files in those formats.

What is SmartSketch Drawing Editor?

SmartSketch Drawing Editor is a 2D CAD application provided by Intergraph when you purchase a Smart 3D system. SmartSketch Drawing Editor is used to view/edit Drawings files from within the Smart 3D environment (Ex., *Drawings and Reports* task).

NOTE: Formerly, SmartSketch Drawing Editor was called *Shape2DServer*. The file extension for SmartSketch Drawing Editor (.sha) is based on the former name. You continue to see Shape2DServer referenced in many places (e.g., directory structure on disk, within the application UI itself, the registry, etc.).

Though SmartSketch and SmartSketch Drawing Editor have different file extensions, .igr and .sha respectively, they both create/maintain Symbol files with the same .sym extension.

What is the difference between SmartSketch and SmartSketch Drawing Editor?

Aside from the color of the swirly-head icons...? There is very little to differentiate the two software applications. SmartSketch Drawing Editor does include certain toolbars and commands relevant only within the Smart 3D environment. If you are working within the Smart 3D Drawings and Reports task, you are using SmartSketch Drawing Editor to view/revise drawings.

However, once you save a Drawings file to disk, you have the choice of using SmartSketch or the SmartSketch Drawing Editor application to view/revise the file (if you have installed SmartSketch).

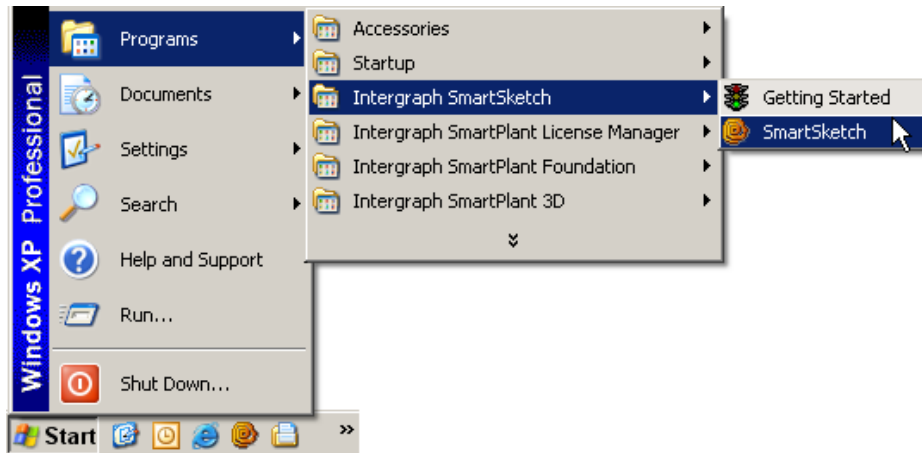
SmartSketch Drawing Editor, just like SmartSketch, supports many different file types; it also includes the *Translators* software (support for reading/writing files from/to AutoCAD or MicroStation).

NOTE: For the remainder of this training guide, SmartSketch is used to refer to either software application. All features demonstrated are present and supported in both applications. If there is a difference, it is noted.

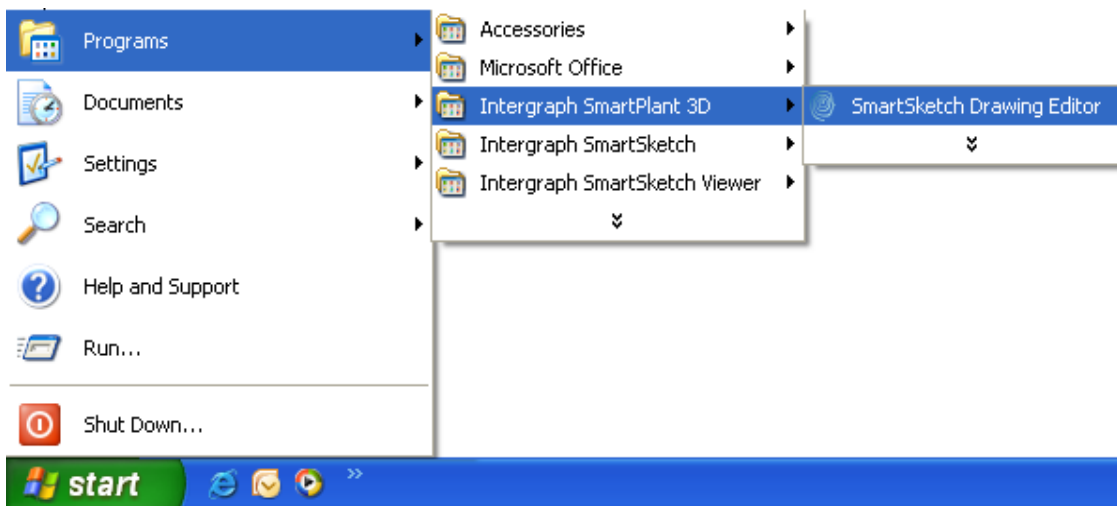
Invoking SmartSketch

SmartSketch documents are collections of drawing sheets and other objects. They have a standard file extension of .igr, comparable to .doc/.docx used in Microsoft® Word™.

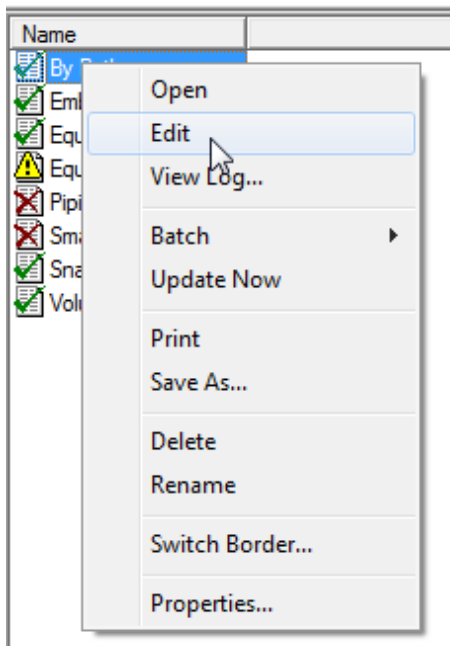
SmartSketch invoked from the Start menu:



SmartSketch Drawing Editor invoked standalone from outside the Smart 3D environment:

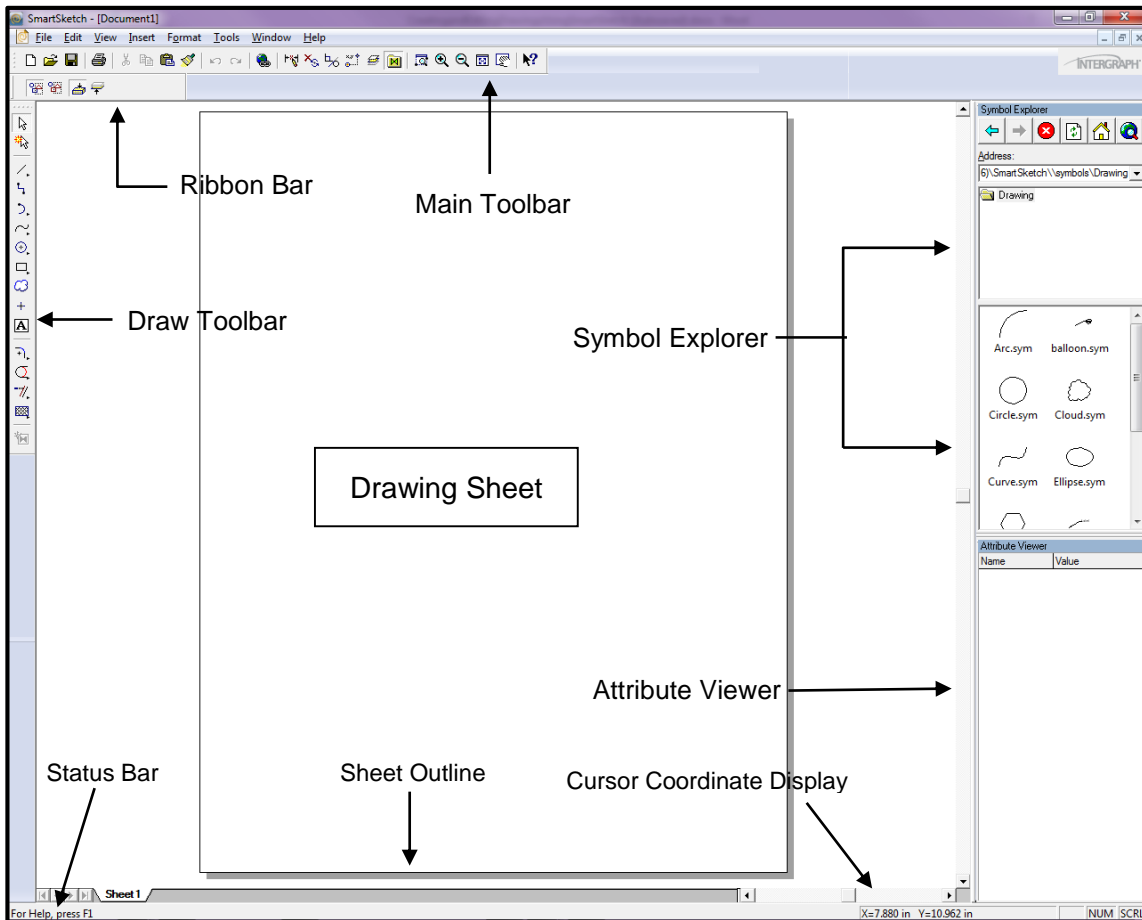


NOTE: Invoking SmartSketch Drawing Editor standalone from outside the Smart 3D environment requires a SmartSketch license.

SmartSketch Drawing Editor invoked from a Smart 3D task:

SmartSketch Working Environment

Before beginning an editing session in SmartSketch, the user should be familiar with its interface tools and operating environment



The SmartSketch working environment consists of the following key components:

- **Main Toolbar** - contains buttons that provide shortcuts to common Windows functions and editing commands, as well as frequently used SmartSketch commands and toolbars.
- **Ribbon Bar** - dynamic toolbar that displays different options depending on which SmartSketch command has been selected.
- **Drawing Sheet** - the area in which graphic objects are placed. There can be one or more drawing sheets in each document.
- **Sheet Outline** - shows the orientation of the drawing sheet and the default printable region.
- **Status Bar** - displays information and messages giving feedback to the user for the selected command.
- **Cursor Coordinate Display** - shows the coordinates of the cursor position in the active drawing window.

- **Draw Toolbar** - contains buttons that activate commands used to draw and modify objects. It may change appearance depending on the template used to create the drawing.
- **Symbol Explorer** - provides a user interface to drag symbols from a directory into the current document.
- **Attribute Viewer** - displays user-defined properties and parameters of an object, symbol, or inserted document.

Toolbars and Ribbon Bars

Toolbars

Toolbars are used in SmartSketch to activate most commands. By default, SmartSketch's **Main** toolbar is located in a "docked" position just below the menu bar. A toolbar may be docked at the top, bottom, right, left or floated anywhere in the workspace. Floating toolbars can also be resized for a different look.



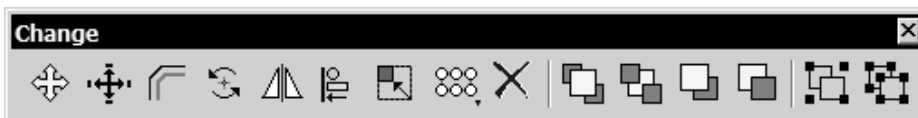
The **Main** toolbar provides direct access to three additional toolbars, two important ribbon bars, and the **Symbol Explorer**. Clicking the appropriate icon on the **Main** toolbar activates these features; the desired toolbar, ribbon bar or the **Symbol Explorer** should then display in or disappear from the application window as it is toggled on or off.



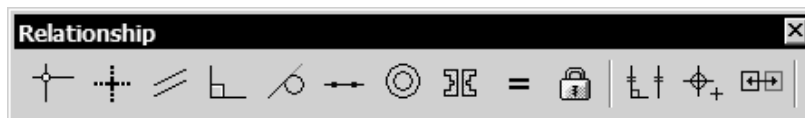
The **Dimension** button on the **Main** toolbar opens or closes the **Dimension** toolbar. The **Dimension** toolbar has commands that can be used to place annotations and dimensions on the drawing, as well as basic measurement tools.



The **Change** button on the **Main** toolbar accesses the **Change** toolbar. This toolbar has commands that can be used to manipulate and modify objects on the drawing.



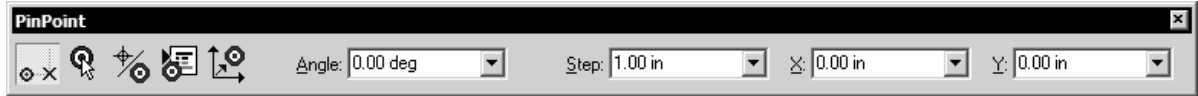
The **Relationships** button on the **Main** toolbar accesses the **Relationship** toolbar. This toolbar has commands that may be used to add or modify relationships on objects in the drawing, and control whether the relationship indicators display as you work.



Ribbon Bars



The **PinPoint** button on the **Main** toolbar toggles the **PinPoint** ribbon bar. This ribbon bar allows placement and modification of objects at a relative distance to some pre-defined point.



The **Layers** button on the **Main** toolbar accesses the **Layer** ribbon bar. This ribbon bar displays a list of the layers on the current drawing sheet, allows new layers to be created and the selection of layers to be displayed, and can be used to change objects to a different layer on the sheet.



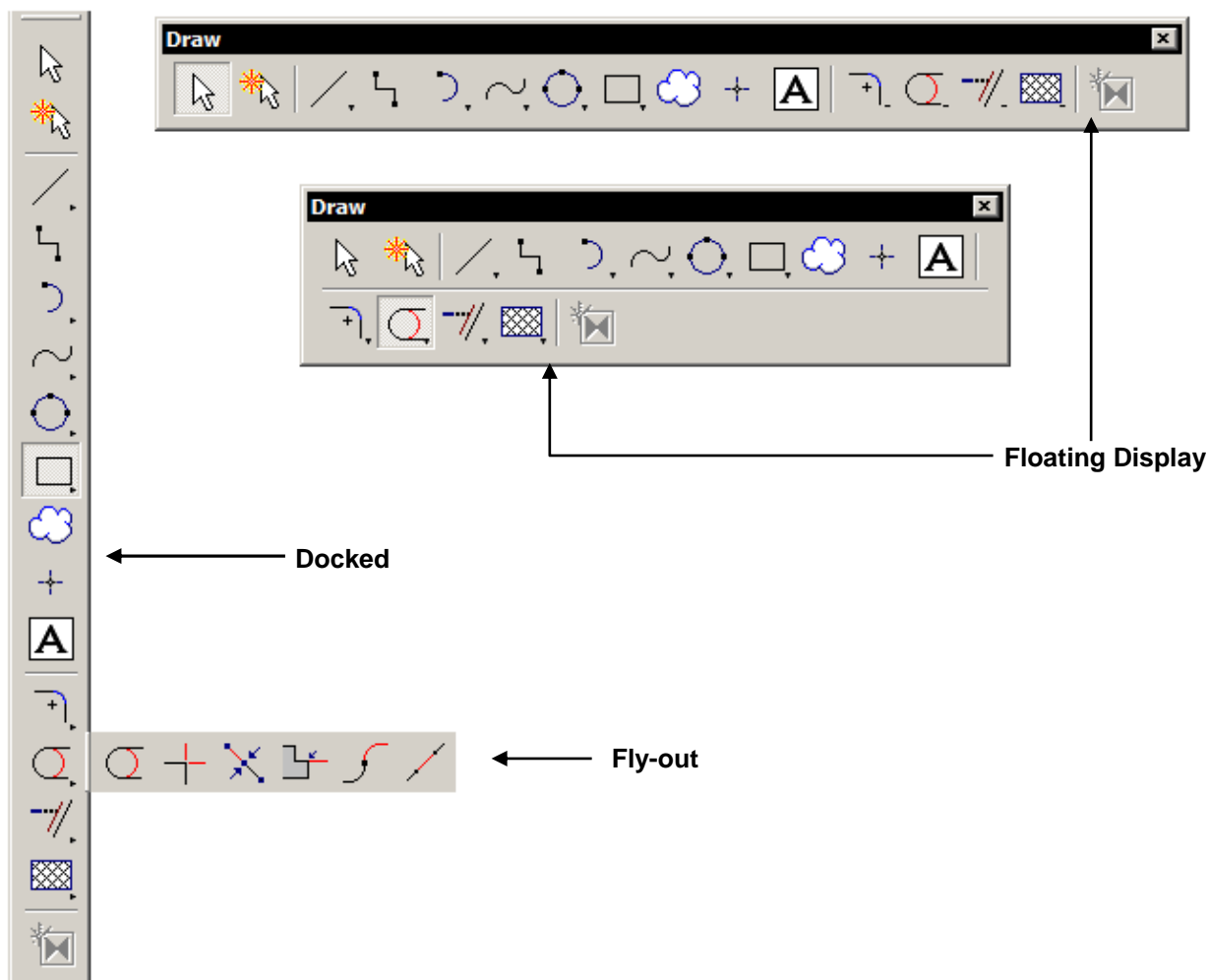
Dynamic Ribbon Bar

The dynamic **Ribbon** bar displays different options depending on which SmartSketch command has been selected. The commands associated with this toolbar will be discussed in more detail throughout the course.



Draw Toolbar

Common drawing commands are located on the **Draw** toolbar. Some command buttons may display fly-outs when you click and hold down the command button. These buttons have a small black arrow in the lower right corner. Fly-outs present a grouping of commands functionally related to the original command displayed.



What determines which toolbars are displayed?

There are several Toolbars entries in the registry for HKEY_CURRENT_USER that determine which toolbars are displayed/not displayed in SmartSketch. The registry settings that are used are determined by how SmartSketch is invoked. SmartSketch Drawing Editor invoked standalone from outside the Smart 3D environment looks in a different registry location than SmartSketch Drawing Editor invoked from a Smart 3D task.

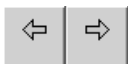
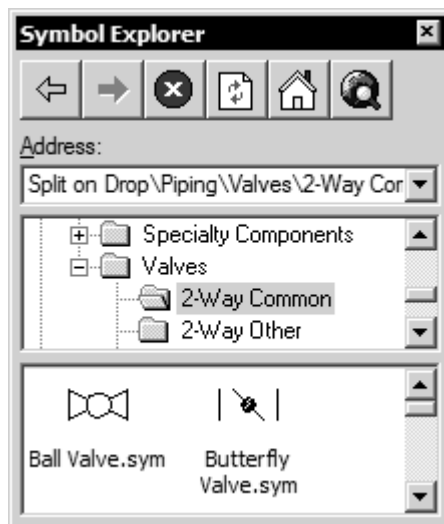
For a specific standard toolbar, such as the Dimension or Change toolbar, to be displayed in both standalone and connected modes, it has to be turned on in both environments. Also, there are some toolbars, such as the Smart 3D Drawings Compose toolbar, that are only functional inside of Smart 3D. Therefore, these toolbars are only displayed when SmartSketch Drawing Editor is invoked from inside the Smart 3D environment. If there are commands on a specific toolbar, such as the Support Utilities Pack toolbar, that could cause problems inside of the Smart 3D environment, then these commands are not displayed on the toolbar when SmartSketch Drawing Editor is invoked from inside the Smart 3D environment.

NOTE: The Draw toolbar and the Schematic toolbar are document centric toolbars. This means the display setting for these toolbars is saved in the document. If you create a new file, these toolbars will display/not display based on their display setting in the template file that was used to create the new file. For existing files, these toolbars will display/not display based on whether they were displayed when the file was last saved. Other toolbar settings are written to the registry under the Toolbars key; therefore, once a toolbar is turned on/off, it remains on/off between SmartSketch sessions.

Symbol Explorer



The command to toggle on or off the **Symbol Explorer** is located on the **Main** toolbar. The **Symbol Explorer** provides a simple user interface to drag symbols from a directory into the current document. You can also use the **Symbol Explorer** to view document directories on your computer or a network drive, as well as display HTML pages on the World Wide Web or your computer.



Back and **Forward** buttons either take you backward to the previous location accessed by the **Symbol Explorer** or forward to the current location or a previous location (depending on how far **Back** was selected). The display will either be a web page or a directory path.



Stop halts the loading of an HTML page.



Refresh reloads the current HTML page or active directory.

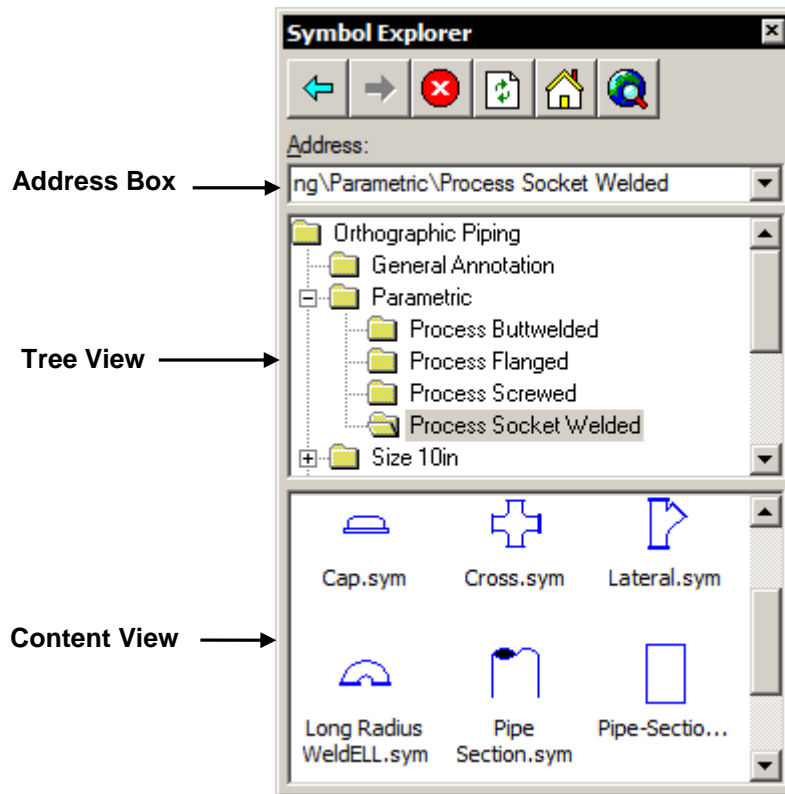


Home takes the **Symbol Explorer** back to the “home” page that is set in the **File > Properties** dialog box on the **Browser** tab. The setting on the **Browser** tab also determines which symbol set is displayed in the **Symbol Explorer** when a file is opened.



Explore Elsewhere accesses the **Browse for Folder** dialog box so that you can navigate to another directory for the **Symbol Explorer** to display.

The **Symbol Explorer** can be docked on the left or right side of the SmartSketch application window. Double-click the bar at the top of the **Symbol Explorer** to display it as a separate window from the main window. You can also resize the **Symbol Explorer** window.



Address Box

The address can be a directory on a local or network drive or an HTTP address to a web page.

Tree View

The **Tree View** pane uses the same navigation as Windows® Explorer™ to reveal or conceal directories from a user-defined root level directory.

Content View

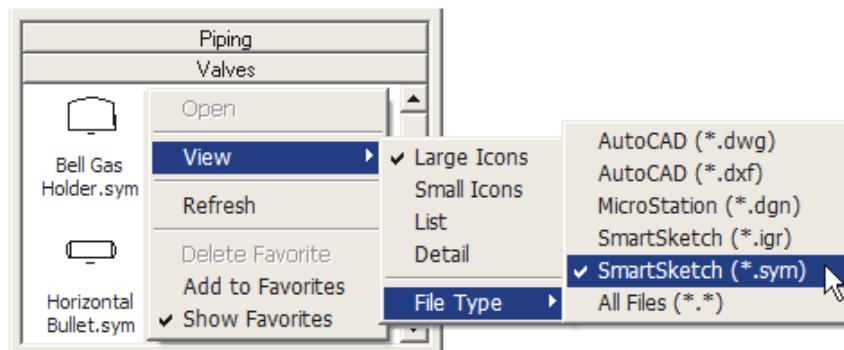
The **Content View** pane displays a list view of content, and can display one or more **Favorite Buttons**. This pane supports the filtering of file types in the list view from the shortcut menu. Symbols can be dragged and dropped from this window.

Symbol Explorer Shortcut Menu

To access this command, pause the pointer in the **Symbol Explorer** window and right-click. You may also refresh the **Content** display of the **Symbol Explorer** from this menu.

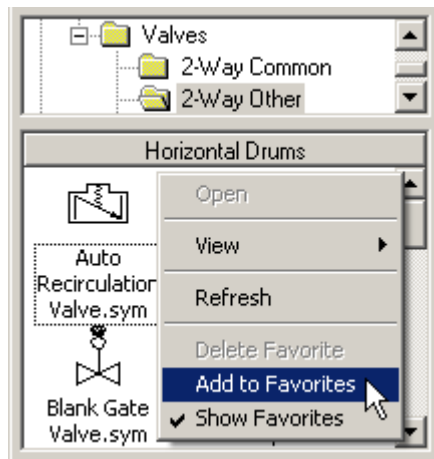
View

Filters the view in the **Symbol Explorer** window according to the document type that you select. For example, if you want to see only SmartSketch documents, you could select the option for **.igr** file types. When you set this option, the **Symbol Explorer** window displays only **.igr** documents. This command also can be used to set up how files are displayed in the content view: **Large Icons**, **Small Icons**, **List** or **Detail**.



Favorite Buttons

Adds a shortcut bar to the **Symbol Explorer** window when you select a document or web page in the window. You can click this shortcut to go to a favorite document or directory at any time

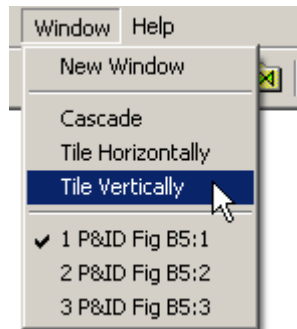


In the **Symbol Explorer** window, navigate to the directory that you want to add to a list of favorites. In the **Content View** window of **Symbol Explorer** window, right-click to activate the shortcut menu and select the **Add To Favorites** command. This will add a bar to the top of the **Content View** window.

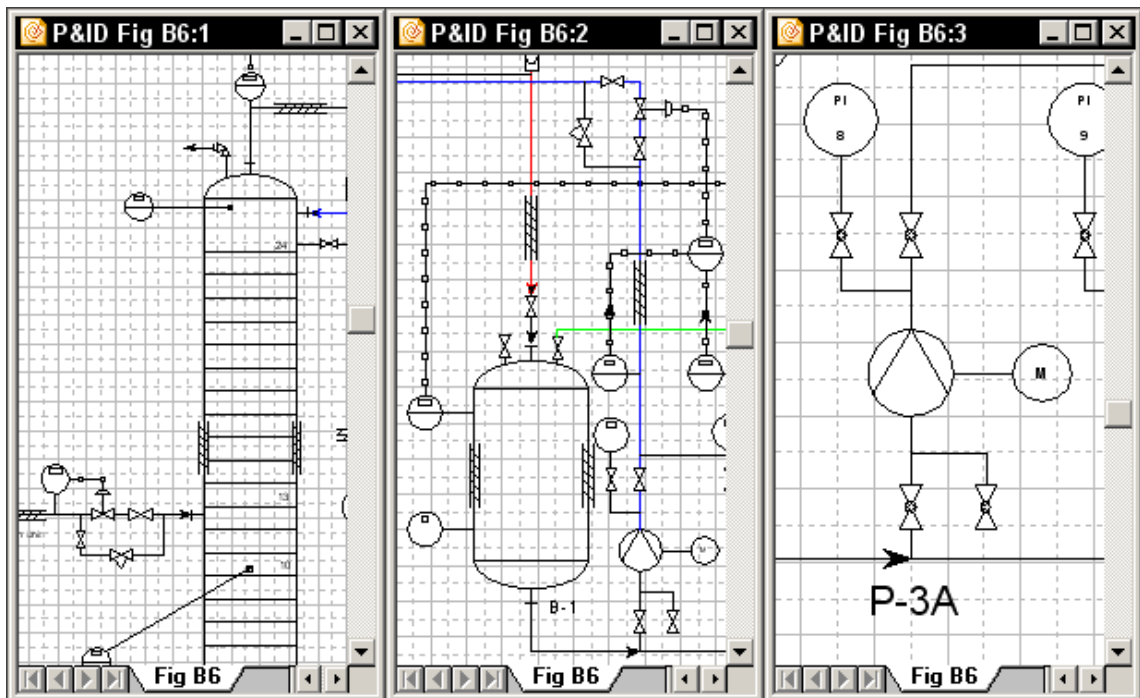
To remove a "favorite" select the bar you want to delete and click the **Delete Favorite** command from the **Symbol Explorer** shortcut menu.

SmartSketch Document Windows

The commands on the **Window** pull-down menu allow new windows to be created or arranged so that more than one section of a document or more than one document can be displayed. Open windows can be minimized and displayed as icons in the application workspace. Multiple windows can also be cascaded or tiled from the **Window** menu.

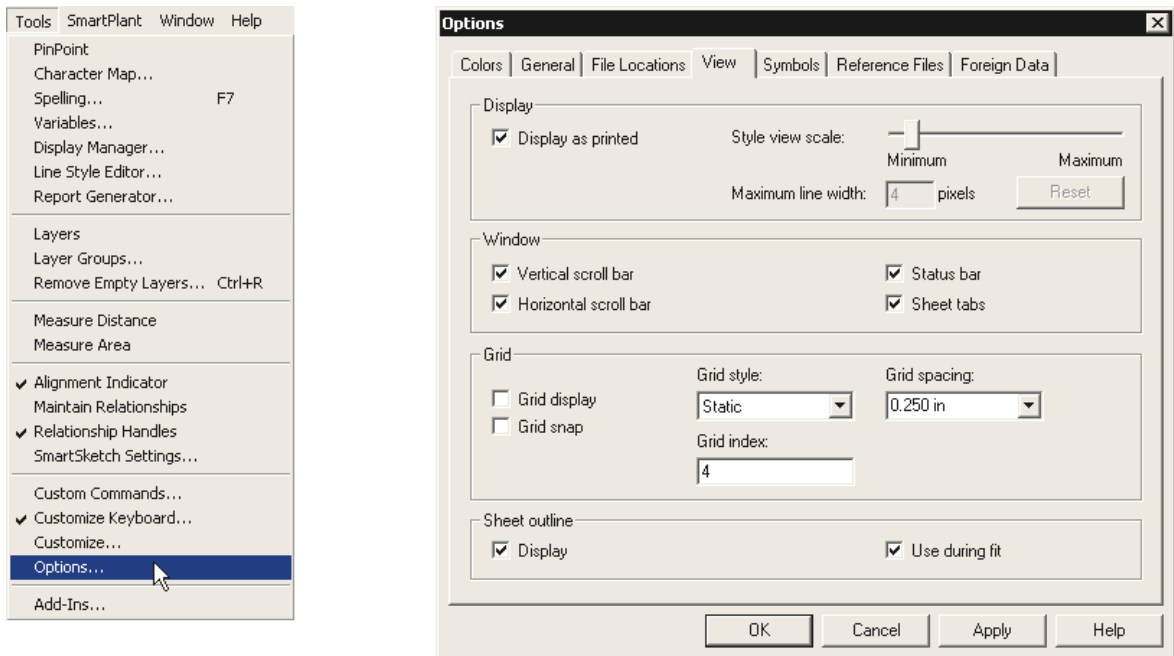


Below is an example of three windows that have been tiled with the **Tile Vertically** selection from the **Window** menu.



View Options

Turning certain window options on or off can change the appearance of the window. To change view options, select **Tools → Options**. In the **Options** dialog box, select the **View** tab.



Display as Printed

Displays the document as it appears on paper.

Style view scale

Controls the width of linear styles as displayed on the screen. This option is only available when **Display as printed** is disabled and will not apply to printed drawings.

Maximum line width

Sets the maximum width allowed for the display linear styles. This option is only available when **Display as printed** is disabled and will not apply to printed drawings.

Vertical Scroll Bar

Displays the vertical scroll bar in the view windows.

Horizontal Scroll Bar

Displays the horizontal scroll bar in the view windows.

Status Bar

Displays the status bar of the application.

Sheet Tabs

Displays the drawing sheet tabs.

Grid

This will be explained in detail in Lesson 3, *Drawing Environment*.

Display

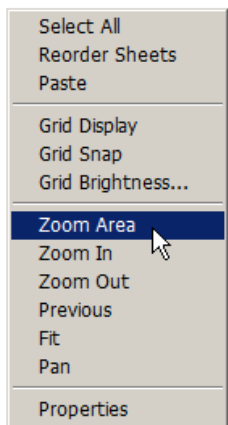
Displays the outline of a drawing sheet so you can view its boundaries.

Use during fit

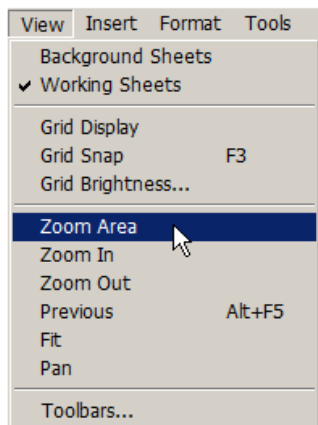
Specifies whether or not to utilize the sheet outline when using the “fit” view command.

Changing View Display

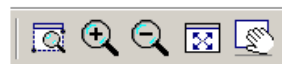
Viewing commands control what is displayed in the drawing window. These commands are available from the pull-down menu bar, the right-click menu and the **Main** toolbar. The view can be manipulated in the following ways:



**Right-Click
Shortcut Menu**



View Pull-down Menu



Main Toolbar

The Zoom commands can be used to display more detail or zoom out to see more of the drawing sheet.



Zoom Area enlarges the display of objects in the active window.



Zoom In enlarges the display of objects around a specified point in the active window.



Zoom Out reduces the display of objects around a specified point in the active window.

Previous View restores the previous view of the drawing sheet.



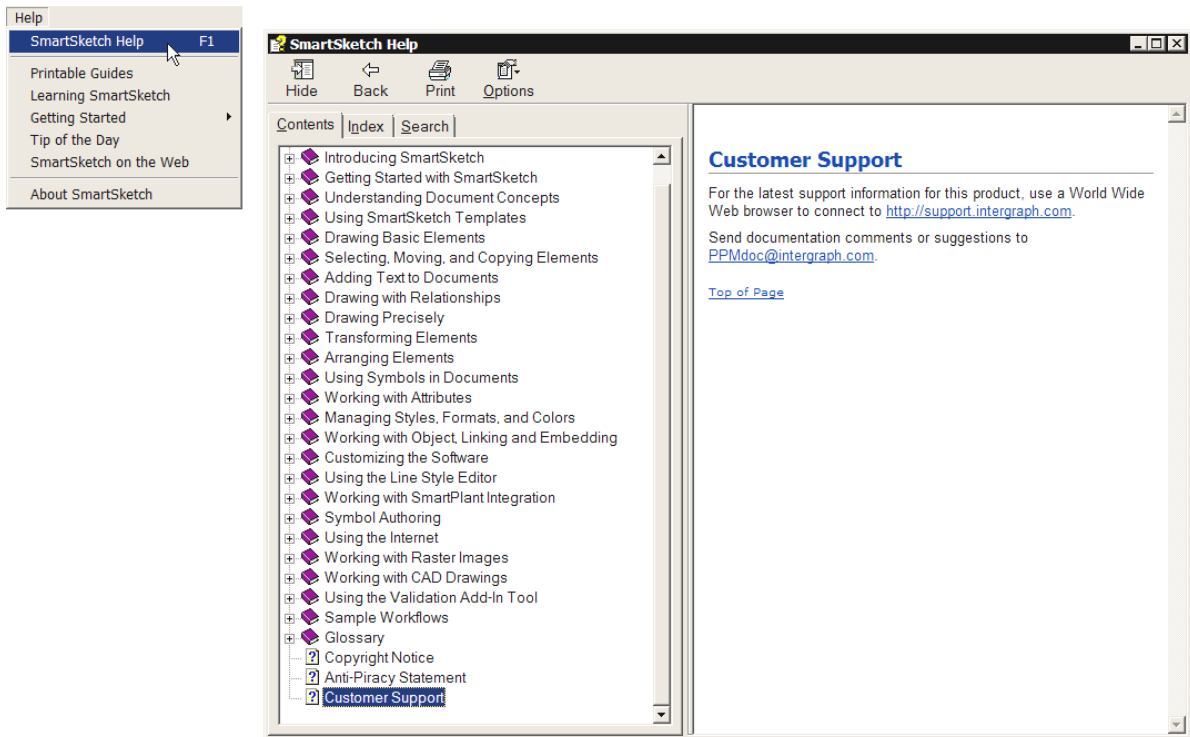
Fit resizes the view to display all drawing graphics in a window or, depending on the method used, fits the view to display just the objects that have been selected with the Select Tool.



Pan moves the drawing sheet to position another part of it in the center of a view.

Accessing SmartSketch Help

Select **Help** → **SmartSketch Help** to open the **Help** documentation. You can view help as a document through the table of contents as displayed in the figure below.



- **Contents tab** - double-click on the desired heading to expand the topic list and the information about the topic will display in the window to the right.
- **Index tab** - search for specific topics or phrases in SmartSketch Help.
- **Search tab** - searches for a specific topic, or lists related topics.



Found on the **Main** menu, activates context sensitive Help on an item in the current window. You can use this command to get help on any item even when the item is not active. After clicking the button, the pointer changes to a northwest arrow with a question mark. Click on the item for which you want to display help.

You can also press **Shift + F1** to get context sensitive **Help** on a command.

SECTION 2

Drawings Files

Topics

- Drawings Files vs. SmartSketch Files
- Drawings File Architecture

Drawings Files vs. SmartSketch Files

SmartSketch documents are used to store and manage information used in a design project. A document is a file that serves as a collection point for the design. It stores information about the project on one or more sheets. Each sheet may contain any number of drawing objects and, using OLE technology, embedded or linked files created in other applications as well (e.g. Microsoft® Excel™ spreadsheets for a Bill of Materials, assembly specifications and descriptions in Microsoft® Word™ .doc files, raster images of a part or location, etc.).

In addition to the visible objects placed in documents, there are several properties or attributes that are defined and stored with the document file.

What is a Drawings file?

A Drawings file is any file generated with the Drawings and Reports task in Smart 3D. A Drawings file has a .sha extension when saved to disk.

NOTE: This document uses the term *Drawings file* explicitly to refer to any file generated with the Drawings and Reports task in Smart 3D.

What is a SmartSketch file?

A SmartSketch file is any file that is created with a SmartSketch application. For example:

- Files generated with the Smart 3D Drawings and Reports task
- Files created/managed with SmartSketch (.igr file extension)
- Files created/managed with SmartSketch Drawing Editor (.sha file extension)

NOTE: The term *file*, *document*, and *drawing* are interchangeable; a SmartSketch *file*, SmartSketch *document*, or *drawing* all mean the same thing.

The .igr and .sha file extensions are compatible and supported equally using either SmartSketch or SmartSketch Drawing Editor. You can open a .sha file with SmartSketch or open an .igr file with SmartSketch Drawing Editor. However, the file always maintains its original extension (you cannot save a .sha file to .igr or vice versa).

A typical SmartSketch file

A typical SmartSketch file contains objects that you draw or place on a sheet of paper (sketching on a sheet of paper is the basic working metaphor of SmartSketch). When you draw or place graphics on the sheet of paper, the objects you create are selectable and editable. For example, after placing a line or circle, you may decide to change its color, line type, width, geometric properties, or even delete the object. This workflow is consistent with any other CAD application you may have used (e.g., AutoCAD or MicroStation).

The following picture (Figure 1) shows an isometric detail in SmartSketch; the data was drawn in SmartSketch. Each individual object (line, valve symbol, dimension, etc.) is selectable and may be modified by the user.

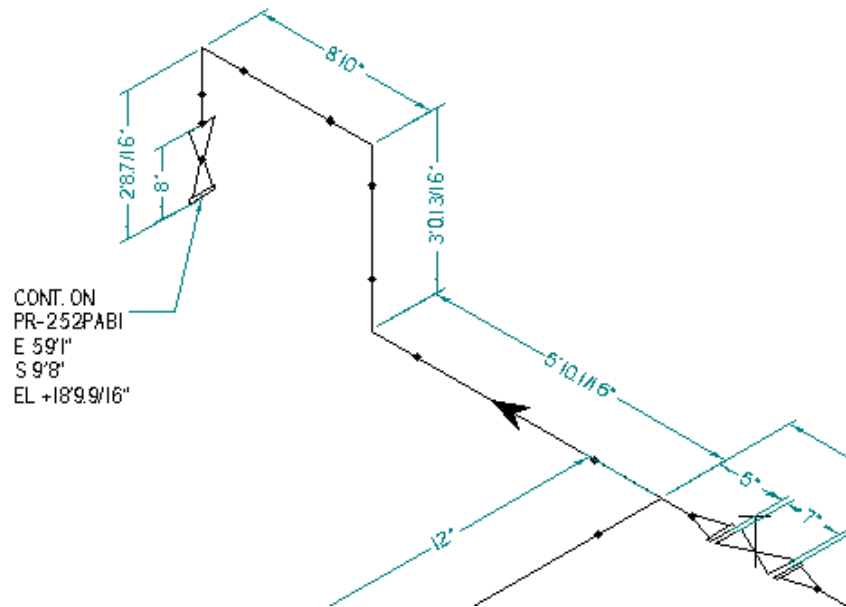


Figure 1—typical SmartSketch file, objects on sheet

Drawings File Architecture

The content/layout of a Drawings file is markedly different from most SmartSketch files. In a Drawings file, many of the graphical objects you see on the sheet of paper reside in another (SmartSketch) document that is embedded inside the document you have opened.

SmartSketch uses Microsoft's OLE technology to embed the document; just as you might embed an Excel chart or graph inside a Word document, the Smart 3D Drawings and Reports task embeds a SmartSketch document within another SmartSketch document. Objects in the embedded document are displayed through an object called a SmartFrame.

Because object(s) displayed via a SmartFrame actually reside in a separate document, they are not selectable and may not be modified by the user directly.

NOTE: Though you may use *PickQuick* or change the *Select Tool* locate mode to *Bottom Up* (from the default *Top Down*) to select an object within a SmartFrame, this does not imply that the object may be changed. If you are unfamiliar with any of these terms, they are discussed in detail in *Section 7 - Selecting and Finding Objects*.

The following picture (Figure 2) shows a portion of a file generated with the Drawings task within Smart 3D.

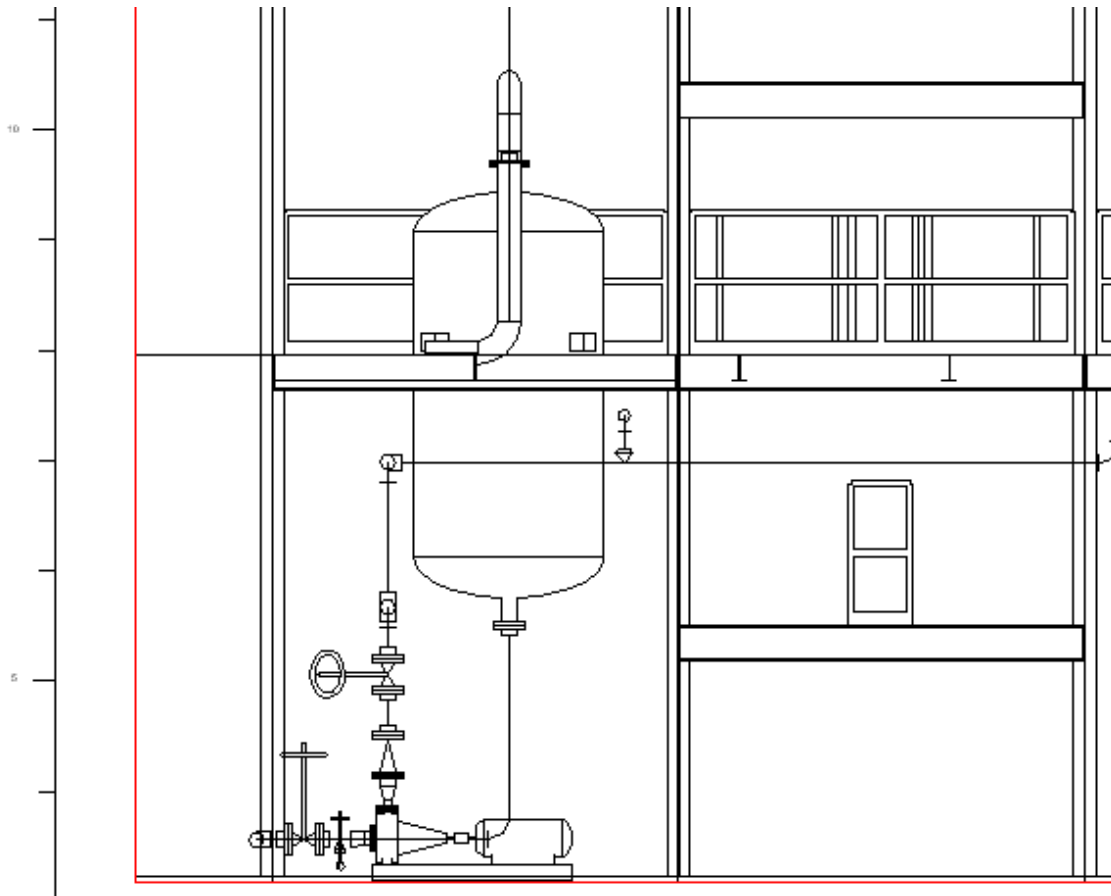


Figure 2—typical Drawings file, objects inside embedded document

NOTE: It is common to call the open Drawings file a *container document*, because it contains another embedded SmartSketch document. It is also called the *outer document*, with the embedded document called the *inner document*.

SmartFrames / Drawing Views

SmartSketch uses an object called a SmartFrame to display any object embedded/linked to a file using OLE technology. Because Drawings files always contain other embedded SmartSketch document(s), Drawings files always contain SmartFrames.

NOTE: Smart 3D has a specific term to refer to the SmartFrames used during Drawings generation: they are called *Drawing Views*.

The data displayed via a Drawing View (SmartFrame) is not editable from the container document. The only way to edit the objects is to open the embedded document, make the changes, and save.

NOTE: To make changes to an embedded document, select the Drawing View and use either the *Edit* menu or right-mouse popup menu to select *Edit/Open* on the object reference. You may also double-click an object in the Drawing View.

Annotations

Annotations, objects like labels, dimensions, leaders, etc. reside in the container (outer) document itself, i.e., the document you opened. You may modify/delete these objects, and the changes you make are remembered when the drawing is updated.

Drawing Scale

Drawing scale is handled very differently in a Drawings file compared to a typical SmartSketch file. In a typical SmartSketch file, if you need to represent a large area on the sheet of paper, you set a drawing scale on the sheet of paper itself.

For example, by scaling the sheet you can represent the footprint of a large building or structure on a standard size sheet of paper. This concept is identical to scale on a map, wherein the legend may read 1 inch = 100 miles. A sheet with a scale of 1:100 means that if you were to print the document, 1 (unit) represents 100 (units), e.g., 1 mm represents 100 mm. This concept allows you to draw at actual size on the sheet of paper itself, with no worries about scale.

However, in a Drawings file, the scale on the sheet of paper is normally 1:1. Scale is set on the Drawing View (SmartFrame) itself, which displays the contents of the embedded document. Because of this difference in the way scale is implemented, a tool was developed, called Scaled Sketching.

SECTION 3

Document Management

Prerequisite to the effective use of SmartSketch is an understanding of how the drawing environment is set up.

Topics

- Creating a new document
- SmartSketch document templates
- Opening an existing document
- Saving a SmartSketch document
- Customizing the keyboard
- Creating a custom toolbar
- The Options dialog box
- Printing a SmartSketch document

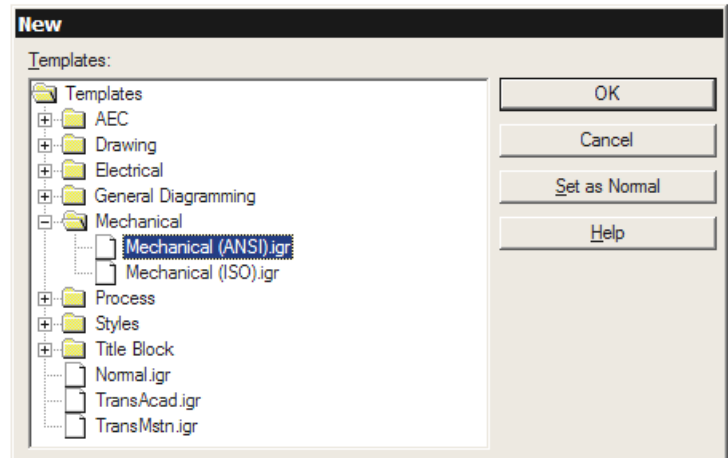
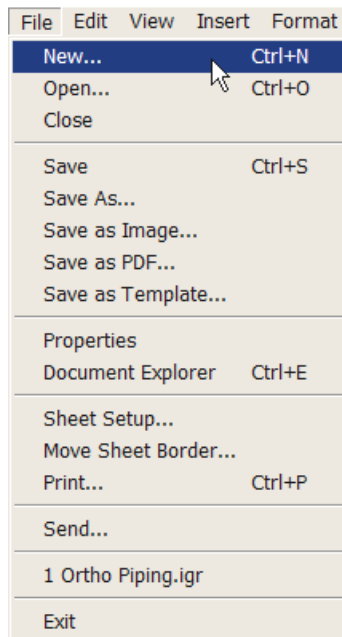
Documents

SmartSketch documents are used to store and manage information used in a design project. A document is a file that serves as a collection point for the design. It stores information about the project on one or more sheets. Each sheet may contain any number of drawing objects and, using OLE technology, embedded or linked files created in other applications as well (e.g. Microsoft® Excel™ spreadsheets for a Bill of Materials, assembly specifications and descriptions in Microsoft® Word™ .doc files, raster images of a part or location, etc.)

In addition to the visible objects placed in documents, there are several properties or attributes that are defined and stored with the document file.

Create a New Document

The **File → New** menu item allows the user to create new documents. SmartSketch provides a number of industry solution templates to base the document on.



To create a new document

1. Select the **File → New** menu item
2. In the **File New** dialog box, choose a template to base the new document on.
3. Click **OK** and SmartSketch will respond by displaying a new document file.

Templates

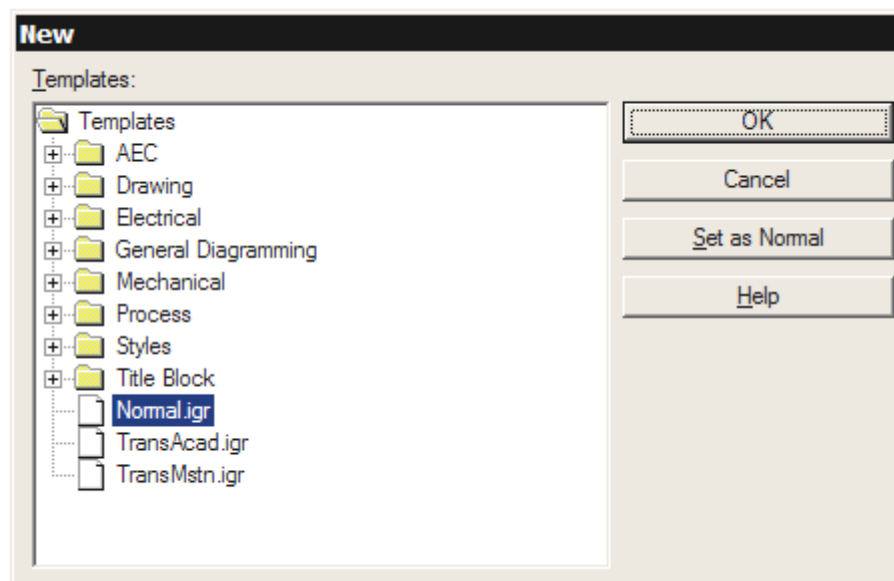
A template is used as a starting point in creating a document. A template is a file that provides settings for such things as text, page formats, geometry, dimensions, units of measurement, toolbars, and styles that are used to produce a new document that conforms to some specified format.

The template that you use depends on the type of information you want to put in the document. For example, if you want to create a drawing that contains a schematic of an electrical component, you can use an electrical schematic template.

When you select a template from one of the industry modules, tools are available that allow you to create either a schematic diagram or a precision drawing. The templates that are delivered with the industry modules will also automatically set the **Symbol Explorer** to display the appropriate symbol set for that template.

Schematic diagrams are primarily created using symbols, connectors, and text at a 1:1 scale. Schematic diagrams do not represent a logical flow. The **Schematic** toolbar is available when you select a template designed to create a schematic diagram.

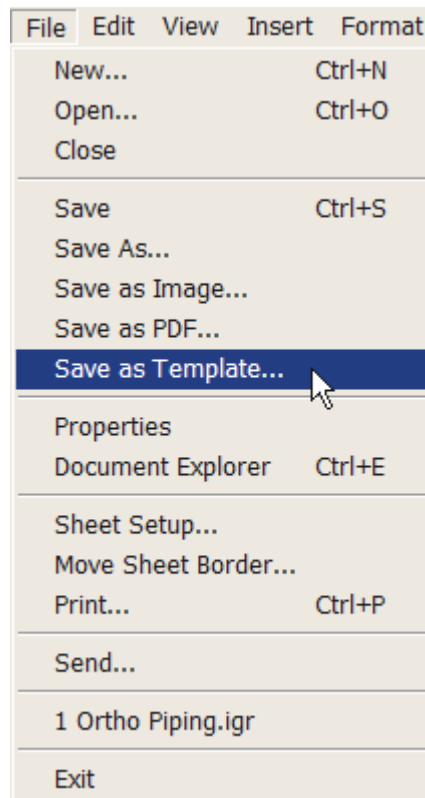
Precision drawings represent physical objects by lines, arcs, curves, rectangles, etc. that you draw precisely using the **Draw** toolbar. Precision drawings are created at real-world scale, and you can maintain relationships between objects in precision drawings.



Saving a Document as a Template

Sometimes you might want to make a copy of an existing document your starting point for creating other documents in the future. You can do this by creating your own template(s). A template defines a collection of settings that you will need in many different drawings. Some of those settings might include drawing sheet settings and scales, actual elements that you need to draw repetitively, and background sheets with customized title blocks.

To create a template, just save the current document with the **Save As Template...** command on the **File** menu. This command saves the document in the **Template** directory located in the directory where the software was installed. A different file extension in the name of the document is not necessary.

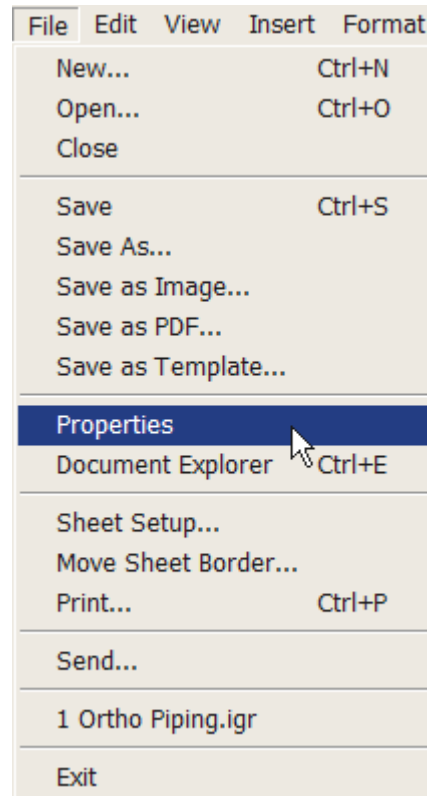


To base new documents on the template, Select **File → New** and select the template from the list. You can also open the template and save the document under another name.

Setting Document Properties

Properties on the **File** menu allows you to view, edit, and store properties for a document. Document properties can include the title, the author, and keywords that identify important information. These properties also can include document statistics, such as document size and the date that a document was created and last modified. Some properties, such as the date the document was last modified, are updated automatically.

To set the properties, you can click the **Properties** command on the **File** menu.



NOTE: You can also right-click on a blank area of the drawing to access Properties from the shortcut menu.

Summary Tab

The **Summary** tab displays the title, subject, author, keywords, and comments for the current document. The software generates the author and template information when the document is created, saved, or printed.

Users can enter or change any entry in the text fields unless the document is write protected.

The screenshot shows a Windows-style dialog box titled "C:\Unit 902\Piping\Ortho Piping.igr Properties". It has five tabs: "General", "Summary", "Statistics", "Units", and "Browser". The "Summary" tab is selected. The dialog contains the following fields:

- Title:** A text box containing "Ortho Piping".
- Subject:** A text box containing "Unit 902".
- Author:** A text box containing "Intergraph Process, Power & Marine".
- Keywords:** A text box containing "Piping".
- Comments:** A large, empty text area with a vertical scrollbar on the right.
- Template:** A text box containing "Ortho Piping (Imperial).igr".

At the bottom of the dialog are three buttons: "OK", "Cancel", and "Help".

Title, Subject and Author

Data fields that allow the user to specify the title of the current document; provide a general description of the current document; and display the name of the person who created the document.

Keywords

Displays a list of words that identify the document when you browse or search documents.

Comments

Shows optional descriptive comments about the current document.

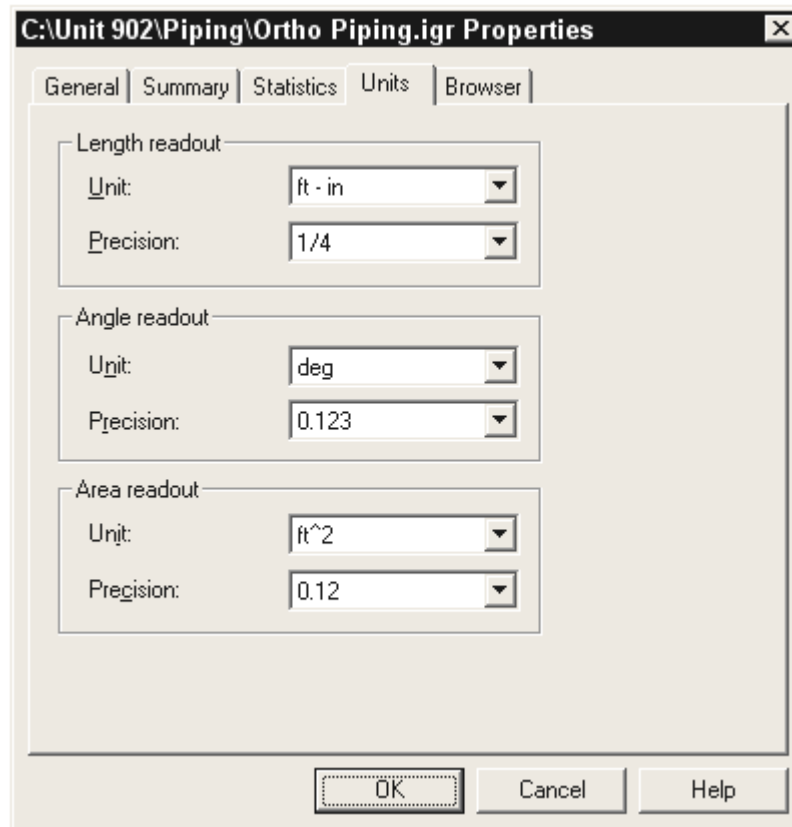
Template

Displays the name of the document template that was used to create the document.

Units Tab

The units of measure settings for a document are stored as a property. If the units of measure are changed, all subsequent measurements entered and displayed are affected.

Units of measure can be set in either Imperial or Metric units for values such as length, area, or angle. The units of measure can be changed at any time and the document still retains complete accuracy of the objects already placed on the drawing.



Length Readout

Sets the unit of measure and precision readout for the length values in a document.

Angle Readout

Sets the unit of measure and precision readout for the angle values in a document.

Area Readout

Sets the unit of measure and precision readout for the area values in a document.

Precision

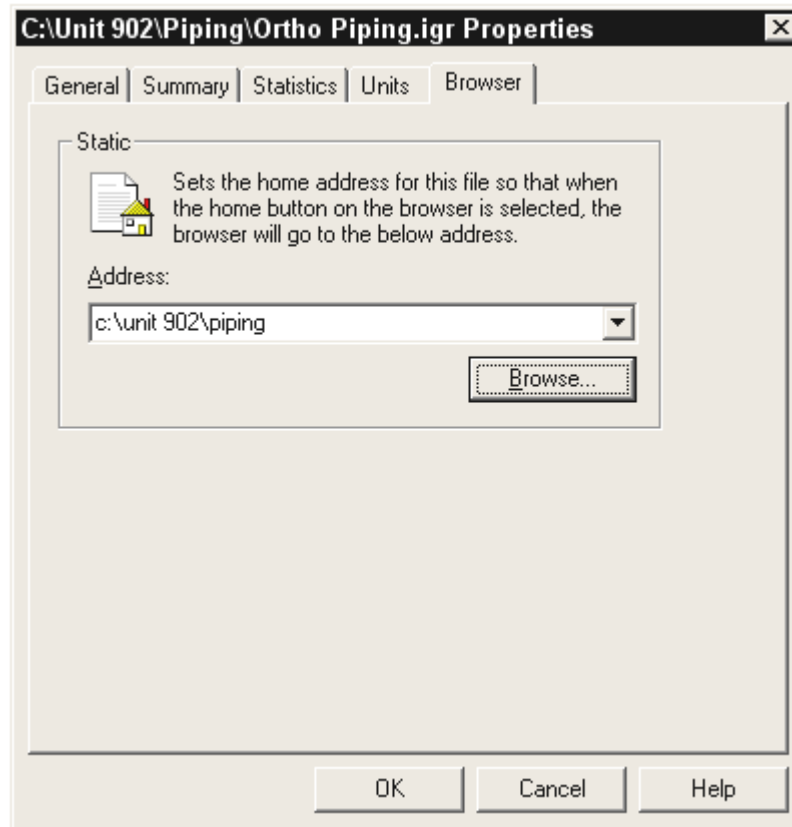
The precision readout sets the number of significant figures to display. It sets the accuracy of the unit readout value. The precision setting does not alter the numbers that you type into the fields, only the display of the numbers in the field. Values ending in 5 are rounded up. For


example, if the precision readout is .123 and you draw a line that is 2.1056 inches long, then the line value length is rounded. The length value appears as 2.106 inches long. If you are using mm as your drawing sheet units, you can have the values display in the fields as 3.5 mm or 3.50 mm.

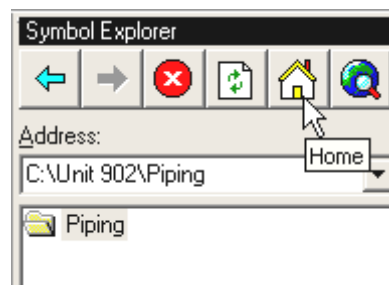
Important: When you set the units of measure for a document, the settings do not affect the dimensional values for the document. You can set units for the dimensional values with the **Dimension Properties** dialog box. You can access this dialog box by selecting a dimension and then clicking the **Properties** command on the shortcut menu. You can also set the dimension units by editing a dimension style using **Format → Dimension**.

Browser Tab

The **Address** box in the **Browser** tab is used to set the home address for the active document. This is normally used to define the location to access symbols associated with a certain type of drawing or design function.



Every time you open this document or click the **Home**  button on the **Symbol Explorer**, the Browser goes to the URL in the **Address** box.



Address setting

Sets the home address for the current document. When you open a document, then open the **Symbol Explorer** and click the **Home** button, the **Symbol Explorer** goes to the address you type here.

The address can be any address that you would use in a web browser, such as Microsoft Windows® Explorer™ or Netscape®.

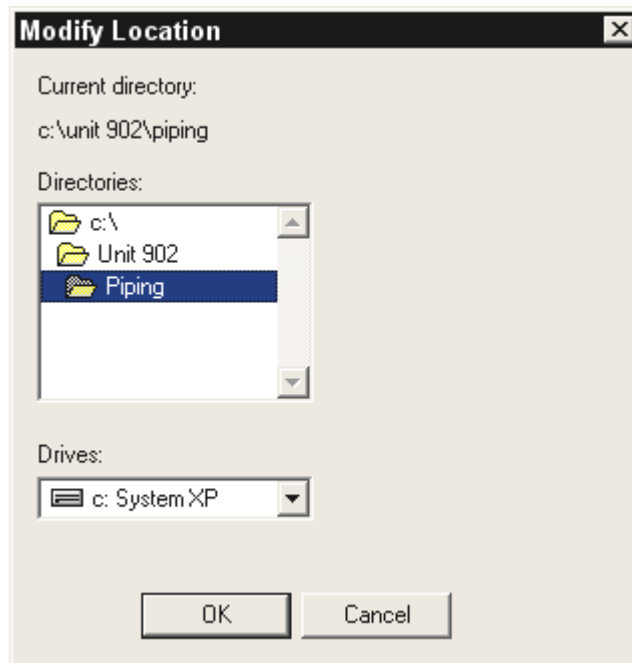
For example, you can type **http://www.intergraph.com**

If you want to set the **Symbol Explorer** to always look at symbols or other files on the local machine, you can enter the full path of the directory containing the files.


For example, you can enter **c:\Drawing Files**.

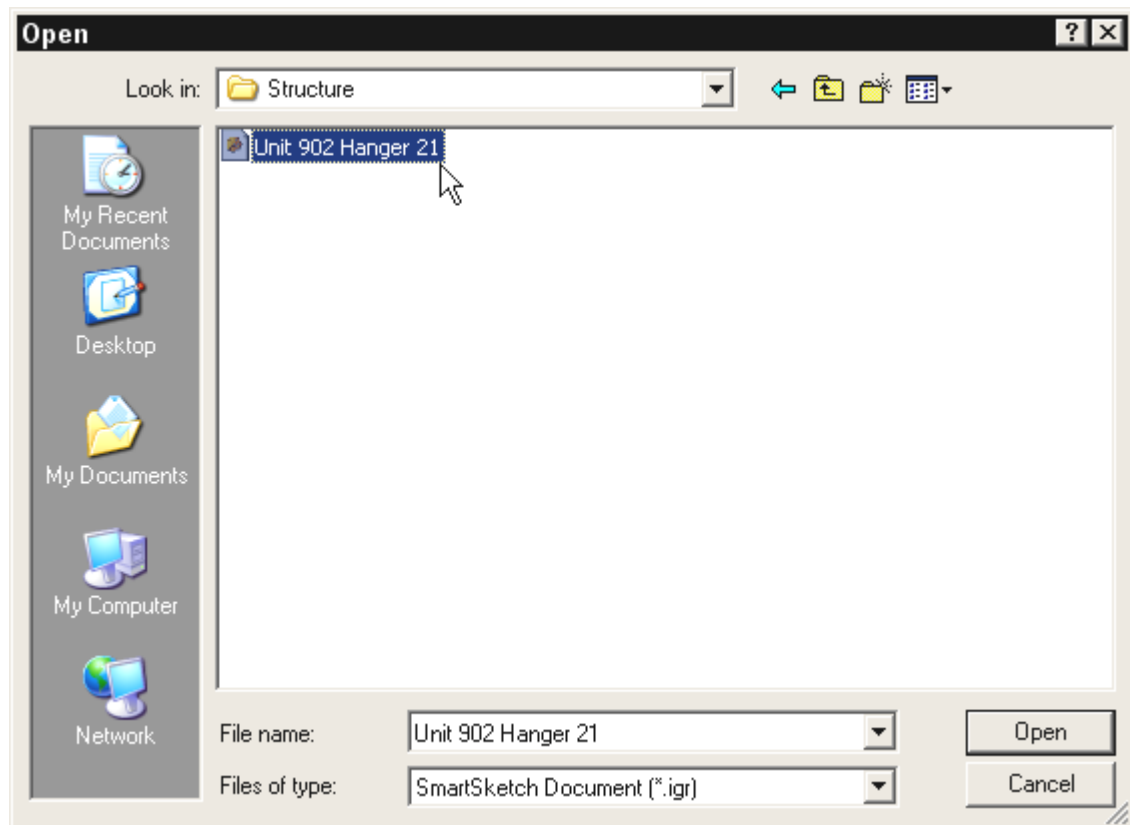
Browse

Accesses the **Modify Location** dialog box so that you can set another directory for the **Symbol Explorer** to look in. You can set the dialog box to look in directories on your computer or a network drive.



Opening an Existing Document

1. From the **File** menu, click **Open**; or select the  icon from the **Main** menu.

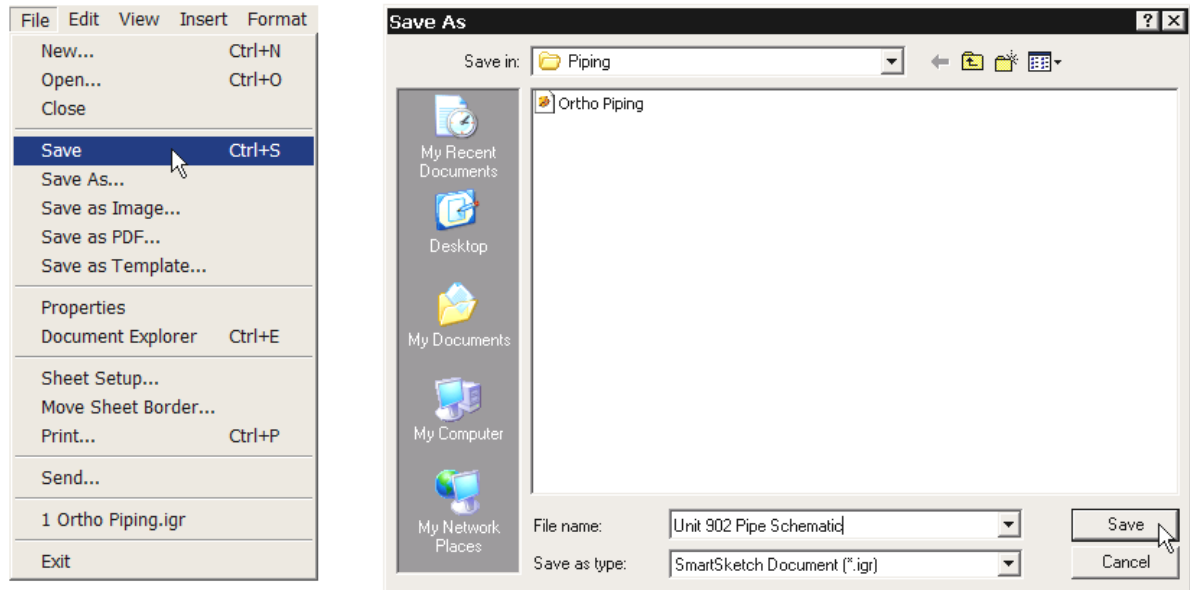



2. Select the directory containing the document that you want to open.
3. Do one of the following:
 - Key in the name of the document or select it from the list of documents and click the **Open** button.
 - OR -
 - Double-click the document filename from the list of documents.

Saving a SmartSketch Document

Save

This option will save the active document with its currently defined name, directory, and format. If saving the document for the first time, the **Save As...** dialog box is displayed so that the document can be given a name and specify a directory and format to save it to. When saving the document, it will remain open on the screen so work can continue.



The **Save**  icon can also be selected from the **Main** toolbar to save the active document.

Save As...

When choosing this option, the active document can be saved to a new document name, directory, and format.

Save As Image...

You can save the graphic data in the document as a raster image.

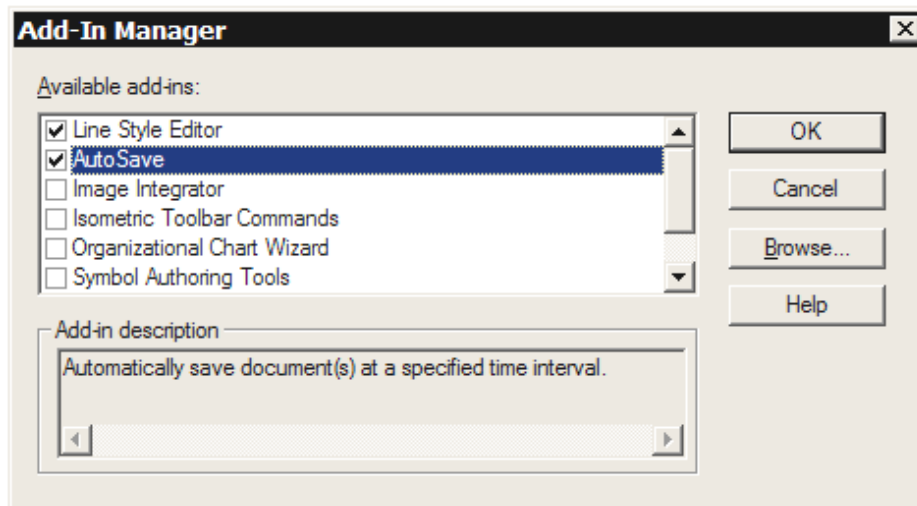
Save As PDF...

Saves the current file as a PDF document. You can control such options as the resolution of the PDF file as well as the compression for embedded images. If you are saving a multi-sheet document, you can specify which sheets to save and whether or not to create bookmarks for each sheet. This command is available only on the **File** menu.

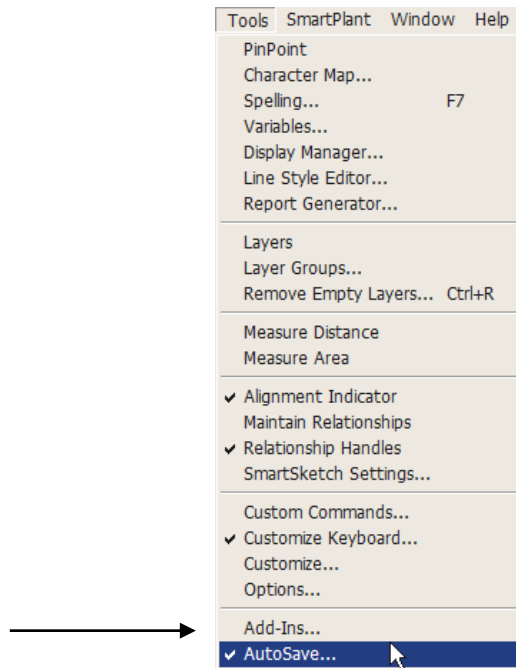
AutoSave

You can set option to automatically save open documents at the interval that you specify. This safety feature prevents you from losing work in case of an unexpected system shutdown. You can set several options for **AutoSave**, including how frequently to save, which documents to save, and whether to be prompted for each save. You can use **AutoSave** when you have one or more documents open.

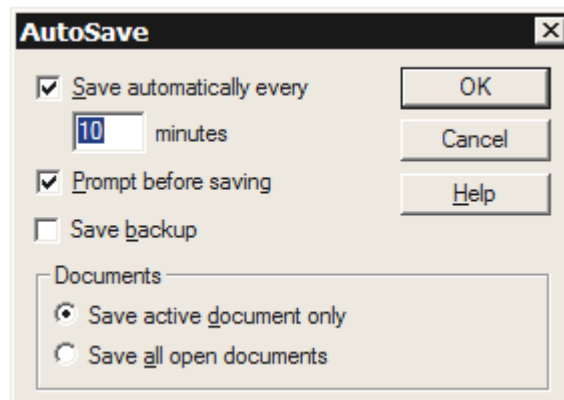
If the **AutoSave** option does not appear on the **Tools** menu, you must activate it by clicking the **Add-Ins** command on the **Tools** menu. Once the **Add-In Manager** opens, select the **AutoSave** option from the list of available add-in commands.



To hide the **AutoSave** option, either turn off the **AutoSave** option in the **Add-In Manager**, or uncheck the **Tools → AutoSave...** menu to disable it temporarily.



AutoSave Dialog Box



Automatic Save Every __ Minutes

Automatically saves open documents at the interval you specify if this option is checked. This check box also will turn off the **AutoSave** option.

Prompt Before Saving

Displays a dialog box to permit user confirmation before AutoSave begins saving documents. You can click one of the following buttons to answer the dialog box question: "Save changes in <document name>?"

Save Backup

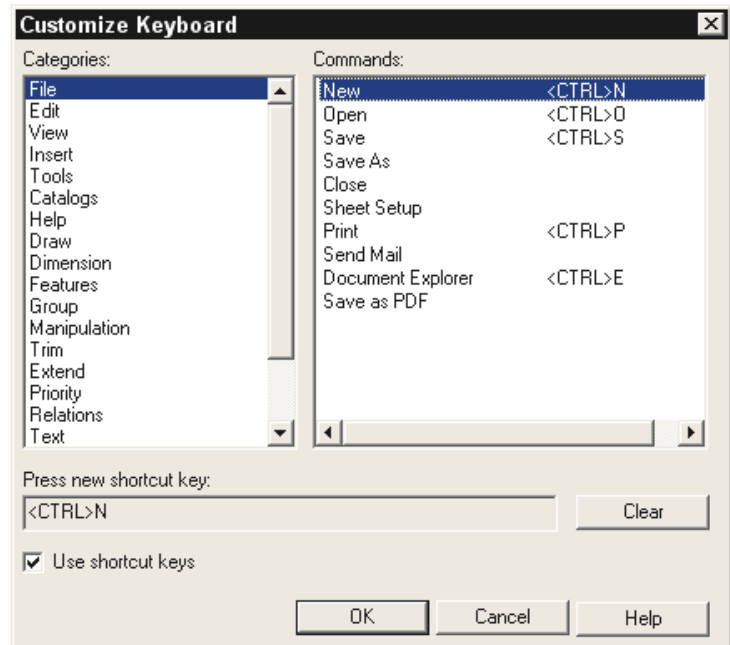
AutoSave makes a backup copy before overwriting the previous document with the current one.

Documents

Saves either the active document (**Save Active Document Only**) or all documents that are open (**Save All Open Documents**).

Customize Keyboard Command

Uses shortcut keys to quickly accomplish tasks you perform frequently. Shortcut keys are one or more keys you press on the keyboard to complete a task. For example, pressing **Ctrl+N** creates a new document or template; just as selecting **New** on the **File** menu or clicking **New** on the **Main** toolbar creates a new document or template.



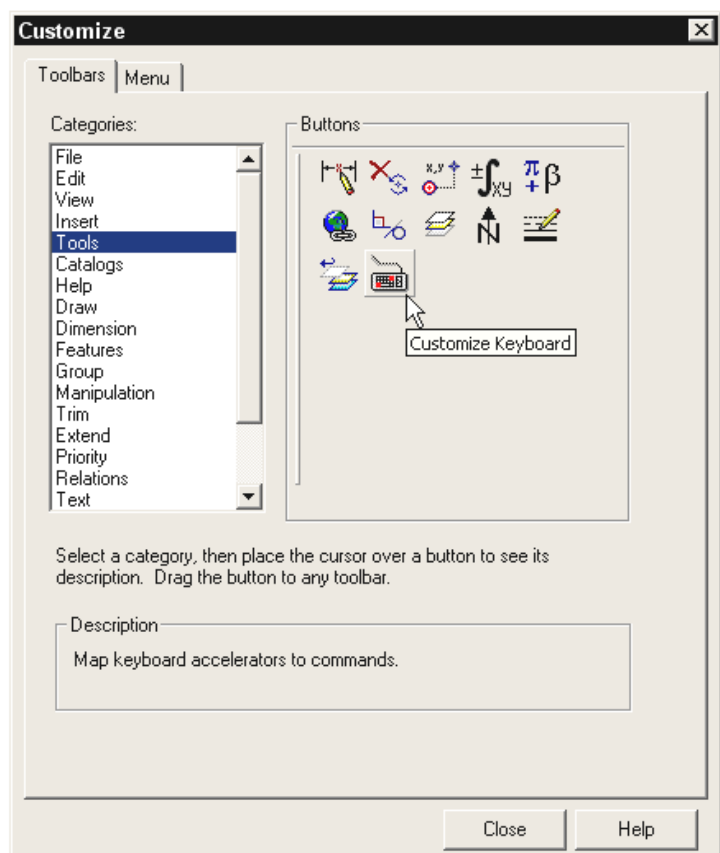
Creating a Customized Toolbar

You can add built-in commands or macros to the toolbars and menus in SmartSketch. For example, the **Measure Distance** command is on the **Dimension** toolbar. If you use the **Measure Distance** command more frequently than any other button on the toolbar, you could add the **Measure Distance** command to the **Main** toolbar. You could also create a toolbar that contains your most commonly used tools.

Toolbar commands can be added to a menu or menu commands to a toolbar. For example, you can place the **Sheet Setup** command on the **Main** toolbar as a button. In some cases there are commands within SmartSketch that have an icon associated with them, and you can add that command to an existing toolbar or create a new toolbar to contain commands that you commonly use.

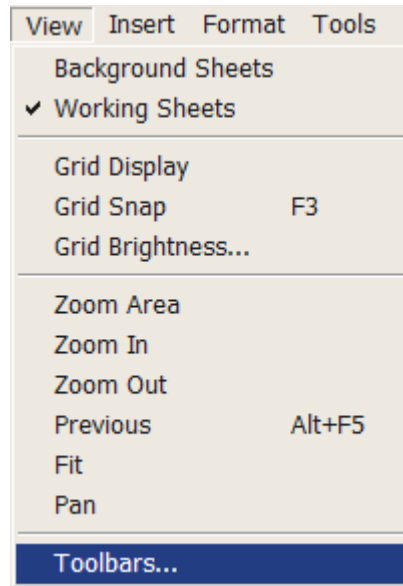
Toolbars Tab

Add or remove commands from toolbars.

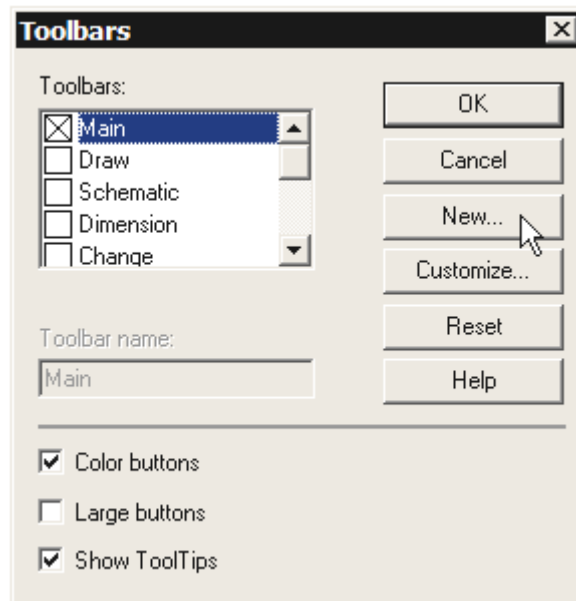


To create a new toolbar

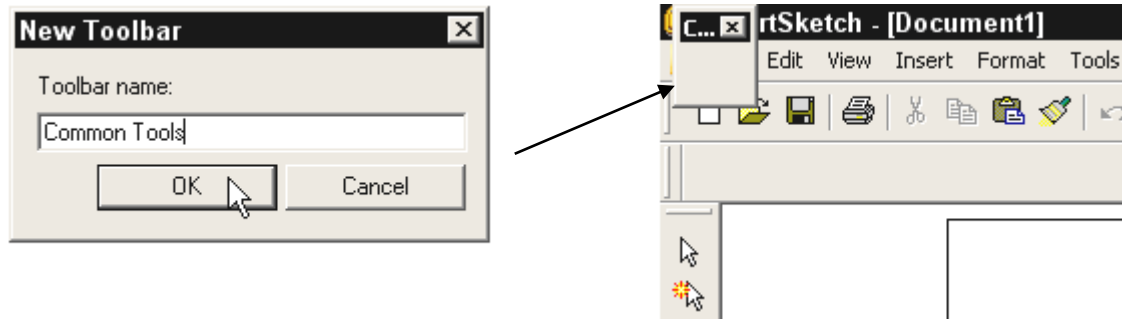
1. Select **View** → **Toolbars**.



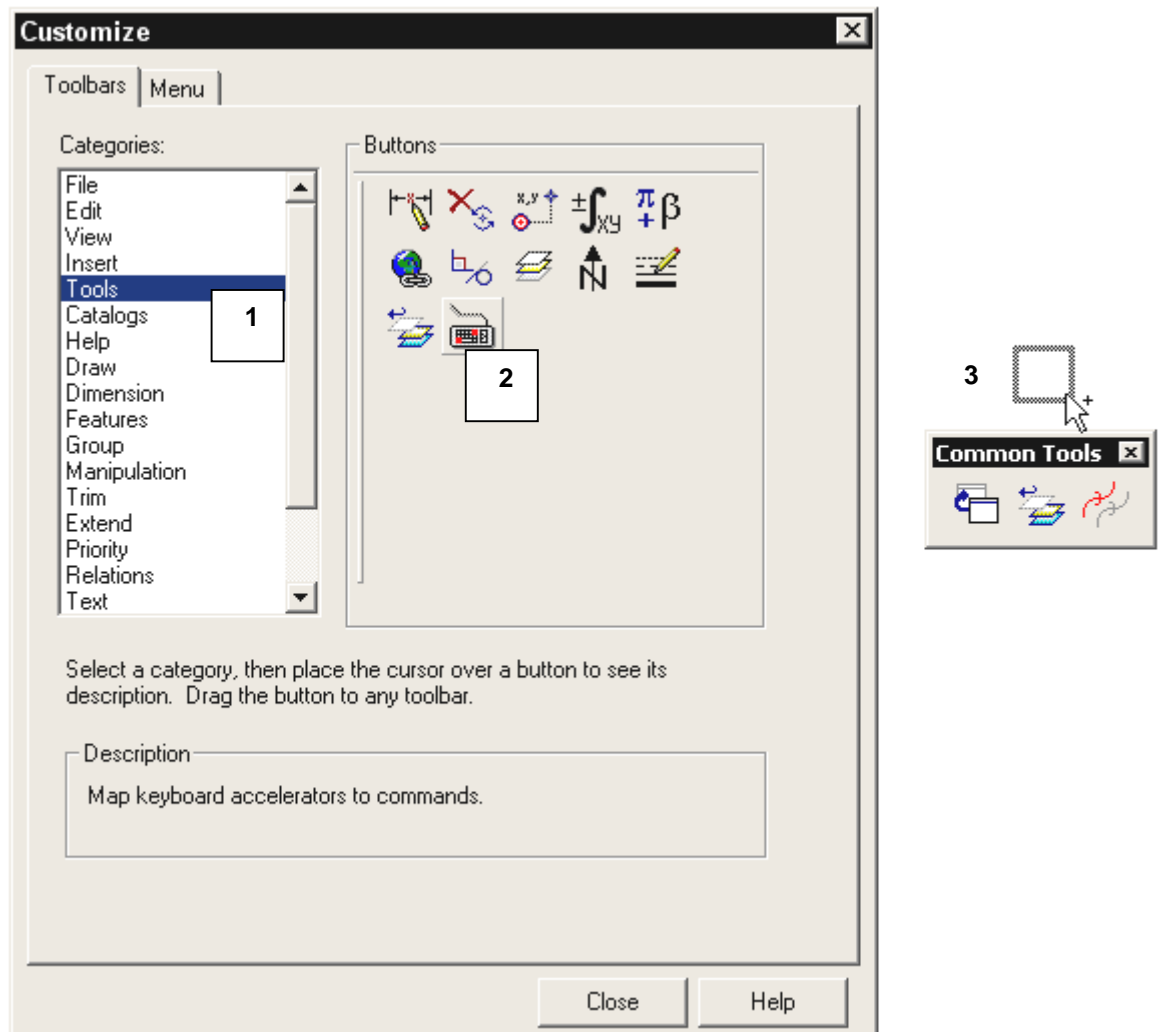
2. In the Toolbars dialog box, click the **New** button.



3. On the **New Toolbar** dialog, type the name of the new toolbar. When you click **OK**, the **Customize** dialog box and the new toolbar will appear (in the upper left corner of the application window) so that you can add buttons to the new toolbar.

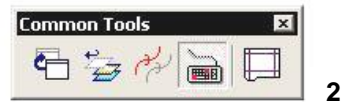


4. On the **Toolbars** tab, click the category that contains the command to be added. **(1)**



Document Management

5. Drag the command button **(2)** from the **Buttons** display area to the toolbar. **(3)**
6. Continue dragging the desired commands to your new toolbar.

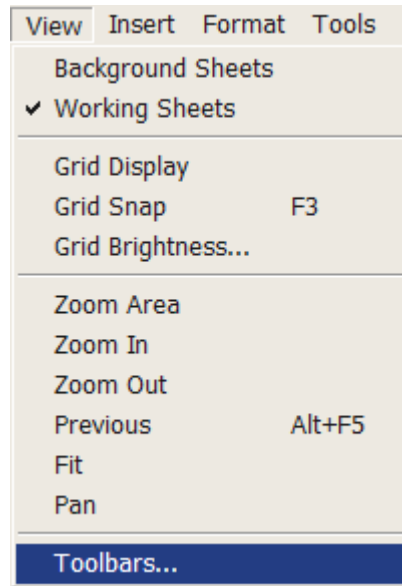


7. Click the **Close** button when finished creating the toolbar.

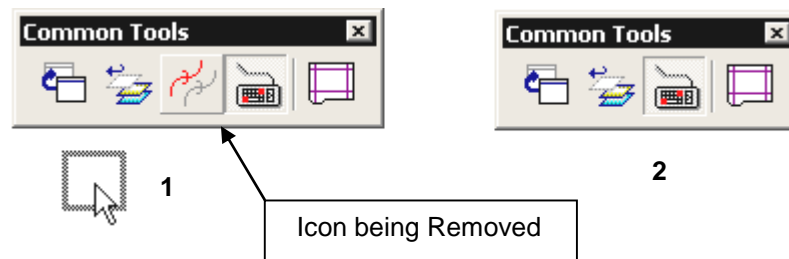
NOTE: The changes that you made to the toolbar appear only if a document is open.

To remove a command from a toolbar

1. Select **View** → **Toolbars**.



2. In the **Toolbars** dialog box, click the **Customize** button.
3. Select the **Toolbars** tab of the **Customize** dialog box.
4. Drag the button you want to remove from the toolbar (1) and drop it outside the toolbar boundary. (2)

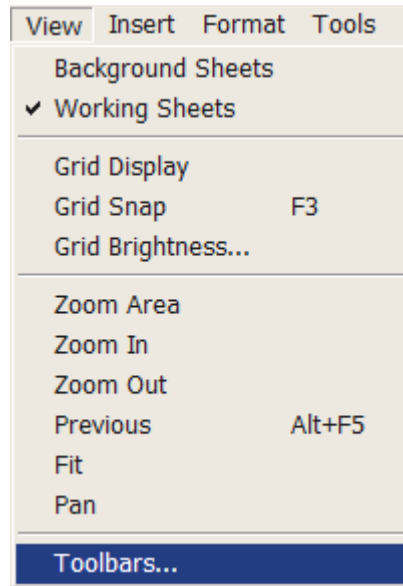


5. Click the **Close** button when finished removing commands from the toolbar.

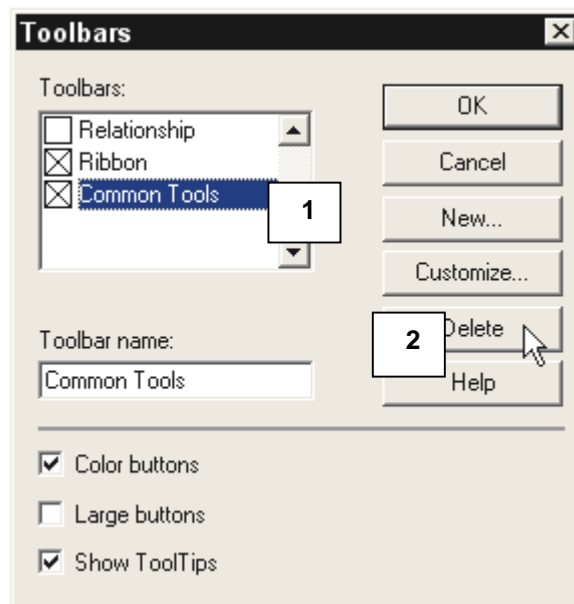
NOTE: Any toolbar can be modified while using the Customize command. Modifying a default or delivered toolbar could remove necessary tools, or cause confusion for other users on the system if unplanned alterations are performed.

To remove a toolbar that you created

1. Select **View** → **Toolbars**.



2. In the **Toolbars** dialog box, choose the toolbar that you wish to delete. (1)
3. Select the **Delete** button (2) and click **OK**.



4. The toolbar is removed as soon as you click the **Delete** button.

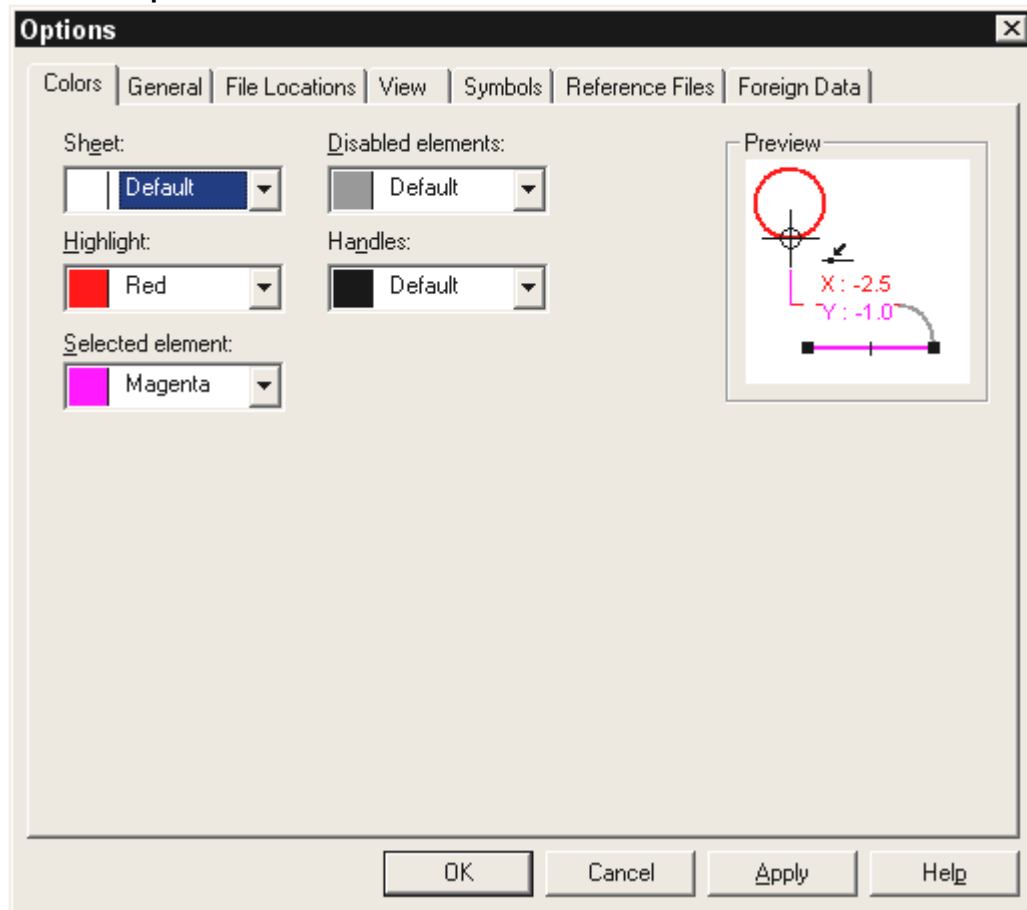
NOTE: You cannot undo the **Delete** toolbar command!

Document Options

Setting Document Colors

The colors used in the document for manipulating objects can be changed to meet your needs.

Click **Tools** → **Options** → **Colors**:



Sheet

Sets the background color of the document.

Highlight

Sets the object highlight color.

Selected element

Sets the color of the selected object.

Disabled elements

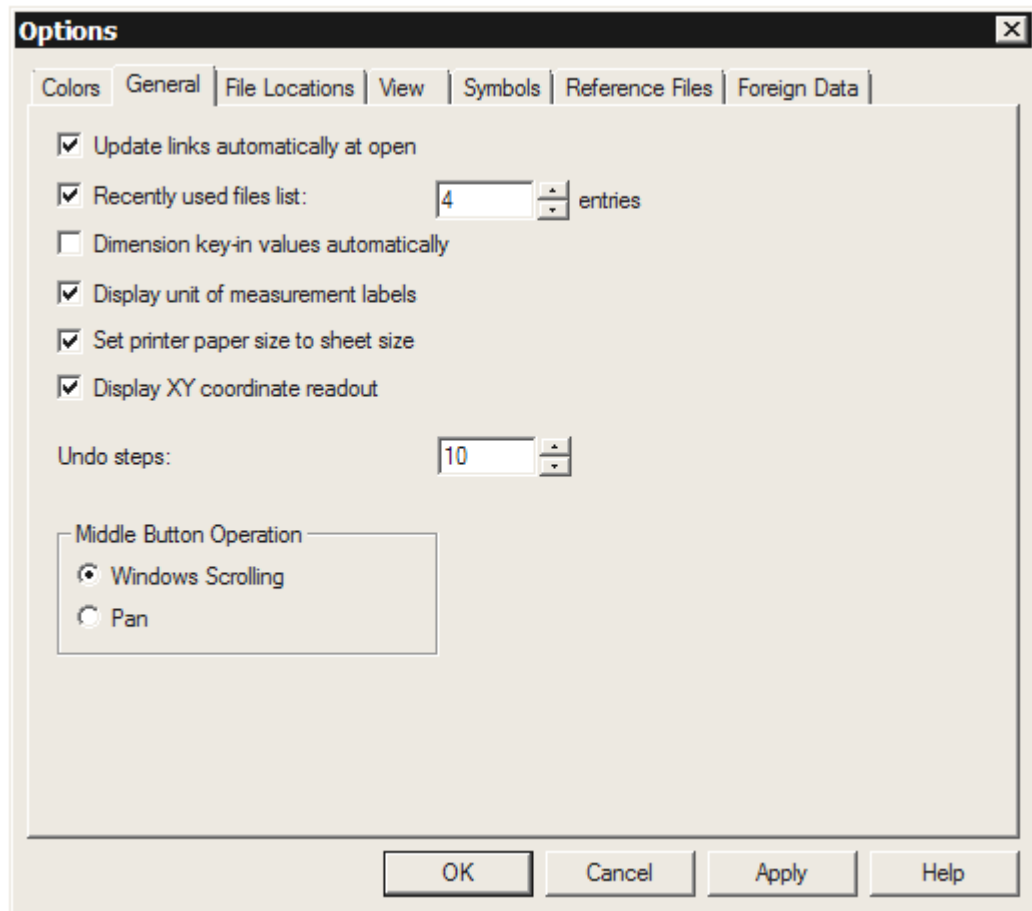
Sets the color of the disabled objects.

Handles

Sets the color of the handles on the objects when they are selected.

General Options Settings

Controls settings such as the display of 3-D effects and the number of entries in the list of recently used files.



Update Links Automatically At Open

Updates links automatically when the document is open and this option is set.

Recently Used Files List

Sets the number of entries for the **Recent Files** area of the **File** menu when the **Recent Files** area is set.

Dimension Key-in Values Automatically

Places dimensions for recognized step values or for values you type in a ribbon field while drawing a geometric object. For example, select **Rectangle** on the **Draw** toolbar. On the ribbon, type a value in the **Width**, **Height**, and/or **Angle** fields, press **Enter** or **Tab**, and then click the location in the **Drawing** sheet where you want to place the rectangle.

Display Unit Of Measurement Labels

Displays the units of measurement in the value field.

Set printer paper size to sheet size

Sets the printer paper settings to match the sheet size.

Display XY coordinate readout

The status bar display field will display the cursor's coordinate position on the drawing sheet when this option is turned on.

Undo Steps

Sets the number of operations that can be undone. You may enter a value anywhere from 1-100.

Middle Button Operation

Sets up the middle mouse button so that it can be used for either scrolling or panning. The default action of the middle mouse button is **Windows Scrolling**.

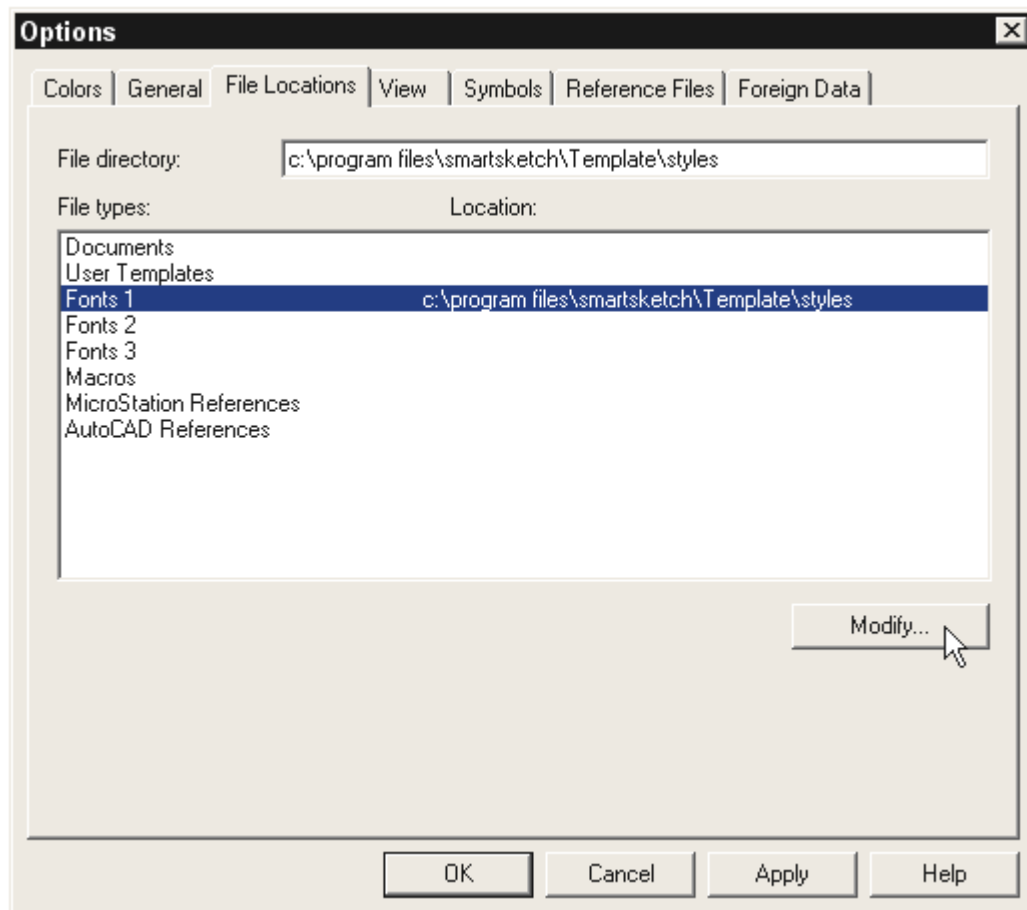
Setting Template and Other File Locations

The **File Locations** tab of the **Options** dialog box specifies the default location for documents, templates, and other objects you create or use in the software. You can set the location for the following types of documents:

- **Documents** - Set directory path used with commands such as **Open** and **Save As**.
- **User Templates** - Set location of user-created templates if not in default directory.
- **Fonts** - Set the path to other font resources such as AutoCAD® or MicroStation® fonts.
- **Macros** - Set location of macros used with **Tools> Custom Commands**.
- **MicroStation references** - Set location of MicroStation® references.
- **AutoCAD references** – Set location of AutoCAD® references.

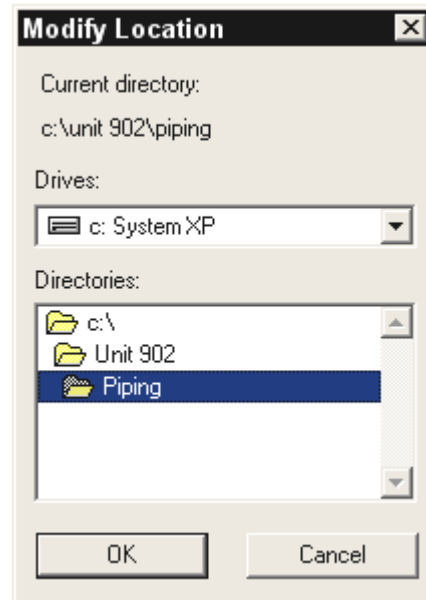
To Set File Locations

1. Select **Tools → Options**.



2. In the **Options** dialog box, click the **File Locations** tab.

3. Select one of the file types in the list and click the **Modify** button.
4. In the **Modify Location** dialog box, select the directory where you want the software to look for the file type.



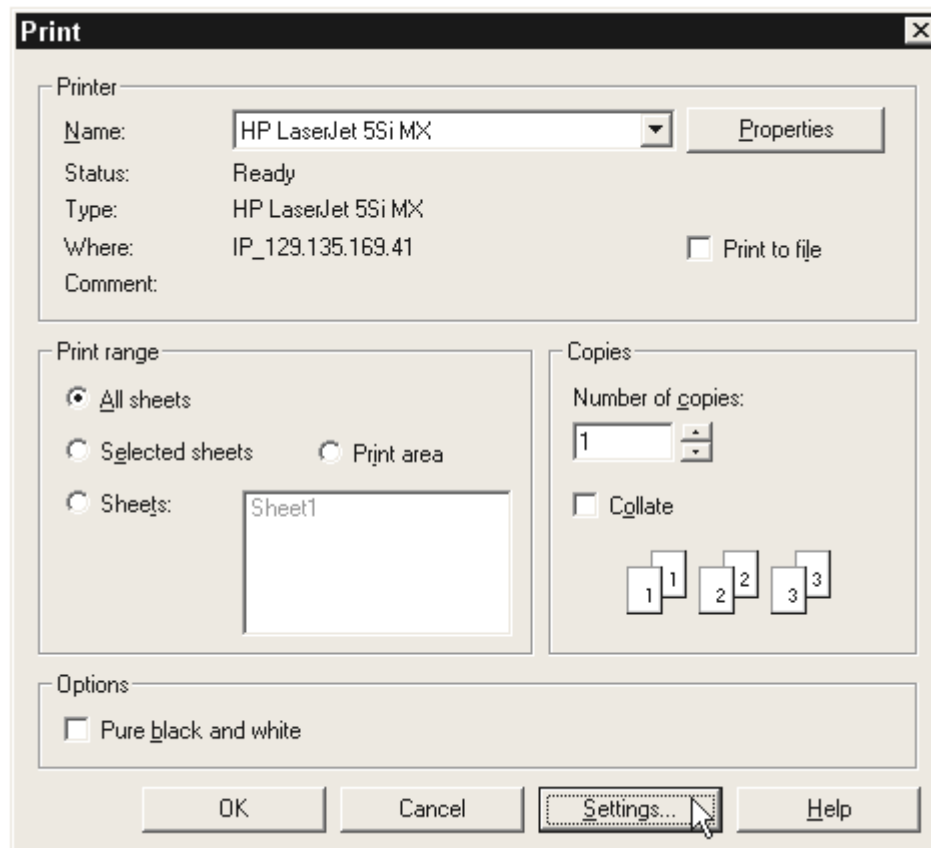
5. Click the **OK** buttons in both the **Modify Location** and **Options** dialog box.

Important: SmartSketch supports the system TrueType fonts delivered with Microsoft Windows® and supplies additional ANSI and ISO TrueType™ engineering fonts. You can make changes to this mapping to define your own font assignments from the **Foreign Data** tab on the **Tools → Options** menu. On the **File Locations** tab, you can set the paths to the font resources by double-clicking on Fonts 1, 2, and 3.

Printing a SmartSketch Document

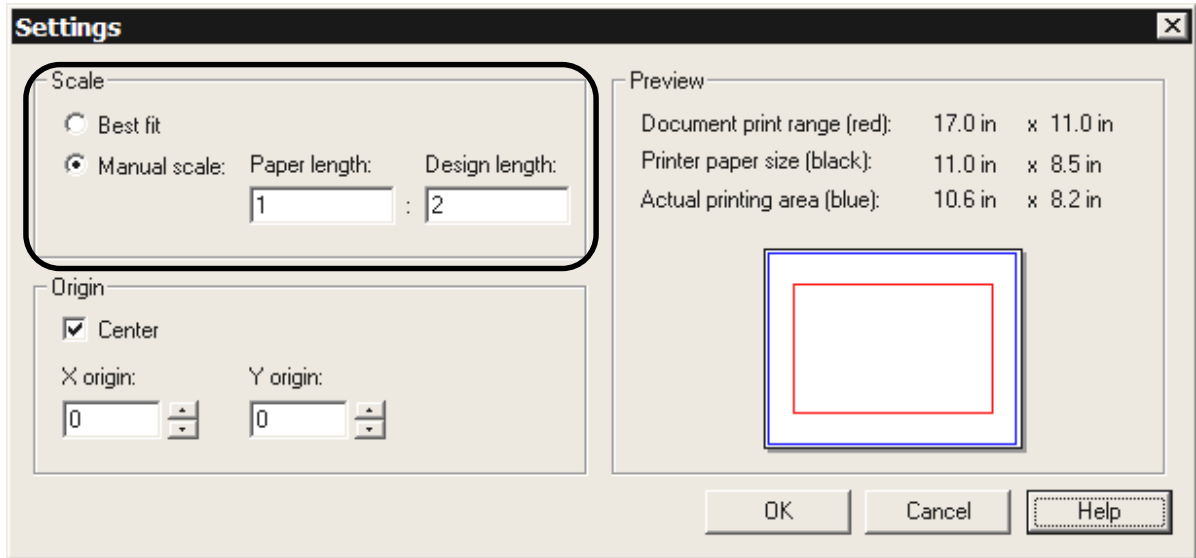
SmartSketch displays the standard Microsoft® print dialog options. This provides a simpler, more direct print setup, reducing the time spent setting up print controls. You can print your documents on many different devices ranging from dot matrix printers to high-end laser printers. Except for the color settings and a few special effects, your drawing should print exactly as it appears on the drawing sheet.

This section will explore a few special settings that are specific to printing a SmartSketch document. It will not cover the standard commands and settings on the **Print** dialog box. We will look at the **Print Scale** and **Print Area** commands.



Print Scale

Clicking the **Settings** button in the **Print** dialog box can set the scale applied to the print area of a document.



Best Fit

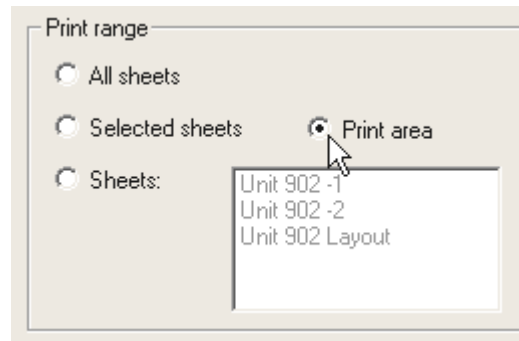
This option scales the selected drawing sheets or print area to fit the printer paper for the configured device.

Manual Scale

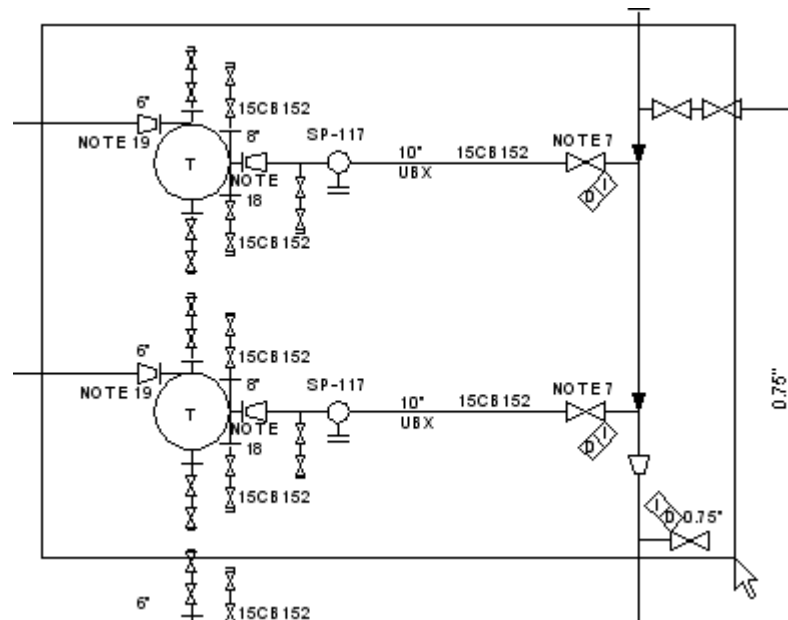
This setting specifies the scale value to apply to the print range during printing. For example, if the print range is a rectangle at 60 ft by 40 ft and you set a manual scale of 1:4, then the printed range appears as 15 in by 10 in on the printer paper.

NOTE: Save your document before you print it. This way, if a printer error or other problem occurs, you will not lose any work completed since the last time you saved the document.

Print Area



1. In the **Print Range** box, click the **Print Area** option and a set of crosshairs will appear in the document.
2. Drag the crosshairs on the drawing sheet to define the area that you want to print.
3. In the **Print Area** dialog box, set the scale and desired origin on paper. Click the **OK** button to submit to the printer.



NOTE: When using the **Print Area** method, if the setting is defined as **Best Fit**, there is very little likelihood that the print will conform to the scale for the sheet setup.

SECTION 4

Drawing Environment

This lesson will discuss the setup of the SmartSketch drawing environment to enable the user to more effectively use SmartSketch.

Topics

- Types of drawing sheets
- Sheet Setup
- Layers
- Grid and Grid Snap

Types of Drawing Sheets

There are two different types of drawing sheets: working sheets and background sheets.

Working Sheets

Working sheets are similar to pages in a notebook. Sketches or drawings can be placed on different drawing sheets in the document. For example, one idea can be drawn for a design on one drawing sheet and another idea on another drawing sheet. The sheet outline shows the orientation and default paper size for the sheet. The size and orientation of the sheet outline can be changed with the **Sheet Setup** command from the **File** menu. New graphic objects can be drawn within the sheet outline or outside the borders of the outline.

Background Sheets

A background sheet is used as a backdrop to the working sheet. Each document template supplies several background sheets. You can use any background sheet delivered in the template or design a custom background sheet per your requirements.

Background sheets can be customized to conform to corporate standards and requirements. Adding company logos, setting various styles and options can create tailor-made background sheets. You may also add common graphics that rarely have a need to change from project to project to a background sheet. A typical customization scheme would be to have a different background sheet for each standard-size drawing (A, B, C, D, or E).

Sheet Tabs

Sheet tabs allow you to manipulate and display the drawing sheets in the document. When opening a document or creating a new one, the tabs are displayed at the bottom of the workspace. There is a tab for each drawing sheet in the document.

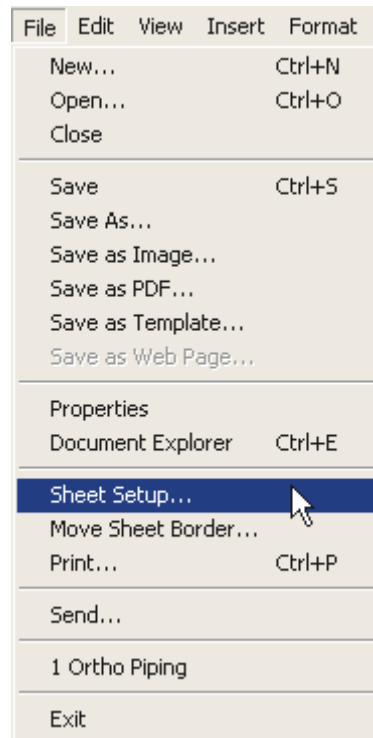
Using tabs to select drawing sheets is like selecting items in a list. Drawing sheet tabs can be used to do the following:

- To select and display a drawing sheet, click a tab. The drawing sheet is displayed and the name of the drawing sheet appears in bold. This drawing sheet becomes the active working sheet. There can only be one active sheet in a document.
- To display a group of drawing sheets, press and hold **Ctrl** and click one or more tabs. To select a range of drawing sheets, click the first tab in the range, hold **Shift**, and then click the last tab in the range.
- To turn off the display of a sheet in a group, hold **Ctrl** and click the drawing sheet's tab. To turn off the display of all drawing sheets in a group, click any tab that is not part of the group.

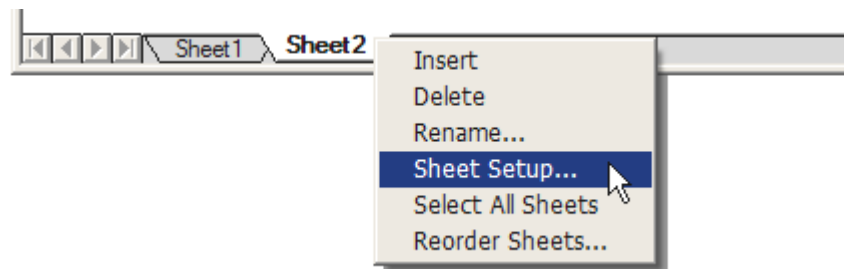
Sheet Setup

Each working sheet has its own setup. Working sheet characteristics, such as the size and scale of the sheet, can be modified with the **Sheet Setup** command from the **File** menu. After setting the options on the **Sheet Setup** dialog box and then selecting the **Save Defaults** button, all new sheets will have the same characteristics. Access the sheet setup dialog box by selecting sheet setup from one of the following:

1. From the **File** menu, select **Sheet Setup**.

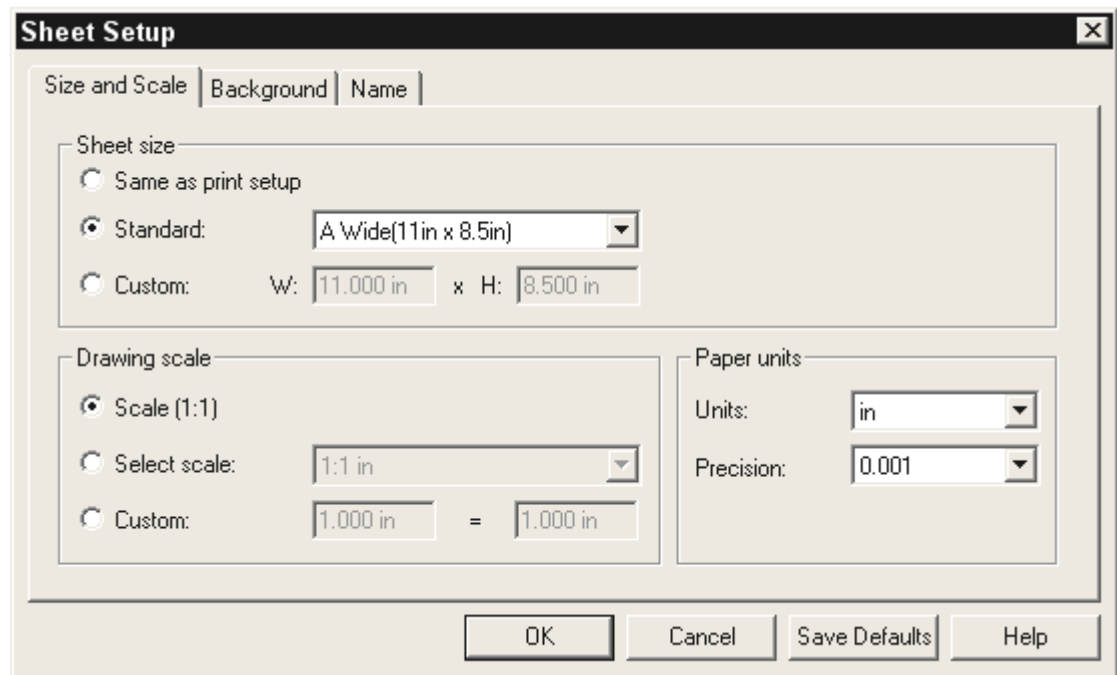


2. Position the mouse pointer over a selected sheet, click the right-mouse button, and select Sheet Setup from the shortcut menu, or just double-click on the sheet tab.



When the scale is specified using the **Sheet Setup** command from the **File** menu, all graphic objects on the sheet are scaled except dimensions and balloon annotations. For example, if the scale is one inch to ten inches (1:10) and a line 30 inches long is placed in the drawing, the line will be 3 inches long when printed. However, a dimension measures the length of the line as 30 inches.

- Dimension and annotation sizes in the sheets are independent of the view scale. For example, the height of dimension text will be printed at the specified size regardless of their screen appearance.
- Graphic objects on the attached background sheet are always displayed at a 1:1 scale. They are not affected by the scale factor for the working **Sheet Setup** command.



Size and Scale

Sheet Size

Sets the size of the drawing sheet.

Same as Print setup

Sets the drawing sheet size using the current print setup definition. For example, if the **printer** is set up as 8 1/2 X 11, the **drawing sheet size** is set up as 8 1/2 X 11.

Standard

Defines the drawing sheet size from a list of standard ANSI and ISO paper sizes.

Custom

Defines the drawing sheet size according to the entered x and y values.

Drawing Scale

Sets options for the scale of the drawing sheet.

Scale (1:1)

Sets the drawing scale to a 1:1 ratio. This means that the representation of the object on the drawing sheet is the same size as the real-world object being described.

Select scale

Sets the drawing scale to a standard ratio. The specified ratio defines the size of the drawing in relation to the size of the real-world object. For a **2:1** ratio, the **2** represents the size of the drawing and the **1** represents the size of the real-world object.

Custom

Defines a custom scale ratio. The first value defines the distance on the drawing sheet and the second value defines what the distance is equal to in the real world.

Paper Units

Sets the paper units for the drawing sheet.

Units

Controls the display of numeric values in dialog boxes that define the size of non-scaled objects. Some examples of non-scaled object values are text height and line width. This setting has no effect on dimension units.

Only value boxes use this option. For example, if working in feet and inches, the user can specify to read and enter values for text height in fractional inches.

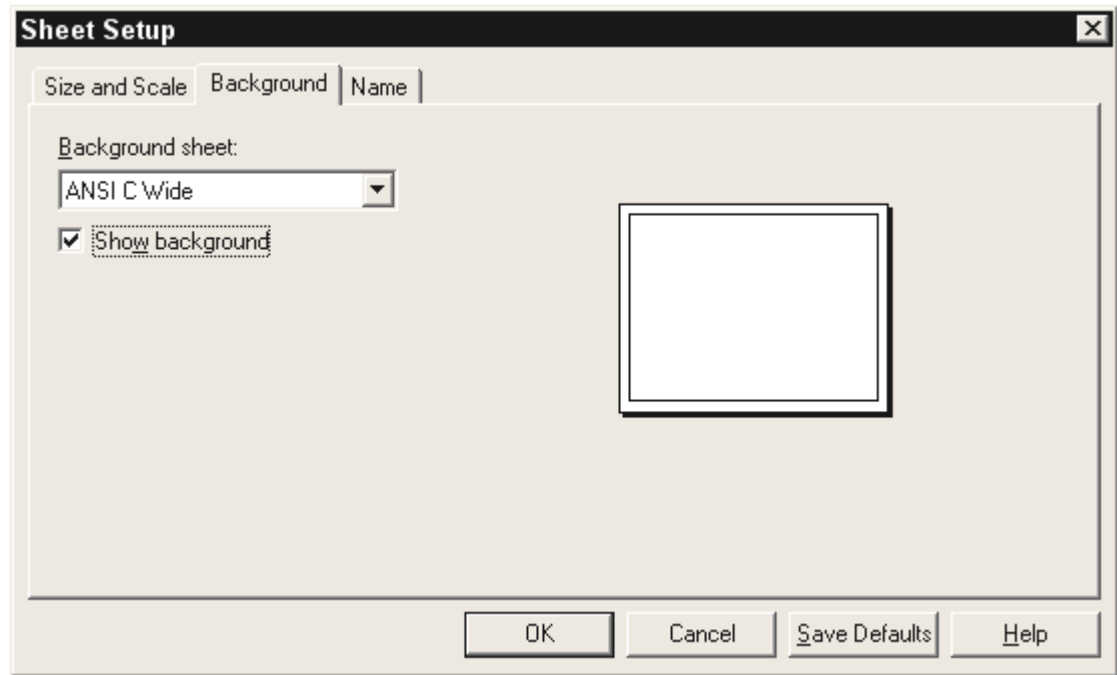
1/8" instead of **0.00'-0.125"**

Precision

Sets the number of significant digits to display, or the accuracy of the unit read-out value. The precision setting does not alter the numbers that can be typed into the fields, only the display of the numbers in the fields. For example, if you set this control to .001 and you draw a line that is 2.1056 inches, then the line length value is rounded up to display as 2.106.

Background Tab

Determines the background sheet display information.



Background sheet

Specifies which background sheet to overlay with the current working sheet. All graphics on the background sheet will be visible onscreen with the working sheet, but cannot be selected or changed while drawing on the working sheet.

Changing the background sheet changes the sheet size on the 'Size and Scale' tab. It displays the size and margin settings of the background sheet relative to the working sheet outline.

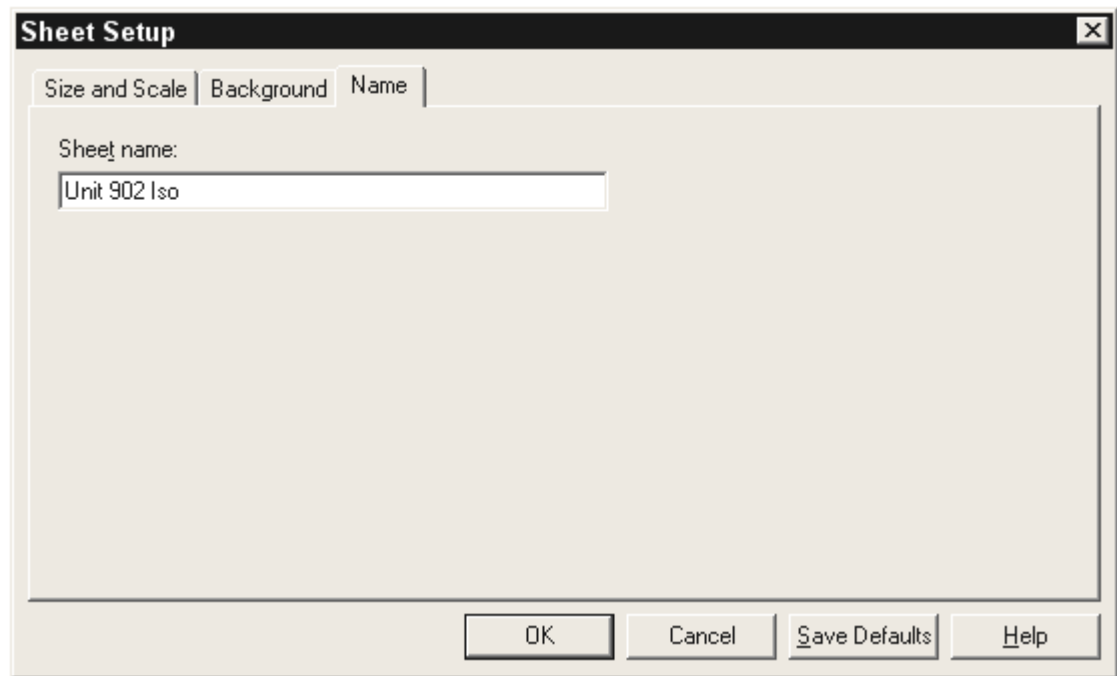
Not selecting a background sheet doesn't change the size or margins of the working sheet, it just removes the display of all graphic objects used from the previous background sheet.

Show background

If on, it displays the background sheet graphics on the selected working sheet. If off, it makes the background invisible, but leaves it attached to the working sheet.

Name Tab

Defines a name for the drawing sheet shown on the sheet tab.



Sheet name

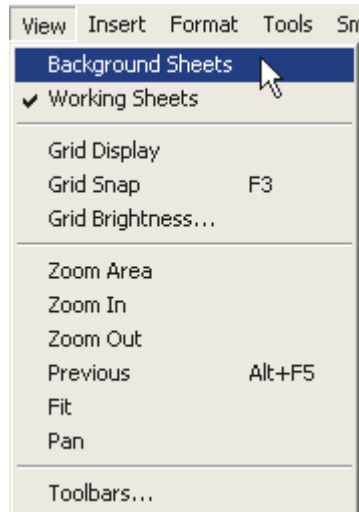
Specifies the name for the drawing sheet

Adding Background Sheets

When a background sheet is attached to a working sheet with the **Sheet Setup** command, the items on the background sheet are displayed and printed. The sheet size in SmartSketch is automatically set to the size of the background sheet being attached. This is so the paper sizes and graphics on both sheets will line up.

To add a new background sheet

1. From the **View** menu, click **Background Sheets**.

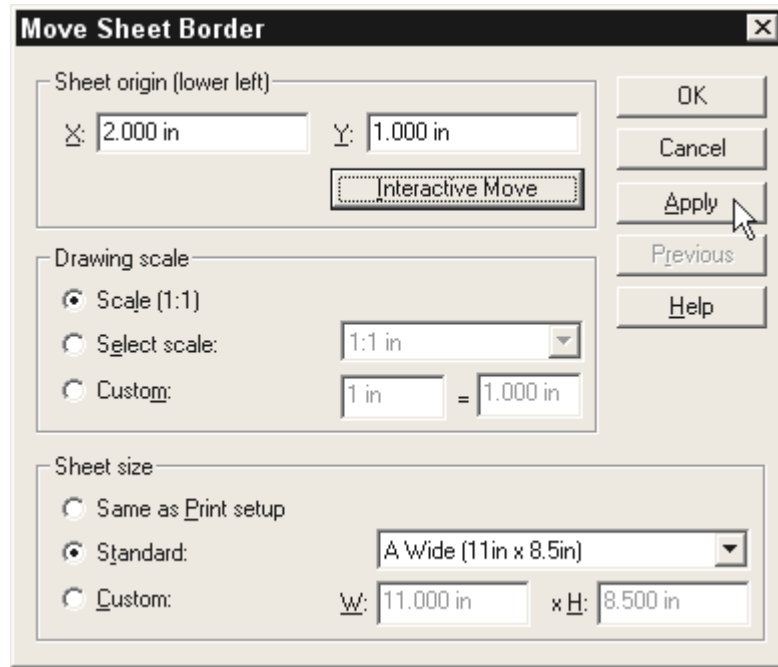


2. Select **Insert → New Sheet**. The background sheet is created using the default drawing sheet settings.
3. Select **File → Sheet Setup**.
4. On the **Sheet Setup** dialog box, set the desired options.
5. On the new background sheet, draw any graphics, such as title blocks and borders, which should be displayed on the working sheets.
6. Select **View → Working Sheets**.
7. Select **File → Sheet Setup** to attach the new background sheet to the working sheet.
8. On the **Sheet Setup** dialog box, select the **Background** tab.
9. Click on the **Background Sheet** list box and select the desired background sheet.

NOTE: Save as a template or a document in order to retain the new background sheet(s).

Move Sheet Border

This command is used to reposition the sheet border. You can also set the scale of a working sheet and select the paper size. The **Move Sheet Border** dialog box can be accessed by selecting **File → Move Sheet Border**.



The **Drawing Scale** and **Sheet Size** options are the same as those found in the **Sheet Setup** dialog box.

Sheet Origin Location

Specifies the X and Y coordinate for the sheet outline. The sheet origin defaults to the lower left corner of the drawing sheet. After setting the new coordinates, click **Apply**.

NOTE: You can click the **Apply** button to preview your settings. When you click the **Apply** button, the settings update the active sheet. However, the settings are not final until you click the **OK** button.

Interactive Move

Allows you to use the pointer to identify a location for the lower left corner of the sheet. Selecting this option automatically attaches the drawing sheet border to the pointer. Click to place the border at the desired location in the drawing. You must click the **Apply** button for your changes to be implemented.

NOTE: While you are moving the sheet border with the pointer, the **Move Sheet Border** dialog box is not displayed. It appears again after you click the pointer to complete the move.

Working With Layers

The **Layer** ribbon bar may be activated from the **Layers** icon on the **Main** toolbar, as well as by selecting **Layers** from the **Tools** pull-down menu.



Layers can help group graphic objects so they can be manipulated more easily on a drawing sheet. Layers also make it easier to keep track of different types of objects. Each drawing sheet in a document can contain an unlimited number of layers. This is similar to each sheet having many transparencies.

Layer Ribbon Bar

The Layer ribbon bar is used to display the active layer, create new layers, change the active layer for drawing or for existing graphic objects, and display the layer status. The ribbon bar may be docked at the top or the bottom of the SmartSketch application.

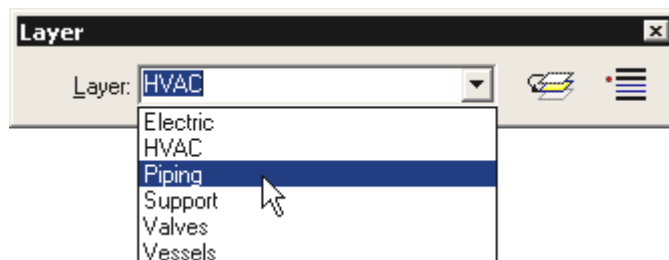


When adding a graphic object to the drawing, it is assigned to the active layer. All drawing sheets from a default template will have a layer named **Default**. If no other layer is created, all graphic objects drawn will be placed on the **Default** layer.

- Typing a new layer name in the **Layer** ribbon bar creates a new layer.



- The active layer can be changed by selecting from a list of all the layers on the active sheet.

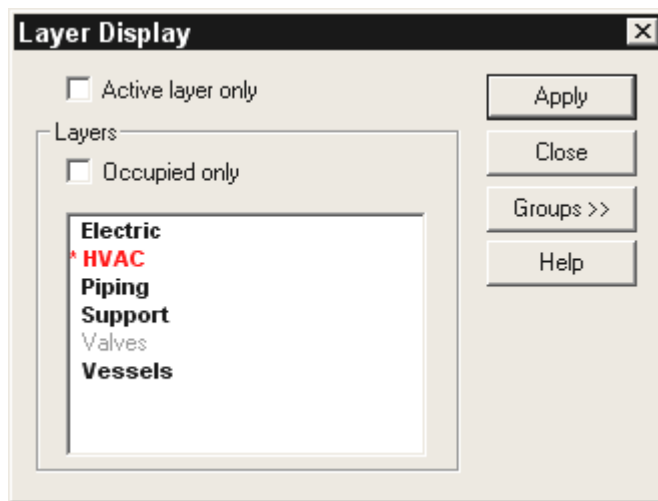


Displaying Layers

To view objects on specific layers, layers can be turned on or off with the **Layer Display** dialog box. The dialog box will be activated when the **Layer Display** icon is selected on the **Layers** ribbon bar.



When turning layers off, the objects assigned to those layers on the drawing sheet cannot be seen. For example, you can assign HVAC components to one layer and valves to another layer. If you turn off the layer for the valves, you see only HVAC components on the drawing sheet.



Active Layer Only

Displays just the active layer. If this option is set and the active layer changes, then the new active layer is displayed and all other layers are hidden.

Layers

Layer Display	Definition
Bold text	Layer is displayed in drawing sheet
Red text preceded by an asterisk	Active layer
Plain gray or dimmed text	Layer will not be displayed in the drawing sheet


Change Layer

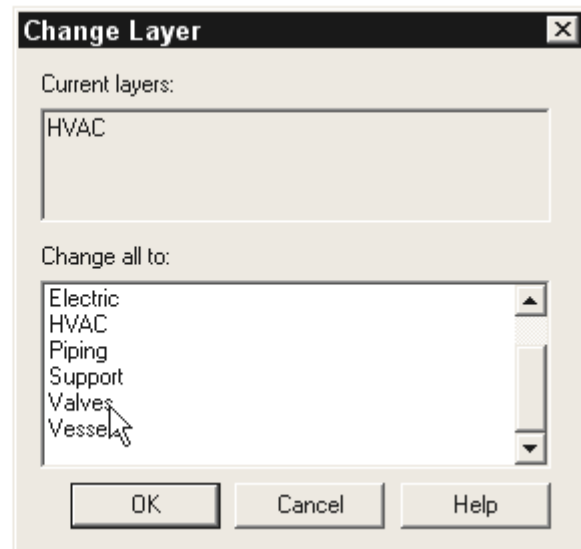
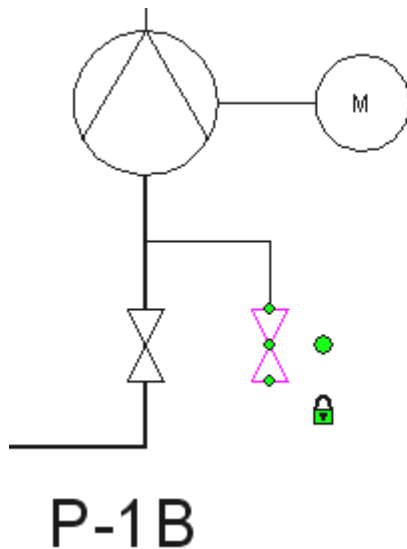
Accesses the **Change Layer** dialog box so that the layer a graphic object occupies can be modified. This option is available only when objects on the drawing sheet have been selected.



To change the layer of an object

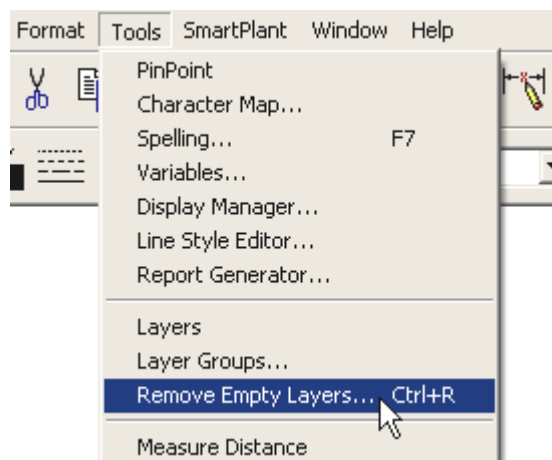
An object can be assigned to only one layer.

1. Using the **Select Tool** , select the objects to change to a new layer.
2. On the **Layers** ribbon bar, select the **Change Layer** button.
3. In the **Change Layer** dialog box, choose the layer that the selected graphic object will be moved to in the **Change All To** list and click the **OK** button.



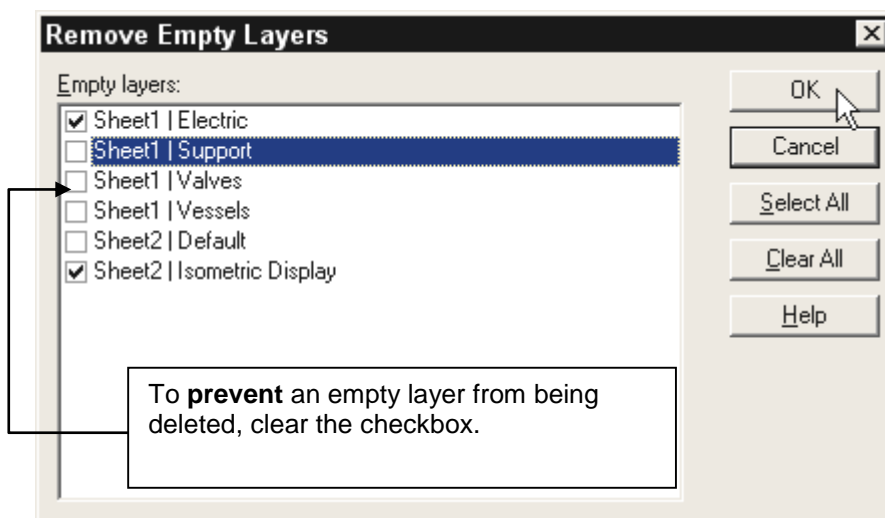
Remove Empty Layers

Removes layers from current SmartSketch document that do not contain data. To access the command select **Tools → Remove Empty Layers**. The shortcut key, **Ctrl + R** will also activate the command. There is a **Remove Empty Layers** icon available on the **Customize** dialog box under the **Tools** category; this icon can be added to a toolbar of your choice.



Once the **Remove Empty Layers** dialog box opens, select the layers to be removed from the document.

NOTE: When using the dialog box, use caution since by default all empty layers in the document are selected.

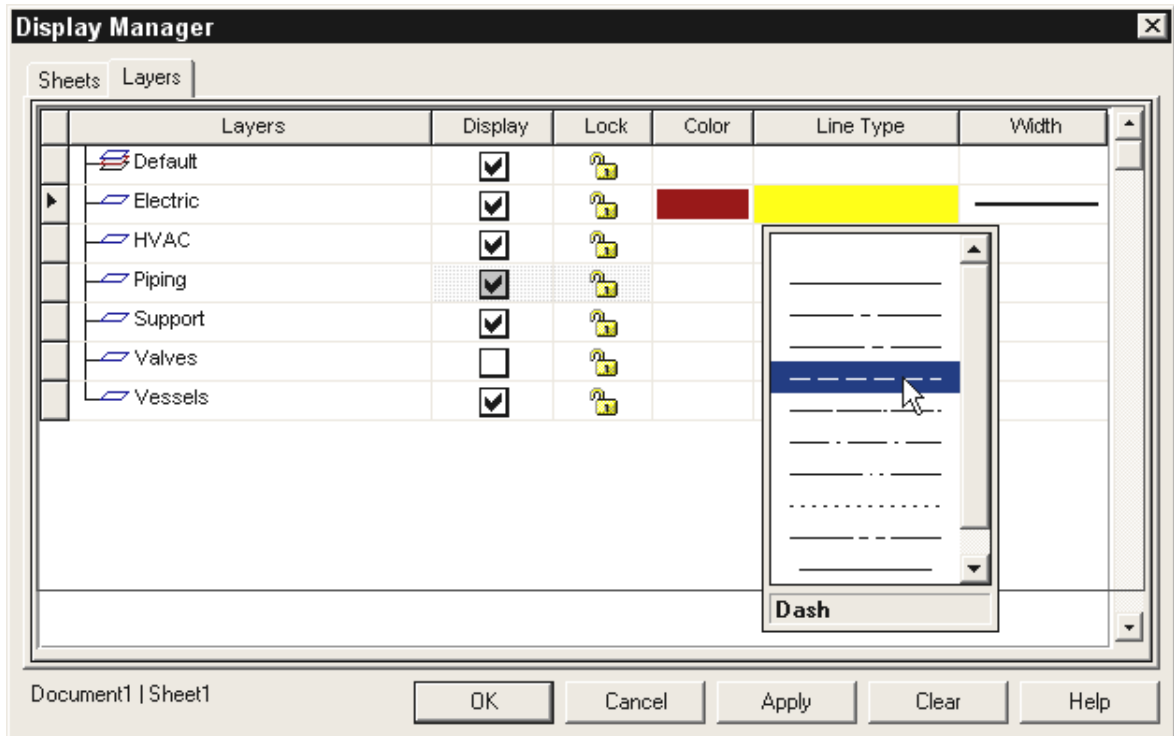


The active layer cannot be deleted, and as such will not be displayed in the dialog box check list.

Display Manager

Specifies the way objects appear in a window for all drawing sheets and their associated layers in the document. The color, line type, and width that you set in the **Display Manager** will override the graphic object's true color, line type, and width.

Select **Tools → Display Manager** to activate the **Display Manager** dialog box. On the **Sheets** or **Layers** tab, set the options you want to use for displaying objects on the drawing sheet. The color, line type, and width settings are applied to all the designated graphic objects on the sheet.

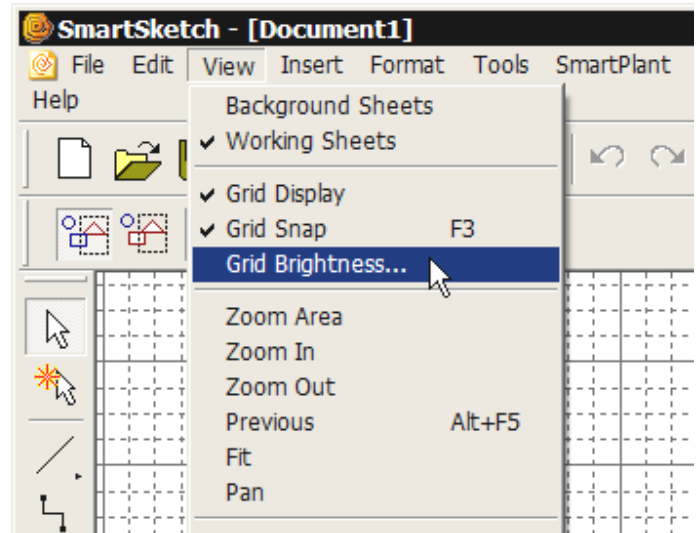
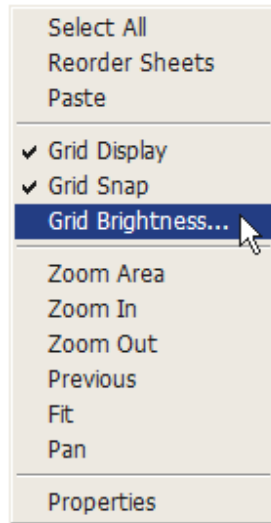


It is possible to lock objects in the drawing by using the **Display Manager**. The command locks or unlocks objects and layers in a view to control whether SmartSketch can select them. If a sheet or layer is locked, you cannot select graphic objects on those layers in the active document or drawing sheet. Conversely, if the lock beside a sheet or layer appears unlocked, the graphic object or layer is not locked and you can locate and modify it.

NOTE: The settings on the **Layer** tab override the settings on the **Sheet** tab in **Display Manager**.

Activating Grid Display and Grid Snap

Grid Display and **Grid Snap** can be turned on and off by clicking the right-mouse button in the drawing space to activate a shortcut menu or from the **View** pull-down menu.



Grid Display

Displays a grid so that graphic objects can be placed with precision. The grid lines are visual aids but are not considered part of the document and do not print.

Grid Snap

Aligns graphic objects with the grid while it is active. The grid can be used as an invisible set of lines in the document that helps to align graphic objects.

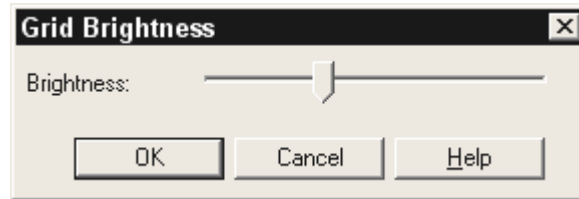
1. **Notes:** When **Grid Snap** is set, but the display is off, graphic objects always align with the nearest intersection of the grid lines whether they are visible or not.

Grid snap does not work while you identify objects that are aligned along grid lines. To override this, press the **Alt** key while identifying these objects.

Relationship and alignment indicators override the grid snap. You can suppress the indicators by pressing **Alt**.

Grid Brightness

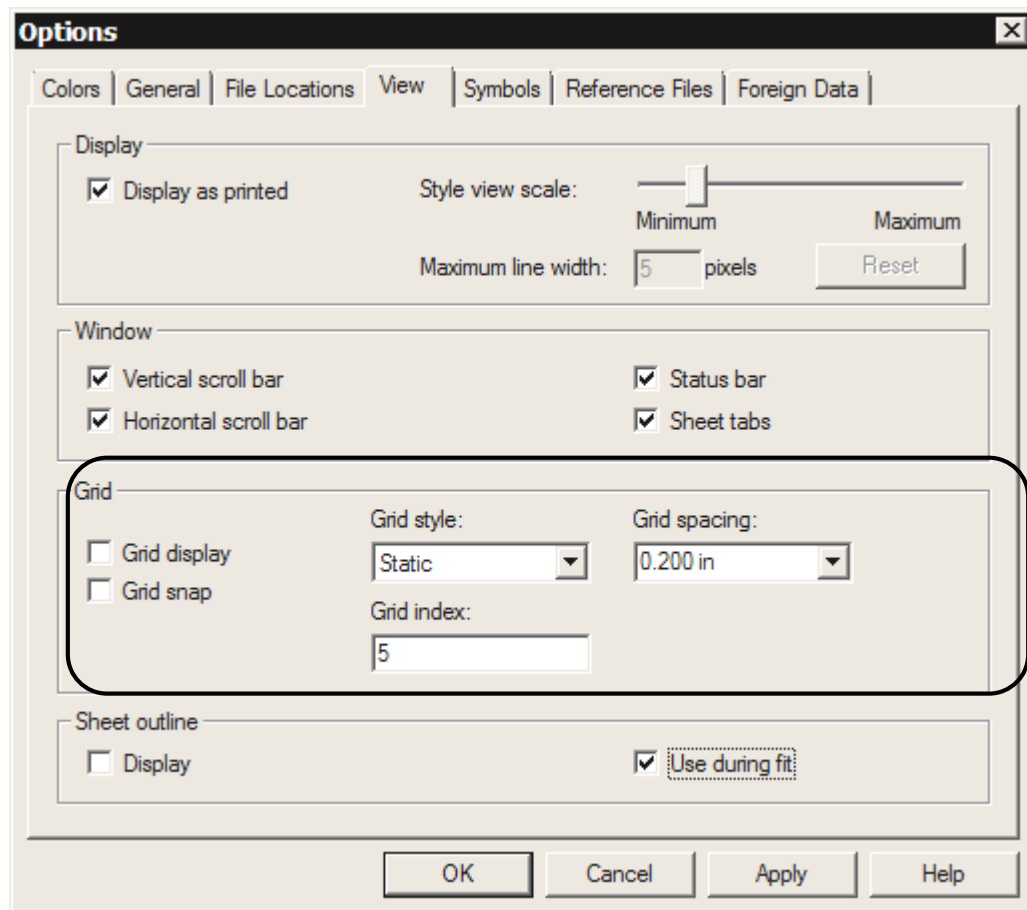
Grids are the net-like horizontal and vertical lines that appear on the **Drawing** sheet, and this setting allows the user to lighten and darken the intensity of the SmartSketch grid to enhance or subdue its impact relative to the drawing graphic objects.



Drag the slider to adjust the intensity of the SmartSketch grid. Moving it to the left lightens the intensity; moving it to the right darkens the intensity.

Grid Settings

The grid and its grid lines allow graphic objects in the document to be placed by aligning them with the grid lines or nearest intersection of the grid lines. The grid can be used to line up graphic objects at regular intervals in a document. The options to set up the grid can be found by going to the **Tools> Options** dialog box, and selecting the **View** tab.



Grid Style

Changes the format of the grid lines to either **Static** or **Dynamic**.

Static

A static grid displays solid grid lines that do not move as you zoom in or out. The grid maintains a constant minimum spacing.

Grid Spacing

Sets the spacing of the major grid lines. This option is available only if **Static** is selected in the **Grid Style** list box. The selected options on the **Units** tab of the **Properties** dialog box determine the units that can be entered, such as inches or centimeters.

Grid Index

Determines the number of index grid lines, also known as minor grid lines, to be equally spaced between the major grid lines. This option is available only if you select **Static** in the **Grid Style** list box.

Dynamic

Zooming in or out, the software dynamically generates the grid lines for a dynamic grid. Set dynamic grid lines to appear at fine, medium, or coarse levels using the **Grid Density** setting. The grid lines appear at common major measurement increments. A dynamic grid displays index lines that intersect with the darker, solid grid lines.

Grid Density

Changes the number of dashes in the index lines, or minor grid lines, between intersections with the grid lines. (This option is available only if **Dynamic** is selected in the **Grid Style** list box.)

SECTION 5

Basic Graphic Objects

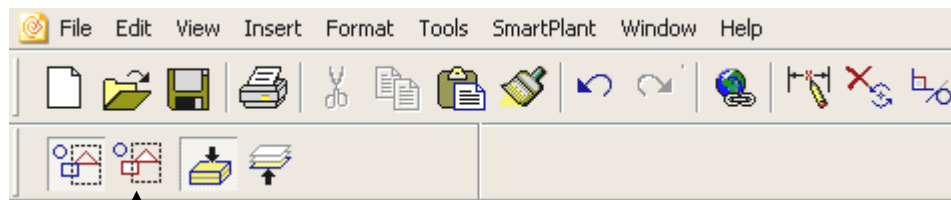
This lesson covers the tools used to place basic graphic objects in a SmartSketch document.

Topics

- Ribbon Bar
- Line Formats and Styles
- Basic SmartSketch drawing concepts
- Create lines
- Create arcs
- Create curves
- Create circles
- Create ellipses
- Create rectangles or squares
- Revision Cloud
- Create points
- Delete objects
- Undo and Redo commands
- PinPoint

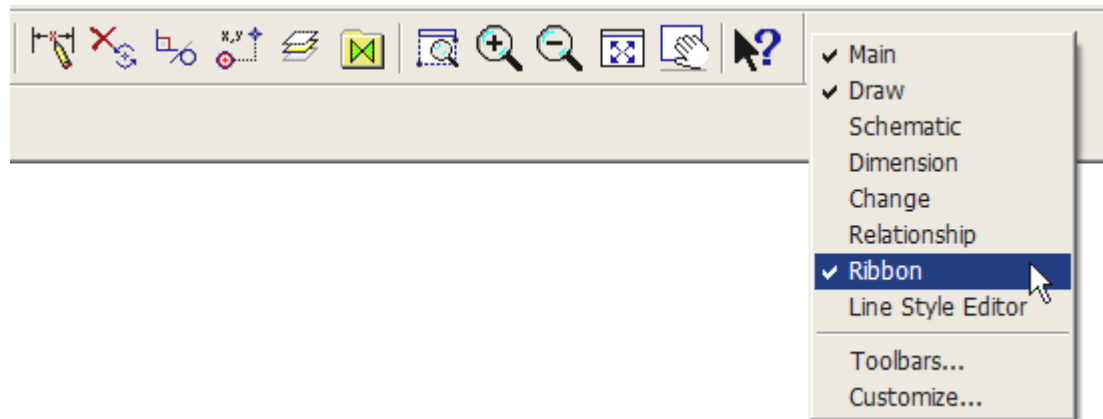
Ribbon Bar

The ribbon bar displays and helps you control various settings depending on the active command or object chosen in the drawing. Essentially the ribbon bar is a dynamic tool that adjusts the options available specific to the tool selected. For example, the ribbon bar displays different options for the **Arc by Center Point** command than it does for the **Rectangle** tool. The options on a ribbon bar work like options on dialog boxes. Any options that you set affect the active command. If you select an existing graphic object to modify it the ribbon bar will expand to display settings for that object. By default the ribbon bar is automatically toggled on when a document is opened in SmartSketch.



Default Ribbon Bar

A quick way to toggle the ribbon bar on and off is to click the right mouse button in the gray area around the toolbars and select **Ribbon** from the displayed list on the shortcut menu.



NOTE: A ribbon bar can be dragged and docked to the top or the bottom, but not to the sides, of the SmartSketch application window.

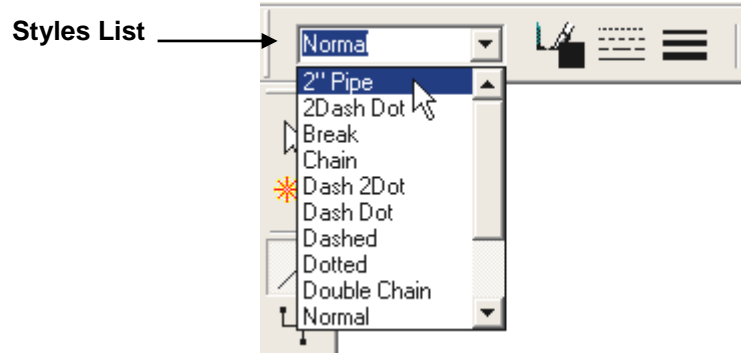
Common Ribbon Bar Settings

There are common settings that will appear on the ribbon bar when a command is chosen. These options are: style, line color, line type and line width settings. When the individual drawing tool commands are discussed, the ribbon bar options specific to that tool will be defined.

Style

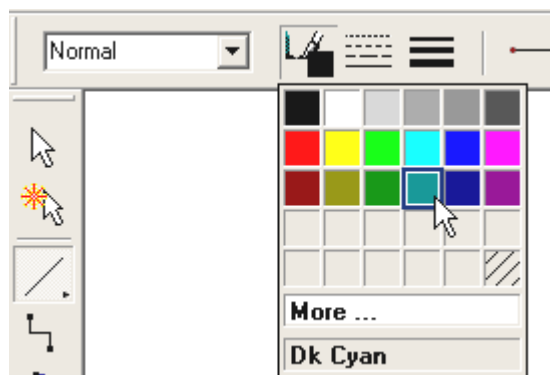
Styles are a collection of formats that can be applied to graphic objects such as lines and circles, text, and dimensions. Using styles ensures consistency within the drawing because the color, line type and line width are pre-defined. If you place or draw one or more objects, the format settings associated with the selected style are applied directly to the graphic objects. SmartSketch is delivered with several styles to choose from, and you can create or manage your own styles using **Format → Styles**.

A line style can be applied to any geometric component such as a line, arc, circle, or other closed shape. In later chapters, additional style formats will be discussed: text styles [applied to text within a text box], dimension styles, [which can be applied to a dimension or balloon] and fill styles.



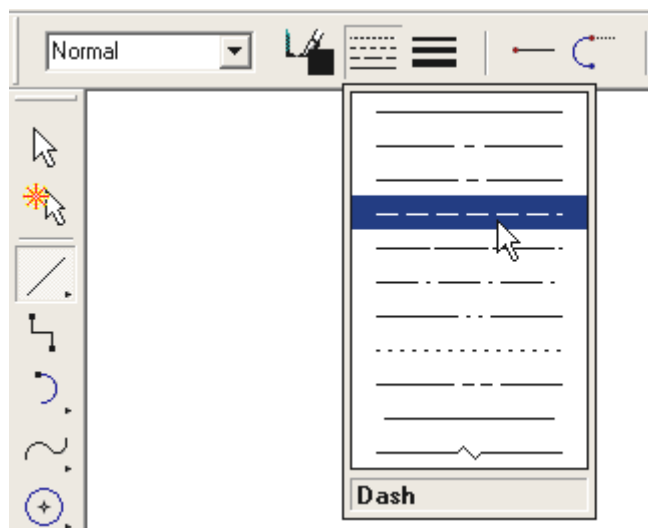
Line Color

Sets the active drawing color. You can click **More...** to define custom colors with the **Color** dialog box.



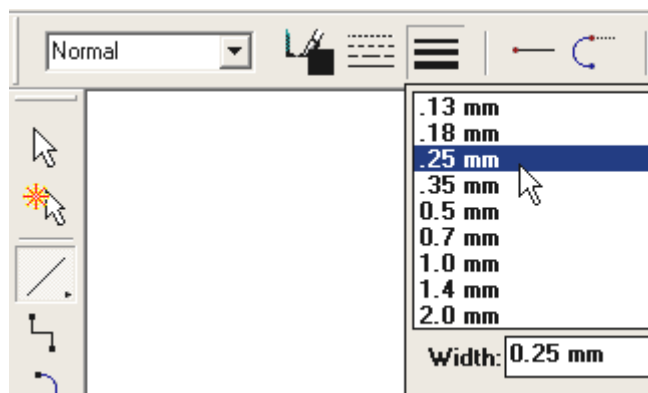
Line Type

Sets the active drawing line type.



Line Width

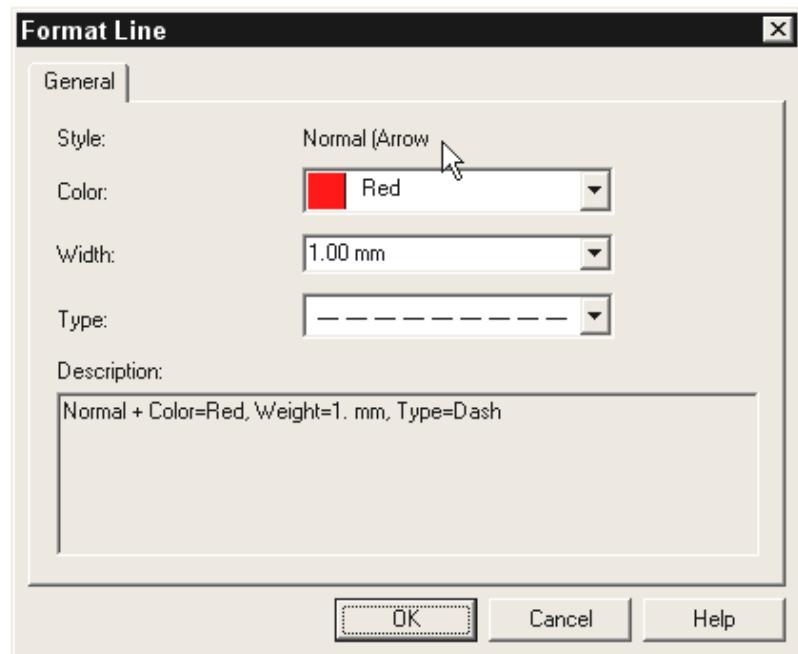
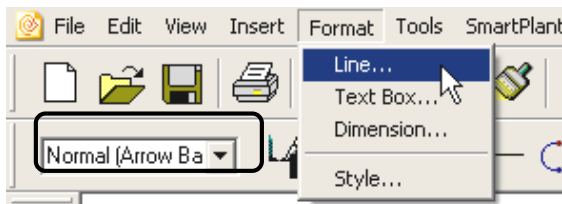
Sets the active drawing line width.



Format Line

Sets the appearance options for graphic objects such as lines and circles. You can access this dialog box by clicking **Line** on the **Format** menu. If the **Format Line** dialog box is setup in a template file, in essence you are determining the starting line format for any future files created from that template. The components that may be set in the **Format Line** dialog box is: Color, Width and Type.

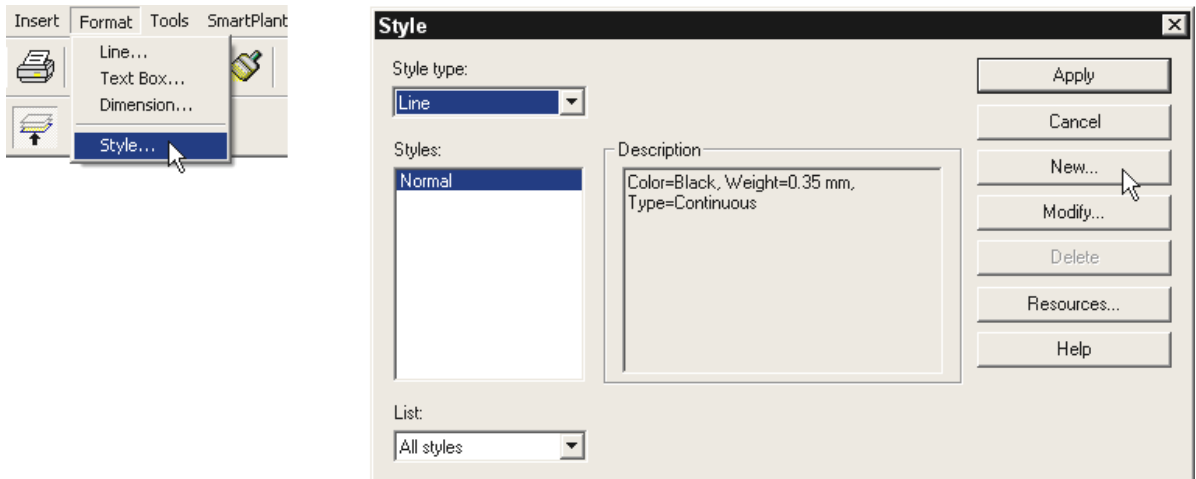
The **Style** option that is set in the ribbon bar will be the line style that is affected by the settings in the **Format Line** dialog box. In other words, if a drawing tool is selected and the **Style** set in the ribbon bar is **Normal (Arrow Backward)**, then the style to be formatted will be **Normal (Arrow Backward)**.



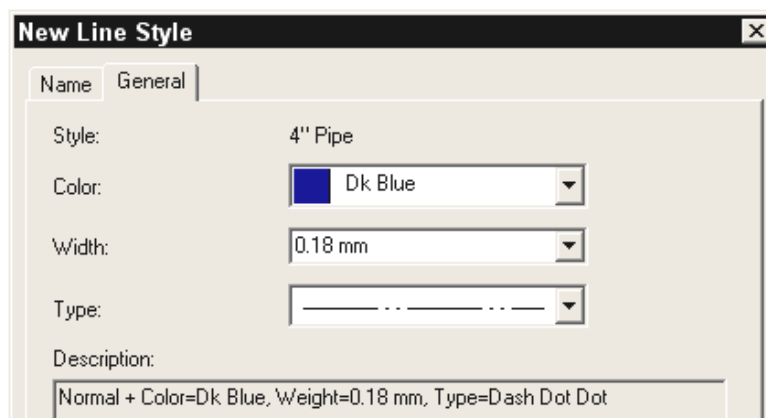
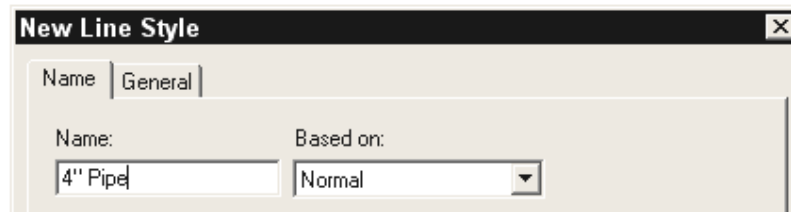
However, changing the current style settings on the ribbon or **Properties > Info** tab overrides the line style formats. Any settings that you made will be lost from that point on. Creating a line style will prevent the loss of the desired formats, and allow you to change **Styles** without losing any color, width or type settings.

Create a Line Style

Using the **Style** command on the **Format** menu, you can create or modify existing line styles to conform to any unique requirements for the drawing or project. The advantage of creating a line style, is that it may be saved to a template so that you can have access to the line style in future drawings, thus saving time when initially setting up a file.



Once the **New Line Style** dialog box opens, you can assign a name to the new line style, as well as associate it with a similar line style that may already exist. On the **General** tab of the **New Line Style** dialog box, you can set the Line Color, Line Type and Line Width of the new line style.



Basic Drawing Concepts

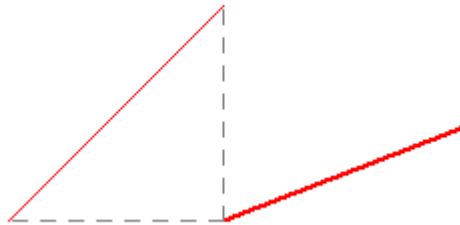
SmartSketch has three basic methods that you can use to add graphic objects to the drawing sheet once a drawing command has been selected from the **Draw** toolbar.

- **“Click”** method – place a graphic object in the drawing by clicking points on the sheet to define the parameters of the object.
- **“Key-in”** method – type in values on the ribbon bar that will define parameters of the graphic object being placed on the sheet. When a drawing tool is selected, the ribbon bar will display options that pertain to the active drawing command.
- **“Drag”** method - hold down the left mouse button after selecting a drawing tool to “sketch” the graphic object.

A combination of graphic and ribbon bar input can be used when placing graphic objects such as lines, arcs, circles and rectangles in the drawing space. SmartSketch indicators and PinPoint can also be used to place graphic objects in the drawing.

Alignment Indicators

Are dashed lines that show alignment while drawing or modifying objects. If the mouse is clicked when a horizontal or vertical indicator line is displayed, the point on the object being drawn or modified will be horizontally or vertically aligned with the object the indicator line leads to.



NOTE: Alignment Indicators may be enabled or disabled from the **Tools** menu. Pressing the **Alt** key will briefly suspend this display of **Alignment** indicators.

PinPoint

PinPoint is a tool that helps you draw and modify graphic objects relative to a known position in a drawing. PinPoint can be used with all drawing commands. It also works with SmartSketch indicators and the **Select Tool**. PinPoint will be discussed in more detail later in this lesson.

Create Line

Line/Arc Continuous

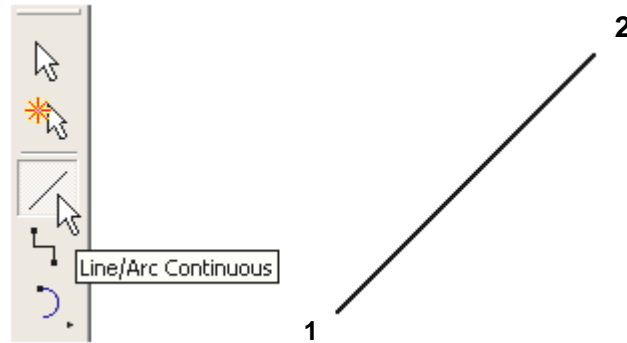


Draws one line or a series of connected lines. When you draw a series of lines, the second point of the previous line becomes the first point of the next line.

You can also draw a continuous series of lines and arcs that can be perpendicular or tangent to each other. Drawing lines and arcs in any combination can create an open or closed shape. The last point of the line or arc is the first point of the next line or arc.

Drawing a line

1. From the Draw toolbar, select the **Line/Arc Continuous** tool.
2. Place a point on the drawing sheet to define the start of the line. **(1)**
3. The line's length is defined by placing a second point in the drawing space. **(2)**
4. To terminate the command, click the right mouse button or press the **Esc** key.



Values can be typed in the **Length** or **Angle** fields on the ribbon bar. A combination of graphic and ribbon bar input can be used when placing the line in SmartSketch. The ribbon bar contains tools that allow you to switch the drawing mode between lines and arcs.



Line



Arc

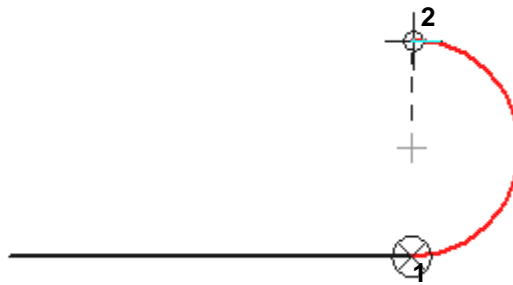
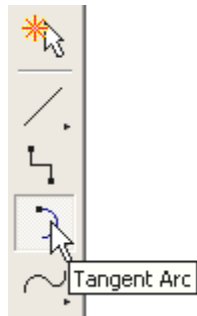
NOTE: **Shift+A** key to switch to arc mode and the **Shift+L** key to switch to line mode.

Create Arcs

Tangent Arc



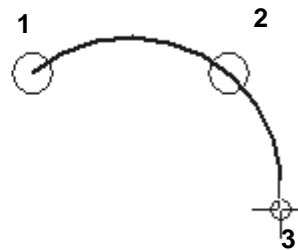
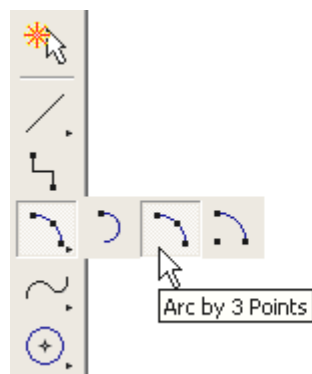
Draws an arc tangent or perpendicular to one or two objects. The end points are not tangent or perpendicular to other objects.



Arc by 3 Points



Draws an arc using three points. The first point defines the start of the sweep angle of the arc (anchor point). The second point defines the midpoint or radius of the arc, with the third point completing the sweep angle of the arc.



Arc by Center Point



Draws an arc using three points. The first point defines the center of the arc and the next two points define the sweep.



Create Curves

Curve



Draws a smooth, open or closed curve. A series of curves can have smooth, symmetric and cusp nodes. The node at the start point and the node at the end point of a curve are always smooth.



When drawing a curve, the open or closed setting for curves is located on the ribbon bar.



Open

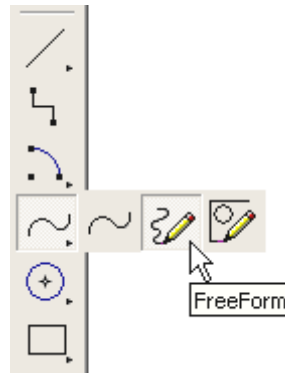


Closed

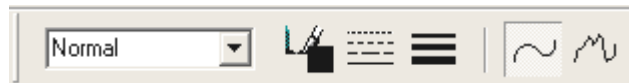
FreeForm



With the left mouse button held down, draw the object. The smoothing options on the ribbon bar will determine the amount of curvature.



If **Smoothing On** is selected, it draws cusp, smooth, and symmetric curves. If **Smoothing Off** is selected, it draws objects that closely resemble the movements of the mouse cursor.



Smoothing On



Smoothing Off

FreeForm Curve types

There are three different types of curves available when working with the **Curve** drawing tool or when **Smoothing On** is selected for the **FreeForm** command.

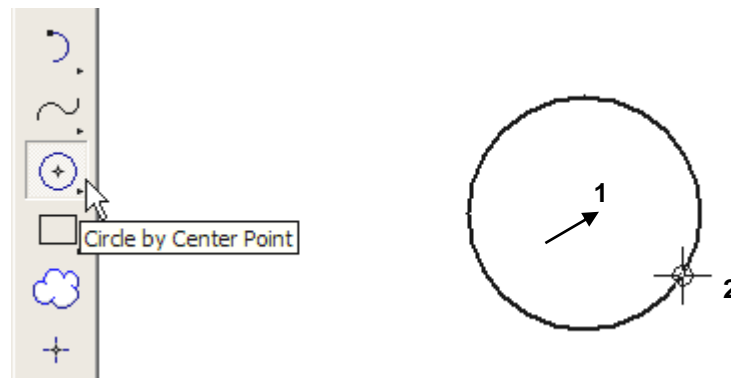
- **Smooth** - Gives a curve a different curvature on each side of the selected node. The start point and end point of a curve is always a smooth node.
- **Symmetric** - Gives a curve the same curvature on each side of the selected node.
- **Cusp** - Makes a curve bend sharply at the selected node.

Create Circles

Circle by Center Point



Draws a circle by first placing a center point and completing the circle by defining the radius.



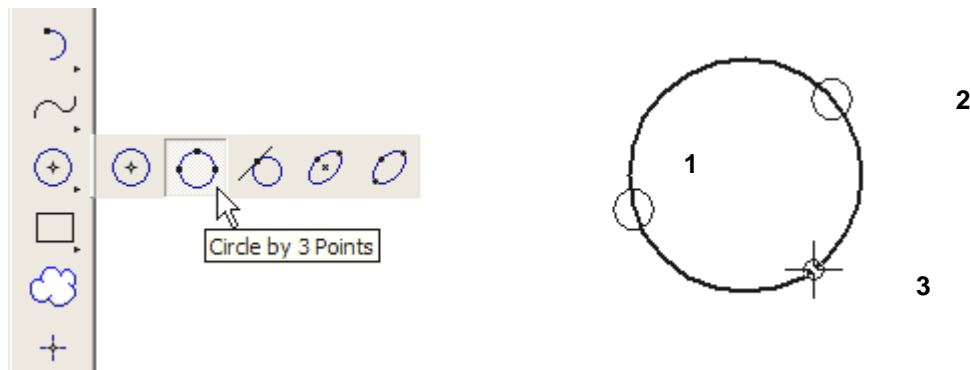
Values can be typed in the **Radius** or **Diameter** fields on the ribbon bar to determine the size of the circle.



Circle by 3 Points



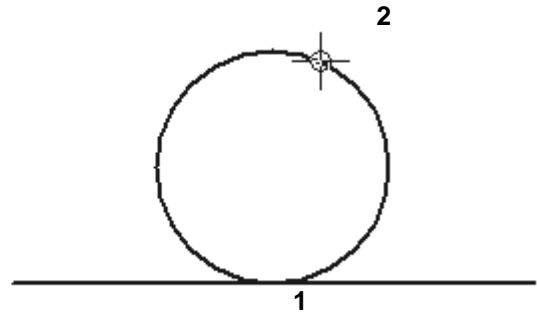
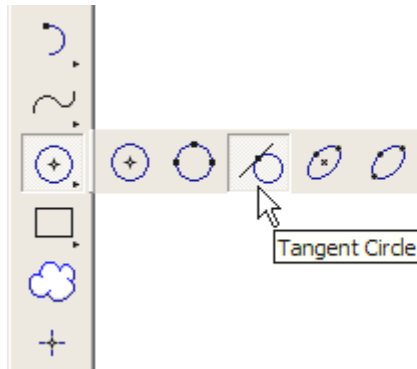
Draws a circle using three points that define the circumference.



Tangent Circle



Draws a circle tangent to one or two objects. The first point will attach the circle to the selected object(s), and the second point defines the radius or diameter of the circle.



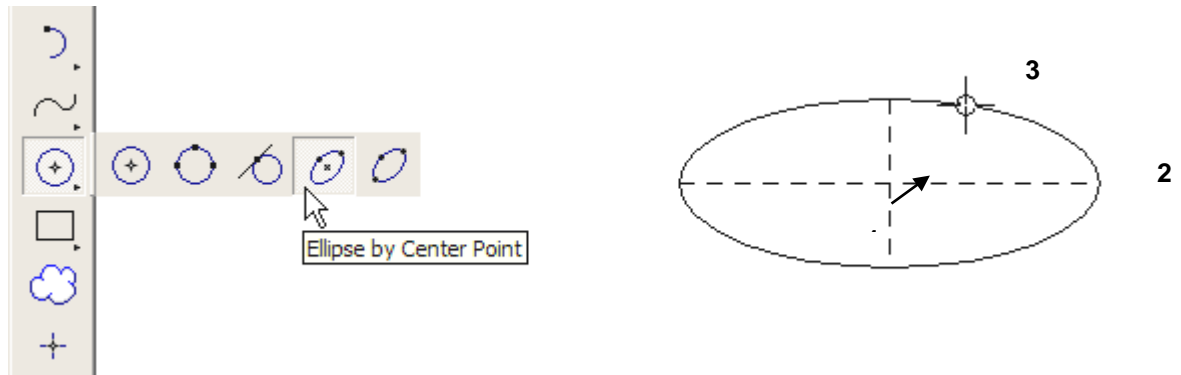
2. **Important:** It is possible to quickly draw a circle using the drag method. Simply hold down the left-mouse button and drag the cursor at an angle, guessing the proper width for the diameter. When you release the mouse button, SmartSketch will extrapolate the distance the cursor moved and use the data to draw a circle using that distance as a diameter definition. This will work with several other tools on the **Draw** toolbar.

Create Ellipses

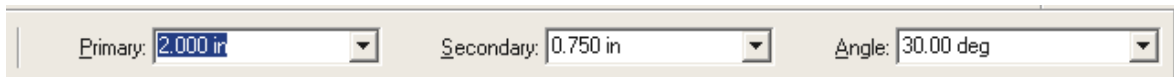
Ellipse by Center Point



Draws an ellipse using the center point and two edge points. The first and second points define half the length of the primary axis and the rotation angle. The last point defines the secondary axis.



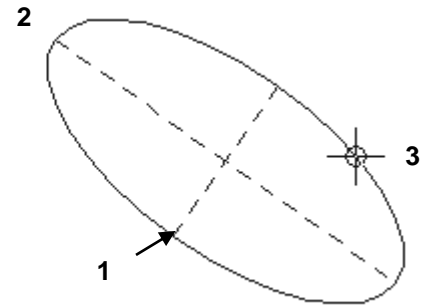
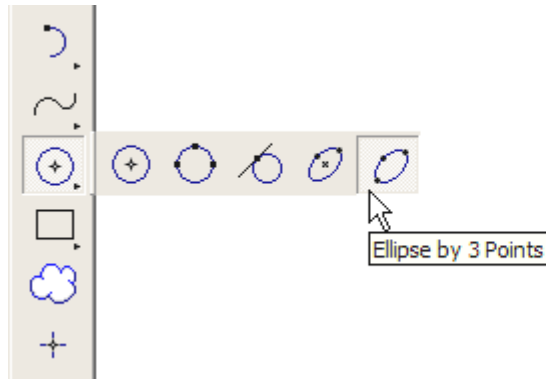
Values can be typed in the **Rotation Angle**, **Primary Axis**, and **Secondary Axis** boxes on the ribbon bar. A combination of graphic and ribbon bar input can be used.



Ellipse by 3 Points



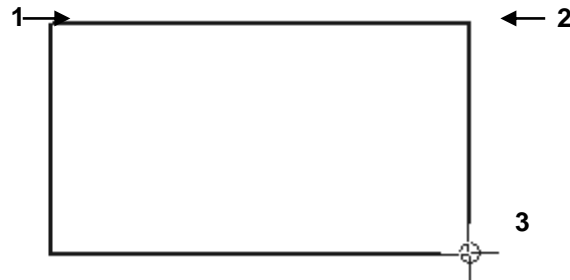
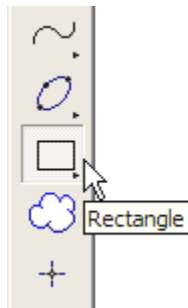
Draws an ellipse using three edge points. The first two points define the length of the primary axis and the rotation angle. The last point defines the secondary axis.



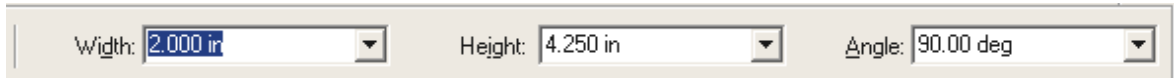
Create Rectangles or Squares



Draws a rectangle using three points. The first two points define the width and rotation angle of the rectangle, and the third point defines the height. It is also possible to draw a rectangle using the same drag method used to draw a circle, using a diagonal line.



Values can be typed in the **Width**, **Height**, and **Angle** fields on the ribbon bar.

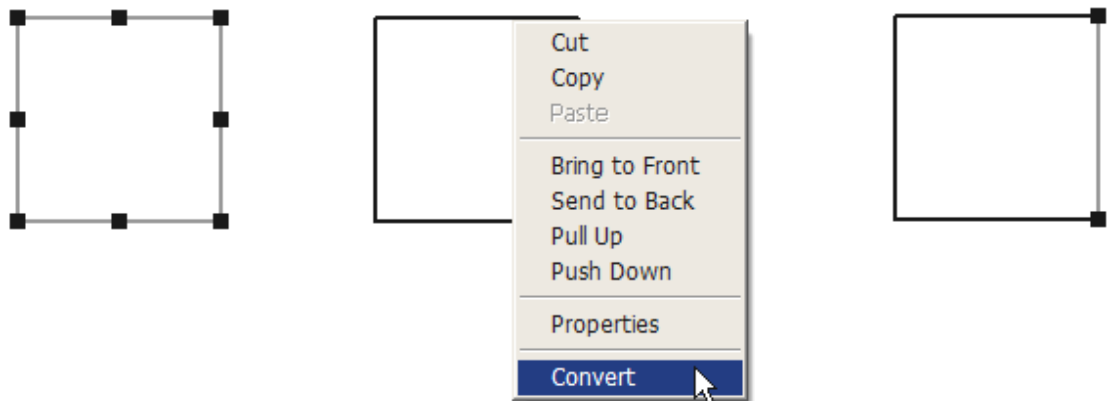


NOTE: To draw a square, hold the **Shift** key while defining the distance between point 1 and 2, then click. The command automatically makes the height equal to the width.

Convert Command (Rectangle)

Divides a rectangle into four individual line segments. To access the **Convert** command:

1. Point to the rectangle and right-click to open a shortcut menu.



2. Select **Convert**. The rectangle is then divided into individual line segments and right angles.

NOTE: The **Convert** command only applies to rectangles.

Revision Cloud



Places a cloud around objects in a drawing. Click a point in the drawing to indicate the start point for the revision cloud, and then move the mouse to sketch the revision cloud. As you move the mouse, the software displays the outline of the revision cloud in red. To close the revision cloud, click again.

Prior to placement, you can define an arc size - **Small**, **Medium**, **Large**, or **Random**. You can also place a label within the revision cloud or add semi-transparent fill within its boundaries.

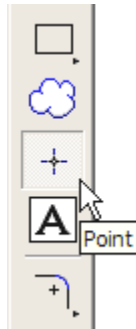
Arc Mode:	Random ▼	Caption:	<input type="text"/>	<input type="checkbox"/> Fill Cloud?
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NOTE: You cannot modify the shape or the size of a revision cloud after placement. To *move* a revision cloud after placement, select the revision cloud and drag it to the appropriate location.

Create Points

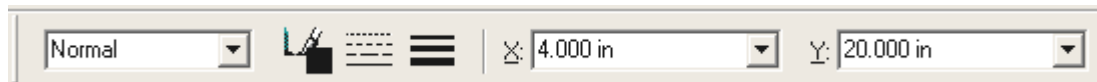


The **Point** command is used to draw points based on the current line width.




The line width setting will determine the size of the point

Instead of clicking to place the point, you can type values in the coordinate boxes on the ribbon bar. The default coordinate origin is located at the bottom left corner of the drawing sheet. You can also use a combination of graphic and ribbon bar input.



Delete Objects

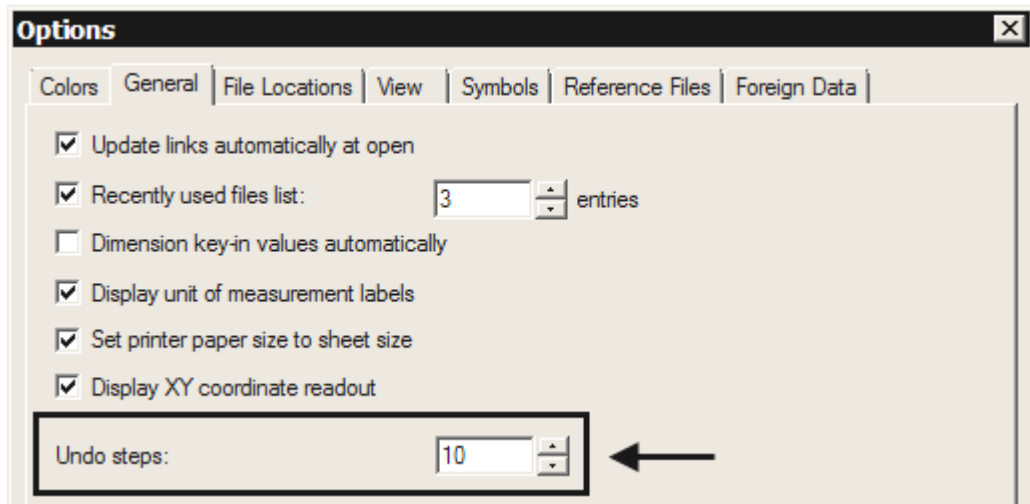


 The **Delete** tool is used to erase objects from the screen. In SmartSketch the graphic object must be selected first using the **Select** tool. The user can then choose the **Delete** tool or select the **Delete** key from the keyboard. Below is one method of deleting an object.



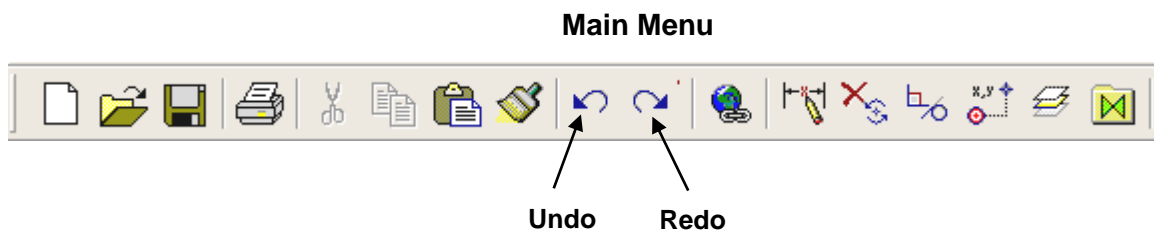
Undo and Redo Commands

The **Undo** command negates the last drawing operation that was performed; and **Redo** reverses the most recent **Undo**. The number of steps that can be undone can be set in the **Options** dialog box. Click **Tools** → **Options** → **General** tab.



NOTE: The number of undo steps is set in the Options dialog box. A maximum of 100 can be set for undo steps.

The undo buffer is reset to zero when a document is saved. Be aware that the **AutoSave** utility does impact **Undo** capability.

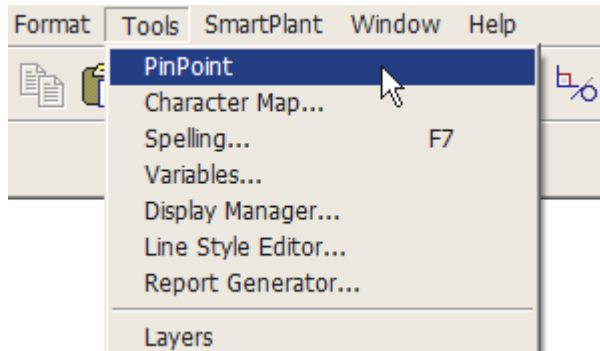


PinPoint



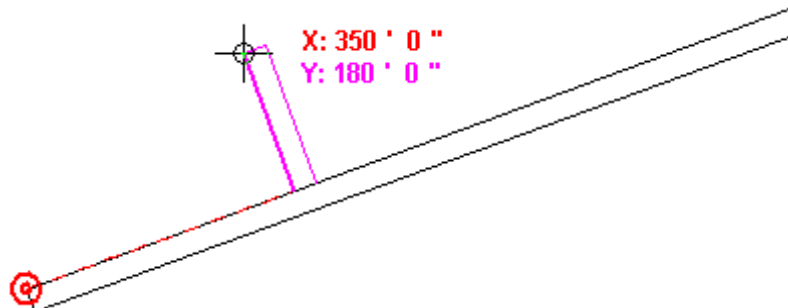
Helps you draw objects with precision by displaying coordinate data at the pointer. The displayed coordinates are relative to a target position that you set and can reset at any time. You can use **PinPoint** with all drawing commands.

To start **PinPoint**, click the **PinPoint** command from the **Tools** menu or select the **PinPoint** icon from the **Main** toolbar.



How PinPoint Works

PinPoint provides coordinate input to drawing commands. The x and y coordinates are relative to a target point that can be positioned anywhere in the application window. The location of the target point can be changed at any time by selecting the **Reposition Target** button on the ribbon bar and then clicking a new position in the application window.



NOTE: When **PinPoint** is activated, **Grid Snap** will automatically disable.

PinPoint Ribbon Bar



Display On/Off

Displays or hides the **PinPoint** help lines and distance values. **F9** is a shortcut toggle.



Reposition Target

Attaches the target point to the cursor so that it can be repositioned. Used to select where the target point is to be. **F12** can be used as an alternate method to move the target.



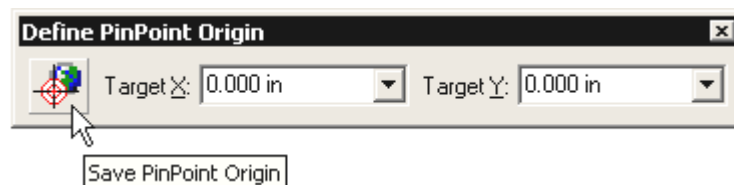
Relative Tracking

If activated the **PinPoint** target will move to the last point clicked while using the drawing tools.



Define PinPoint Origin

Allows the user to locate the origin in X, Y coordinate points for the document using the **Define PinPoint Origin** ribbon bar. The **Save PinPoint Origin** button on the ribbon must then be pressed in order to save this defined origin position within the document.

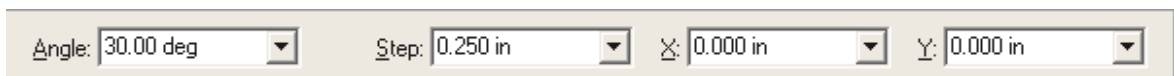


Reposition Target to Origin

on the **Define PinPoint Origin** ribbon bar.

Angle

In its default orientation, the **PinPoint** X-axis is horizontal. To re-orient it to any angle, set the angle on the **PinPoint** ribbon bar. Positive values rotate the horizontal line counterclockwise. Negative values rotate the horizontal line clockwise.



Step

Specifies the **PinPoint** step value. The step value is an incremental distance along the **PinPoint** coordinate axes. When the distance between the target position and the current cursor location is an increment of the step value, the related coordinate value and help line become bold. This can be useful to maintain consistent spacing between objects.

X:


Locks the horizontal distance between the target point and the current cursor location to the value specified. **F10** is a shortcut toggle for this lock setting.

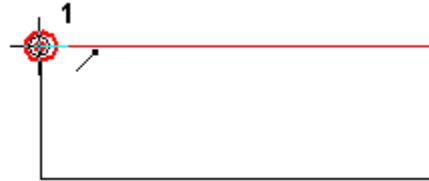
Y:

Locks the vertical distance between the target point and the current cursor location to the value specified. **F11** is a toggle to lock/unlock this setting.

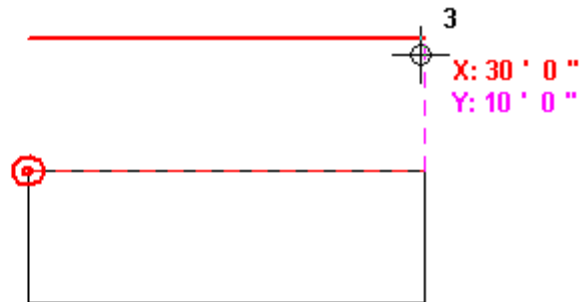
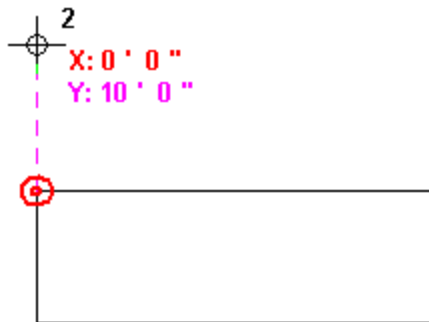
When moving the cursor, **PinPoint** dynamically displays the horizontal and vertical distance between the current cursor location and the target point. Help lines show the **PinPoint** X and Y-axes and the PinPoint orientation.

Draw with PinPoint

1. From the Main toolbar, select PinPoint .
2. Click where the target point is to be.



3. Select any drawing command. When moving the cursor, PinPoint displays the coordinates of the current cursor location in relation to the target point (2), (3). To provide precision input to the current command, press when the coordinate display indicates that the cursor is in the correct position or type coordinate values in the X and Y ribbon bar boxes.



NOTE: Hitting the **Esc** key will set **PinPoint** to the **0,0** coordinate position on drawing sheet (the lower left corner).

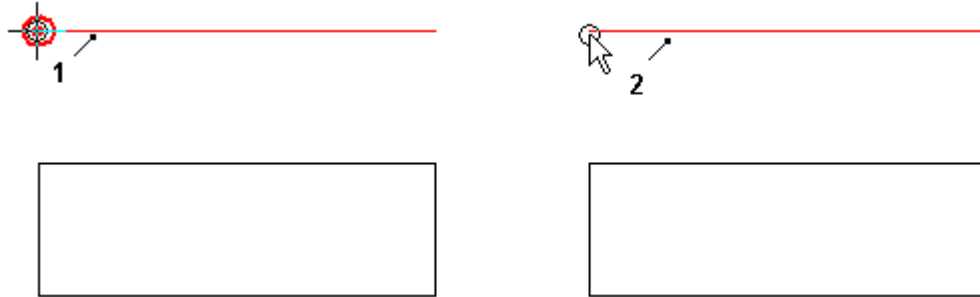
Move with PinPoint

PinPoint can be used with the **Select** tool to move an object a precise distance in x and y relative to a known position in the drawing.

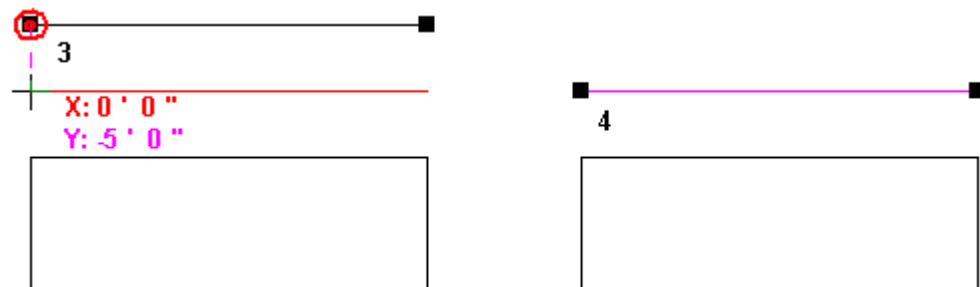
1. From the **Draw** toolbar, click the **Select Tool** 

2. From the **Main** toolbar, select **PinPoint** .

3. Place the **PinPoint** target point in your drawing. **(1)**



4. Position the **Select Tool** cursor over the object to be moved. **(2)** SmartSketch indicators can be used to locate keypoints on the object.
5. When SmartSketch indicates the keypoint to be used as the handle point for the move operation, begin dragging the object. PinPoint displays the distance between the target point and the reference point as you drag. **(3)**



6. Release the mouse button when the object is in the desired location. **(4)**
7. Click in a blank area of the drawing space, to deselect the graphic object.

SECTION 6

Complex Object Tools

This lesson covers the tools used to place complex graphic objects in a SmartSketch document.

Topics

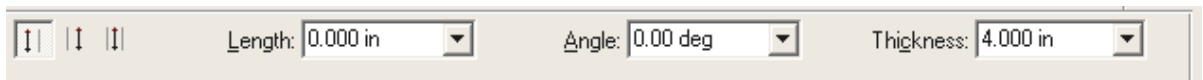
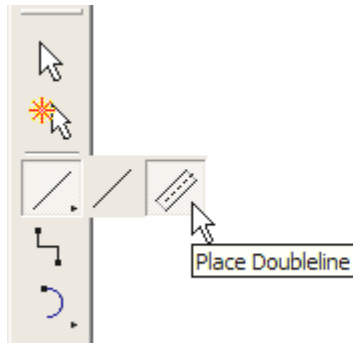
- Double Line
- Connectors
- Polygons
- Fillet
- Chamfer
- Isometric Drawing Tools

Double Line

Place Doubleline



Place Doubleline draws a doubleline or a series of connected doublelines. A doubleline is two lines drawn simultaneously, with a pre-defined gap between the lines. Corners are cleaned as you place the lines. This command can be very useful in drawing an architectural floor plan for a house or a factory, or anything represented by parallel lines. **Place Doubleline** also miter joints and trims as you draw. It puts endcaps on single lines.



Left Primary Line

Indicates that you are drawing the doubleline from the left side.



Right Primary Line

Indicates that you are drawing the doubleline from the right side.



Center Primary Line

Indicates that you are drawing the doubleline from the center.

Length

Sets the length of the primary line. This box accepts only positive values.

Angle

Sets the angle of a line. This box accepts positive or negative values. A positive value is counterclockwise from the x-axis, and a negative value is clockwise from the x-axis.

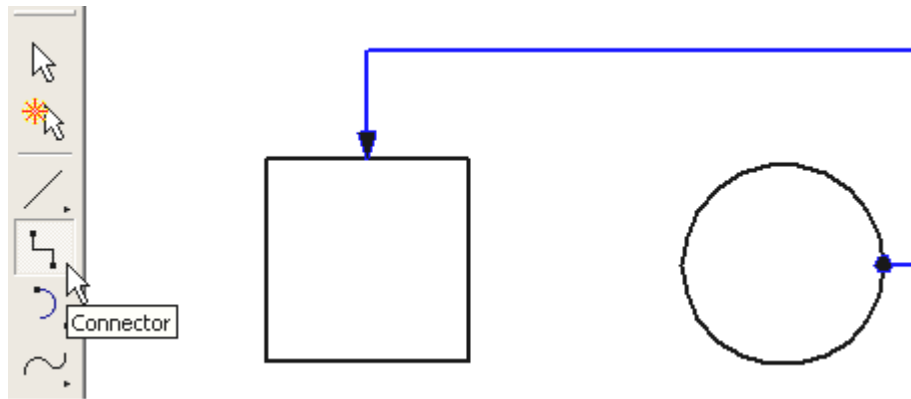
Thickness

Determines the distance or gap between the two lines which make up the doubleline.

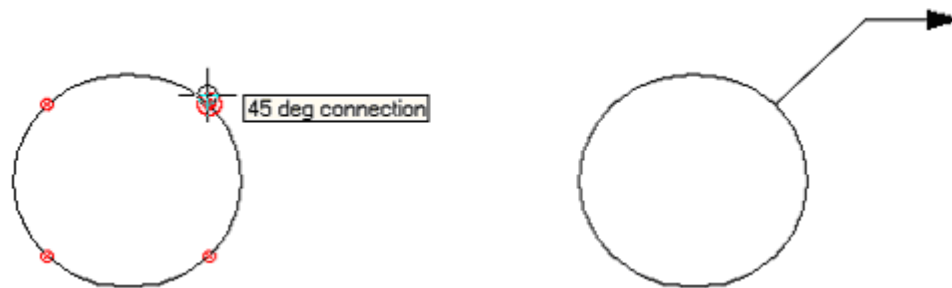
Connectors



The **Connectors** command uses intelligent routing to maintain connections between objects and symbols. A connector will connect to native graphic objects such as lines and circles as well as complex objects like symbols and groups.

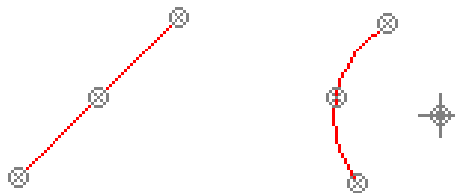


Symbols can be pre-defined with specific connect points which are recognized by the **Connector** command. Connect points on symbols can display tool tips and have predefined angles.



Attaching Connectors

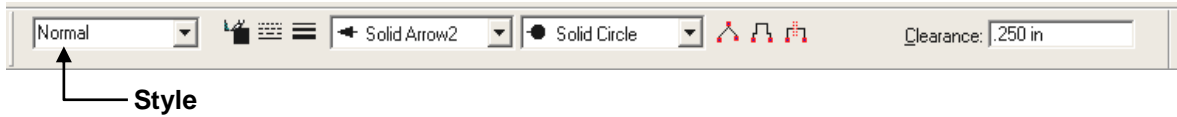
- **Key points:** Hollow gray circles with Xs indicate potential key points (such as endpoint, midpoint, and others) on an object, when the object has been located by the cursor.



- **Connect points:** Hollow red circles with Xs indicate predefined connect points on a symbol.
- **Any point on an object:** A solid red filled circle is displayed when the pointer locates an object at a point other than its key points.

Connector Ribbon Bar

Most of the options on the **Connector** ribbon bar are the same whether you are placing or modifying a connector. The only difference will be when you select a connector for modification. When selecting existing connectors, in addition to **Diagonal Mode**, the tools to **Insert Segment** or **Split Connector** are available.



Style

Sets the line style for the connector.



Line Start Terminator

Sets the terminator at the starting end of the connector.



Line End Terminator

Sets the terminator at the end of the connector.

Clearance

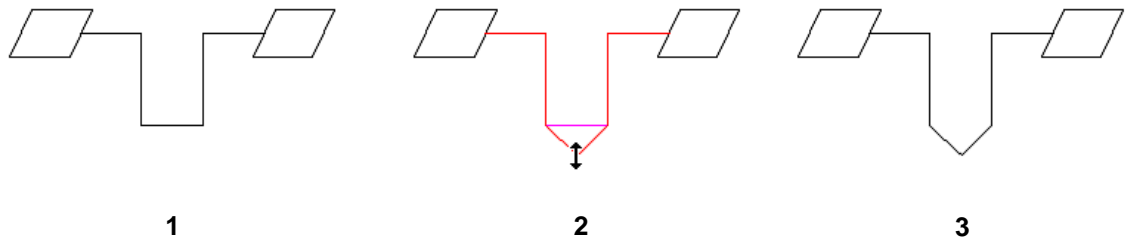
Specifies the amount of space that is maintained between the connector and the connected object. (Also known as range avoidance.)



Insert/Move Vertex Mode

At placement time, switches the connector placement between drawing horizontal and vertical line segments or drawing **Diagonal Mode** line segments.

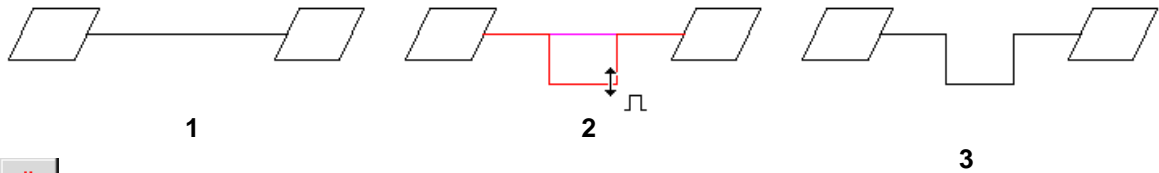
When modifying a connector the command will insert a vertex to a line segment of the connector for diagonal placement. Depressing the **Alt** key while placing a connector can also activate this option.





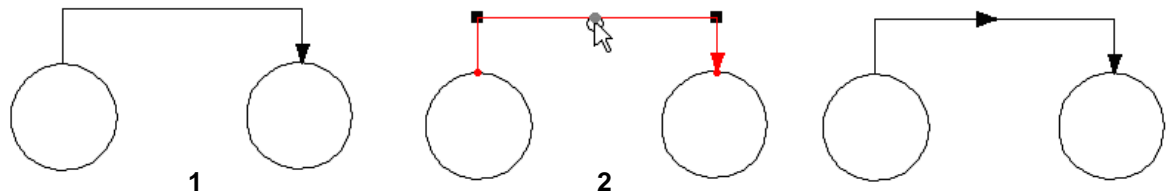
Insert Segment Mode

Inserts line segments into the connector. This option is available only when you are modifying a connector. Selecting a connector and pressing the **Alt+S** keys can also activate this option.



Split Connector

Splits a selected connector at a point defined by a mouse click. The **Split Connector** also allows the user to identify a symbol and split the connector based on two points on the symbol – effectively breaking the connector into two parts. Selecting a connector, holding down the **Shift** key and clicking on the connector at the desired location to split it in two can also activate this option.

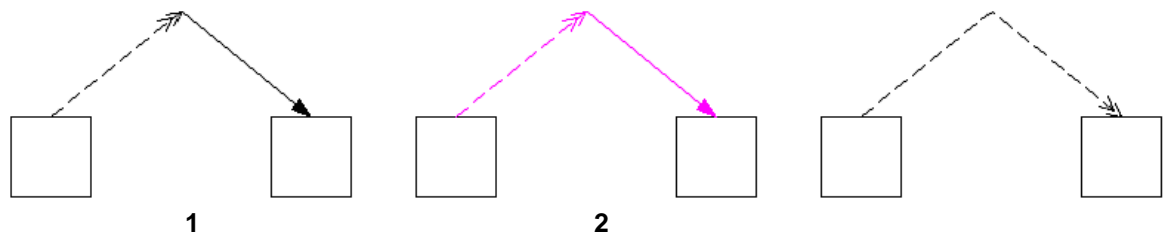


NOTE: The **Split Connector** should not be used to trim a single connector to the edge of a symbol. In this instance, the connector should be modified directly with its end handles.



Merge Connector

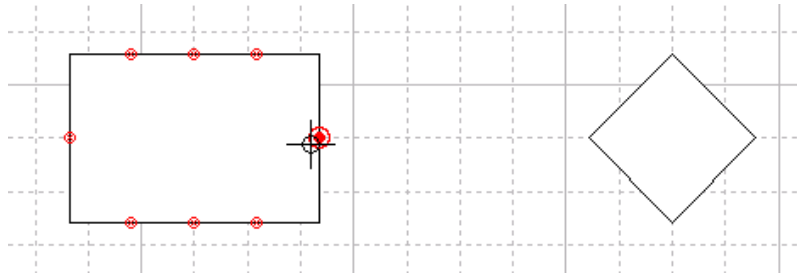
Takes two connectors and merges them to form a single connector. To activate the command, both connectors should be selected. The command will be disabled if only a single connector or more than two connectors are selected. The first connector selected drives the manner in which the merge occurs such that the properties of the first connector (e.g. Line Style, etc.) selected will be applied to the second connector.



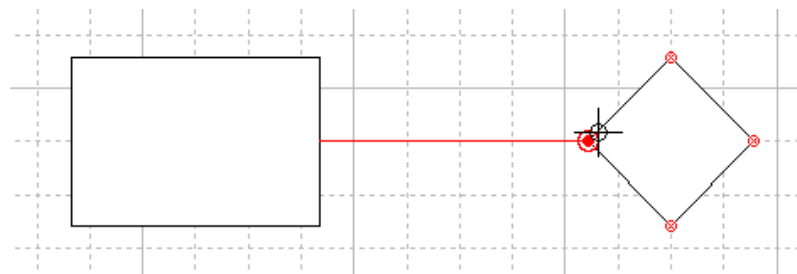
To Attach a Connector to an Object

1. Click to begin drawing the connector or move the pointer over desired object to highlight the potential key points or connect points.

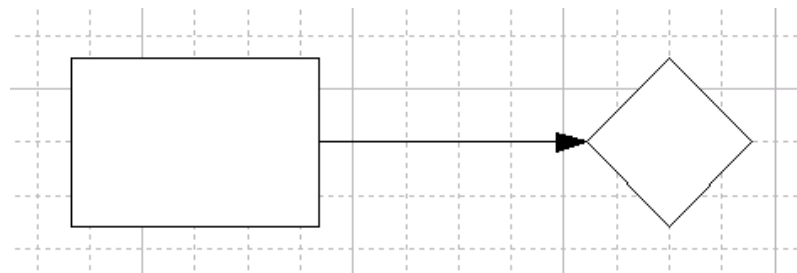
The predefined connect points appear as red circles with Xs, and keypoints appear as gray circles with Xs.



2. Click to start the connector at a predefined connect point, keypoint, or point along an object.
 - The direction in which you move from the start point determines the angle at which the connector is drawn.
 - If the object itself is rotated, the takeoff angle is adjusted appropriately so that the connector is perpendicular to the object.
 - If you press the **Alt** key while drawing a connector, the connector attaches to a connect point at any angle.
3. Pass the pointer over the second target object.



4. Click to end the connector at a pre-defined connect point, keypoint, or point along an object.

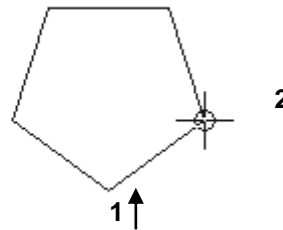
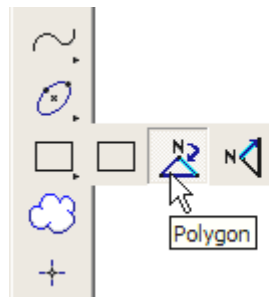


Polygons

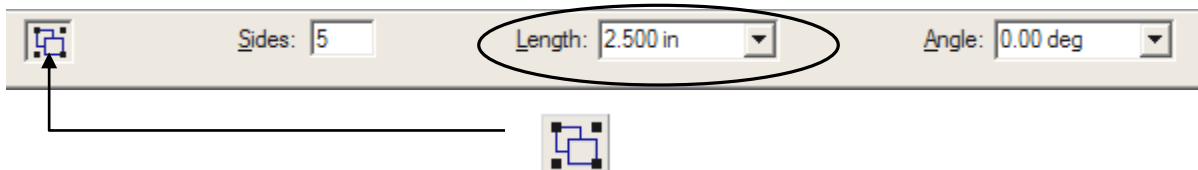
Polygon



The first point placed in the drawing will anchor a corner of the polygon. The second point will complete the placement of the polygon. As the distance between the first point and second point increases or decreases, the scale of the polygon will be affected because this tool constrains all edges to be equidistant. For best performance, do not create a polygon that has more than 25 sides.



By default, the polygon is placed as a grouped object. To break up the polygon into individual components upon placement, turn off **Group after Placement** on the **Polygon** ribbon bar. Values can be typed in the **Length** and **Angle** fields on the ribbon bar.

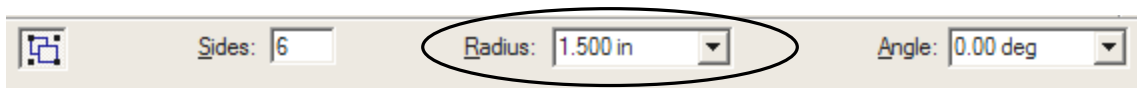
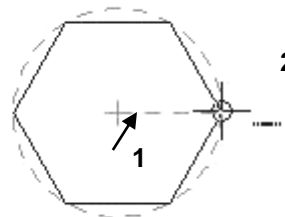
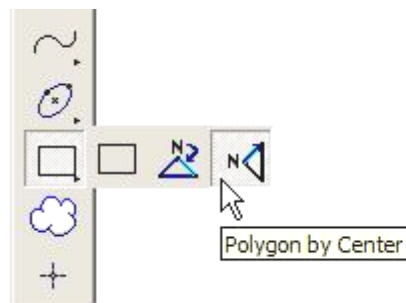


Group after Placement

Polygon by Center



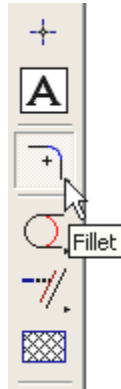
Draws a polygon by first placing a center point and a second point to define the radius. Values can be typed in the **Radius** and **Angle** fields on the ribbon bar.



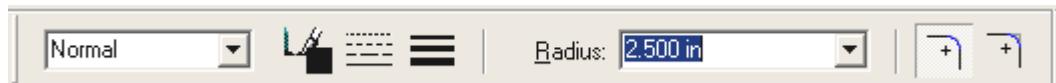
Fillet



The **Fillet** command is used to construct a fillet between two geometric objects. The objects can be arcs, lines, circles, ellipses, or curves.



Values can be typed into the **Radius** field to create a precise fillet. Whether you use the “drag” method, the “click” method or the “key-in method” make sure that the appropriate trim setting is selected. The **No Trim** toggle will prevent the lines from being truncated when a fillet is placed into the existing line work, while **Trim** will truncate the lines.



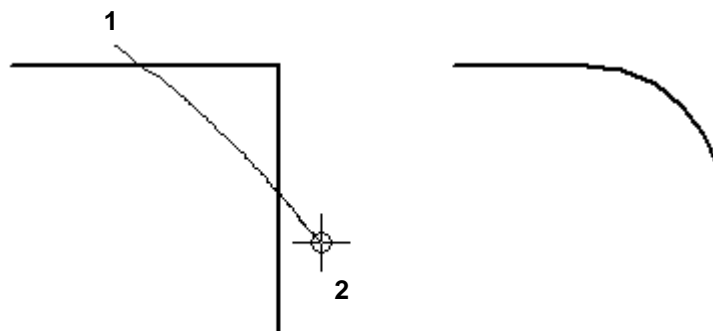
Trim



No Trim

A fillet can also be drawn at a corner with one click. Position the cursor over a corner, then press the left-mouse button. The value in the **Radius** box is active until a new value is entered, so one corner after another can be selected to draw fillets with the same radius.

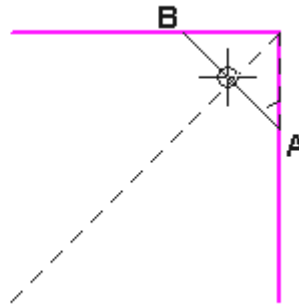
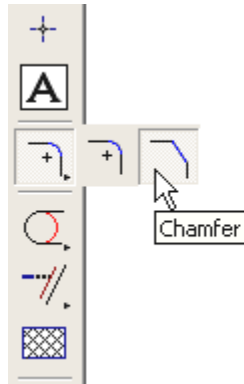
You can also draw fillets by dragging the cursor over the two objects that you want to draw the fillet between.



Chamfer



The **Chamfer** command draws a chamfer, or bevel, between two linear objects. You can control the chamfer angle and the setback distances for both objects. As with the **Fillet** command, a chamfer may be created using one of the three basic drawing methods: drag, click or key-in.

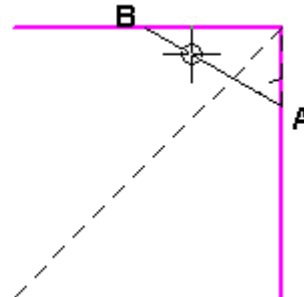
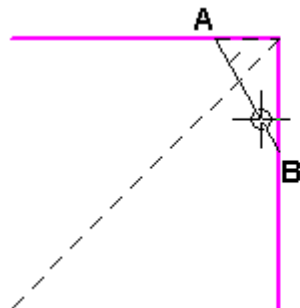


A chamfer can be drawn only between linear objects. For a precise chamfer, values can be typed into the **Set back a:** and/or **Set back b:** fields. **Set back a:** determines the distance that the first linear object will be set back from the corner. **Set back b:** will determine the setback distance for the second linear object. The **Angle** field will set the angle of the chamfer.

Angle: 45.00 deg	Setback a: 1.000 in	Setback b: 1.000 in
------------------	---------------------	---------------------

A chamfer can be drawn with equal chamfer angles. Select the two linear objects to use. When the chamfer is displayed dynamically, move the cursor until the desired setback distance is displayed, then click.

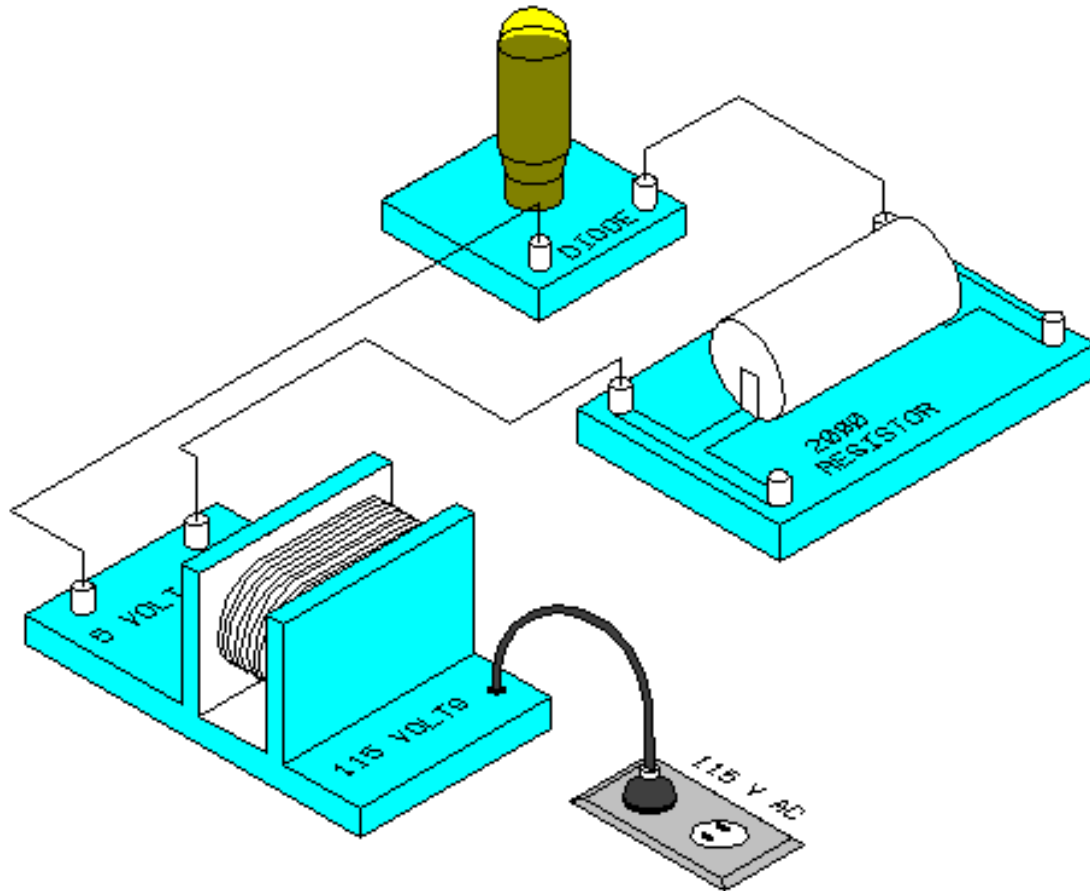
The setback values for the linear objects can be switched. After selecting the linear objects to use, move the cursor to either side of the temporary line display, then click.



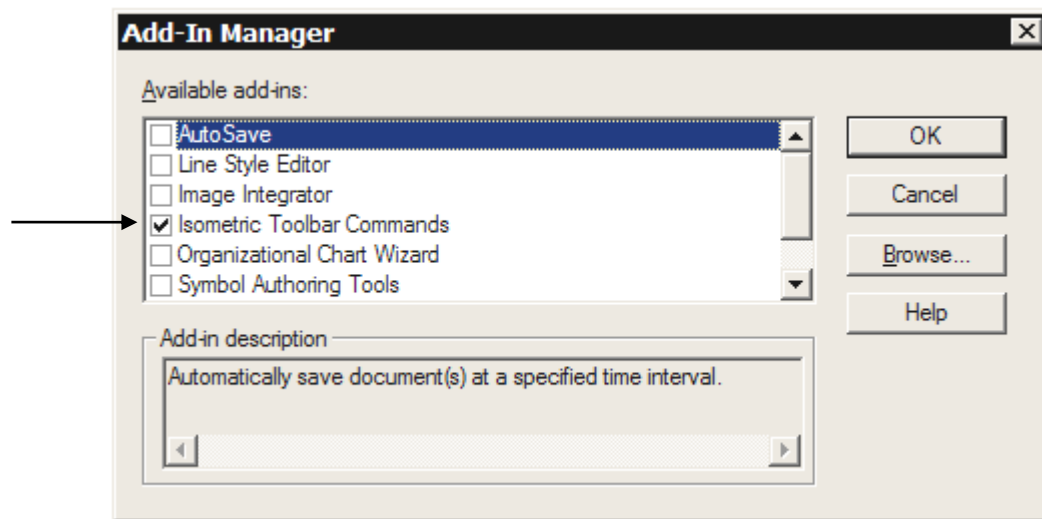
You can also dynamically draw chamfers by dragging the cursor over the two objects that you want to draw the chamfer between. When using this method, the edit boxes on the ribbon bar are not active.

Isometric Drawing Tools

SmartSketch provides tools to create 2D drawings that represent 3D objects, such as cubes in an isometric drawing. An isometric drawing is not a true 3D drawing, because you cannot view the drawing in perspective or from another angle. However, you can create a 3D effect by drawing lines, circles, and rectangles at the 30, 60 or 90-degree axes for isometric layouts. This functionality allows for accurate isometric drawings, schematics, and technical illustrations.



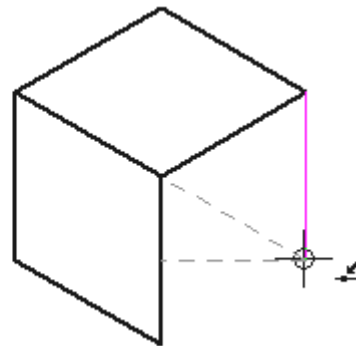
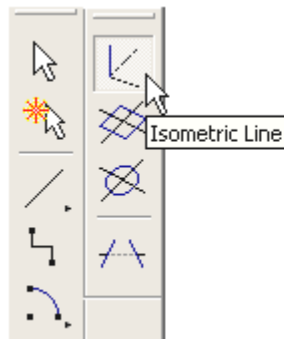
To access the isometric tools, open the **Add-In Manager** from **Tools> Add-Ins**. Select the **Isometric Toolbar Commands** option to open the **Isometric Tools** tool bar.



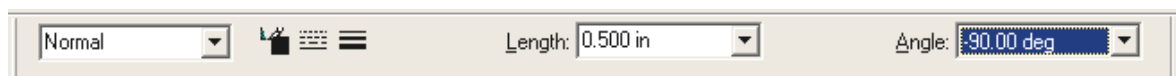
Isometric Line



The **Isometric line** command allows you to draw lines that appear at 30, 60, or 90 degrees in an isometric drawing. This tool creates 2D lines that look like they are in a 3D plane.



The “drag” method does not work when using commands on the **Isometric Toolbar**. You can draw objects using the “click” placement method or by entering values in the ribbon bar. Unlike the **Line / Arc Continuous** command, the **Angle** values cannot be changed, the only options are 0, 30, 90, 150. Negative angle values are also available in the option menu.

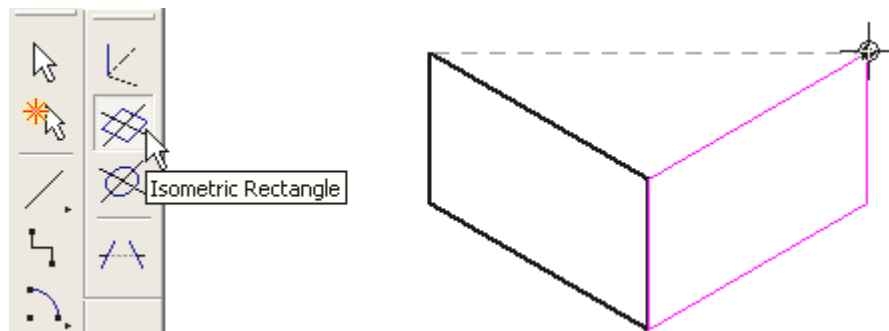


Isometric Rectangle



The **Isometric Rectangle** command allows you to draw an isometric rectangle. This tool can be used to create 2D rectangles that look like 3D rectangles. The first click defines a “start” point, the second click determines the length of the rectangle base and the third click will determine the opposing edge, completing the rectangle. Values can be typed in the **Width** and **Height** fields on the ribbon bar.

If **Maintain Relationships** is not toggled on when using **Isometric Rectangle**, then the rectangle will be comprised of four separate lines that do not stay connected when you move each line.



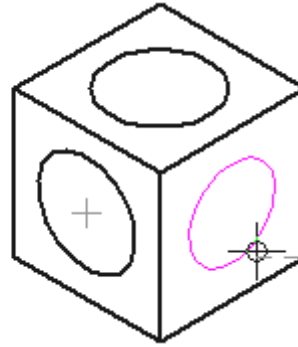
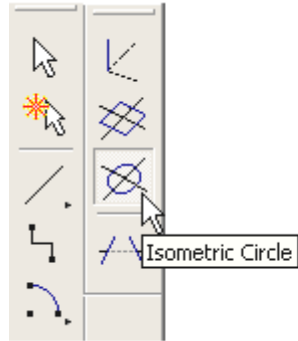
There are three different planes (axes) available when drawing an isometric rectangle or circle: **Top**, **Left Side** and **Right Side**. The plane should be selected before drawing the isometric object.



Isometric Circle



This command allows you to draw an isometric circle. This tool can be used to create 2D circles that look like 3D circles. The first point will define a center or anchor for the circle, and the second point will determine the radius. Values can be typed in the **Radius or Diameter** fields on the ribbon bar.



Segmented Style



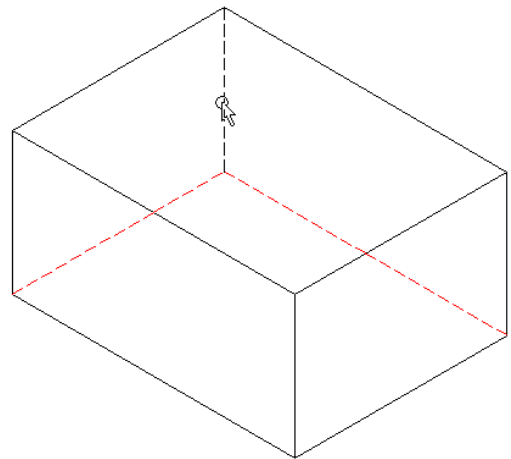
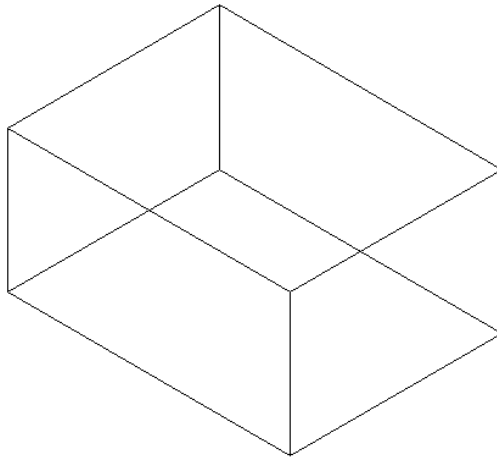
The **Segmented Style** command applies a line type, line width, or color to part of an isometric object. When you apply the line type, line width, or color, the format extends to the next object that crosses or bounds the object to which you are applying the line type, width or color.

This is handy if you want to change the appearance of just part of an object.

To Apply a Style to Part of an Object



1. Click the **Segmented Style** button.
2. On the ribbon bar, click the desired style, line type, line width, and/or color to apply to part of an object.
3. Select the object to apply the style to. The format extends to the next object that crosses or bounds the object to which you are applying the line type, width, or color.



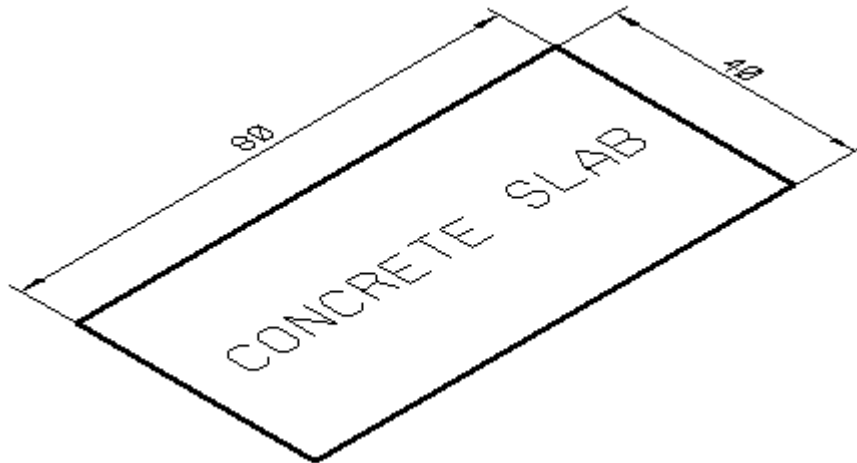
3. **Notes:** The **Segmented Style** command does not recognize symbols as a bounding object.
4. If you change the size of the object, you must re-apply the line type, width or color, with the **Segmented Style** command.

Dimensions and Text in Isometric Drawings

You can also access dimension styles that you configure to appear at an angle for isometric drawings. These styles are available for any type of dimension, except chained dimensions and angular dimensions. A document can be created based on one of the Technical Drawing templates and access these dimension styles.

Some of the dimension styles available are:

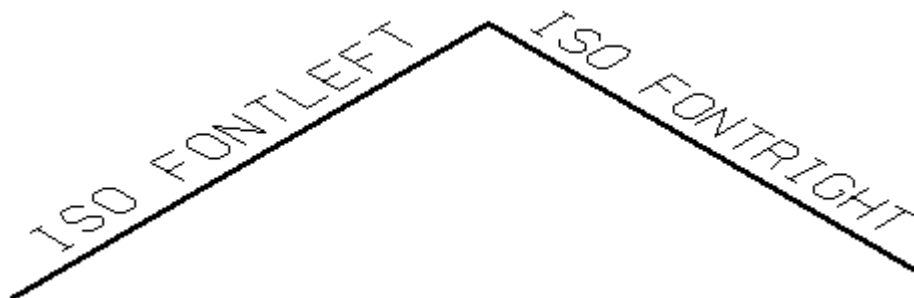
- ANSI ANSI (LEFT) ANSI (RIGHT)
- ISO ISO (LEFT) ISO (RIGHT)



Special fonts for isometric drawings are also available in some templates. These fonts are displayed at an angle.

A couple of the fonts available are:

- ISO_FONTLEFT
- ISO_FONTRIGHT



Double-clicking an object while using the **Select Tool** invokes the **Text Label** command, which rotates the text to align with the object or object to which it is associated.

SECTION 7

Selecting and Finding Objects

Discuss the various methods SmartSketch employs to locate existing objects in a drawing.

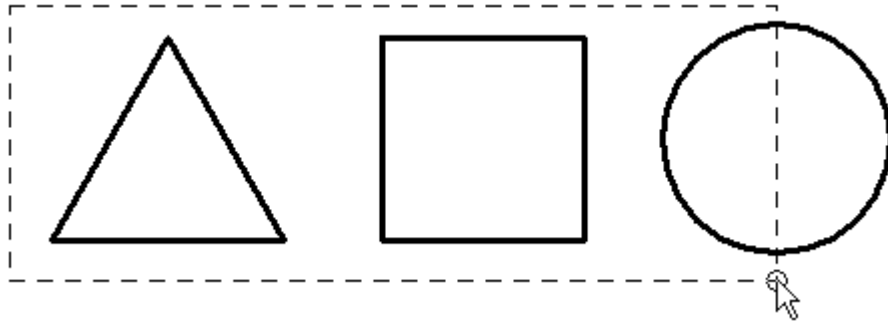
Topics

- Select Tool
- SmartSelect
- PickQuick
- Find

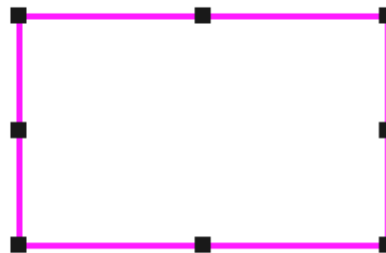
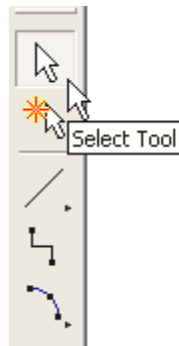
Select Tool



The **Select** tool can be used to select a single object, a group of objects, or all objects that lie within a specified area. Creates a select set area, or “fence,” by drawing a rectangle around points that you define. When you drag the mouse to define a rectangular fence, a dashed rectangular outline dynamically appears as you drag. When you reach the desired size, release the mouse button to create the fence. The dashed fence outline disappears and the objects are selected.



When an individual object is selected, the object changes to the selection color and the object's handles are displayed. Handles are solid squares at significant geometry positions on a selected object, such as end points and center points. Handles allow you to directly modify an object by dragging a handle to change the object's shape. When you select multiple objects or grouped objects, they change to the selection color.



Use one of the following methods to clear a selection:

- Click in free space
- Press the right-mouse button in free space.
- Select another object without holding the **Shift** key.

NOTE: The **Shift** or **Ctrl** key allows you to add or drop objects from a selection.

Select Tool Ribbon Bar

The ribbon bar contains settings that allow the user to determine if manipulation commands, like delete, move, copy, and rotate, act upon all objects in the selection set.



Inside

This command is toggled on by default. Only objects that lie entirely inside the fence are added to the selection set.



Overlapping

Objects that lie inside and objects that overlap any portion of the fence are added to the selection set.



Top Down

This command is toggled on by default. Specifies that groups of objects are located as opposed to individual objects in a group.



Bottom Up

Specifies that individual objects in a group are located as opposed to the whole group.



Expand

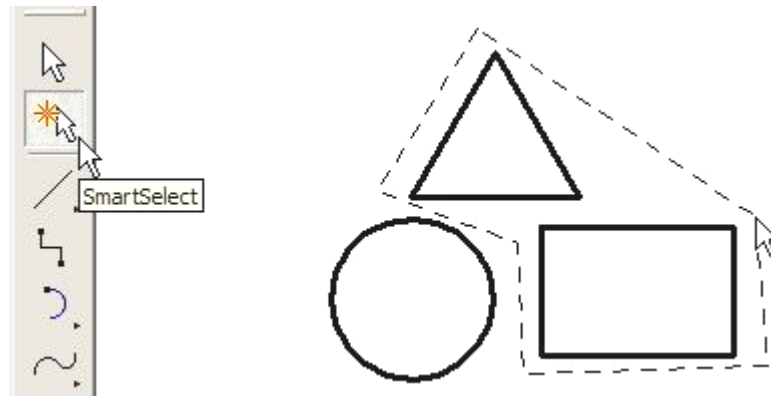
Displays another ribbon bar depending on the items that you selected on the drawing sheet. This button appears when you select ten or more items on the drawing sheet. When you click it, another ribbon bar appears.

NOTE: The default selection ribbon bar is displayed only when the **Select Tool** is active and no objects are selected. Once you select an object, the selection ribbon bar is replaced with a ribbon bar for editing the selected object.

SmartSelect



The **SmartSelect** tool creates a select set by drawing a rectangular or polygonal fence around objects based on points that you define. Objects will be selected when you press the right-mouse button to terminate that session with the command.



SmartSelect Ribbon Bar

The ribbon bar contains settings that allow the user to select a rectangular fence shape or user-defined polygon fence shape. In addition, the user may turn on/off a filter tool that can be used to locate specific types of drawing objects. Finally, the ribbon bar contains four commands that are also found on the **Select Tool** ribbon bar: **Inside**, **Overlapping**, **Top Down** and **Bottom Up**.



Rectangle

Creates a select set area, or “fence,” by drawing a rectangle around points that you define. When you drag the mouse to define a rectangular fence, a dashed rectangular outline dynamically appears as you drag. When you reach the desired size, release the mouse button to create the fence. The dashed fence outline disappears and the objects are selected.

Polygon

Creates a select set area, or “fence,” by drawing a polygon around points that you define. Following the first point, continue to click the left-mouse button to add additional points to define the scope of the polygon “fence”. Right-click to end the polygon. The dotted outline disappears and the objects are selected.

Chain Selection

Select other elements that are endpoint-connected to the selected element.

Turn Filter On/Off

Switches on or off filtering for drawing elements. When filtering is turned off, the **Locate Filter** command is disabled.

Locate Filter

Opens the **Define Filter Locate** dialog box where you can specify a filter for the selection of specific drawing objects. Filters allow you to select specific types of drawing objects, or all drawing objects.

Invert Selection

Remove selected elements from the select set while adding unselected elements to the select set.

PickQuick

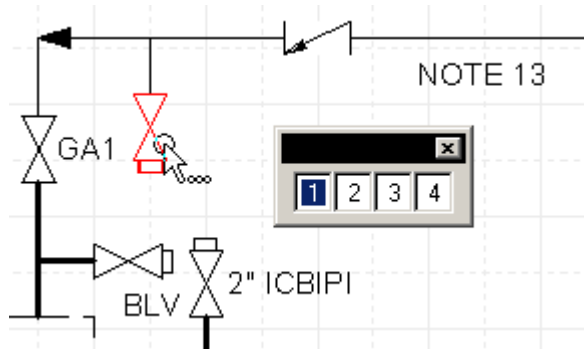
PickQuick aids in the selection of individual objects when there are several clustered very tightly together. If the cursor is placed where multiple objects overlap, three ellipses will appear at the lower right of the cursor. When the left mouse button is pressed, a numbered toolbar appears. As the cursor slides over a number, the corresponding object highlights. Picking that number selects the corresponding object.



PickQuick tool cursor

To select an object with PickQuick

1. Position the cursor over the objects to be selected from and pause the cursor there.
2. When the cursor changes to an ellipsis (three circles), click. The software displays the **Selection** toolbar near the cursor, with a button for each selectable object.



3. Move the cursor over the **PickQuick** buttons without clicking to highlight the corresponding objects. Click on the desired number to select the corresponding object.

Find

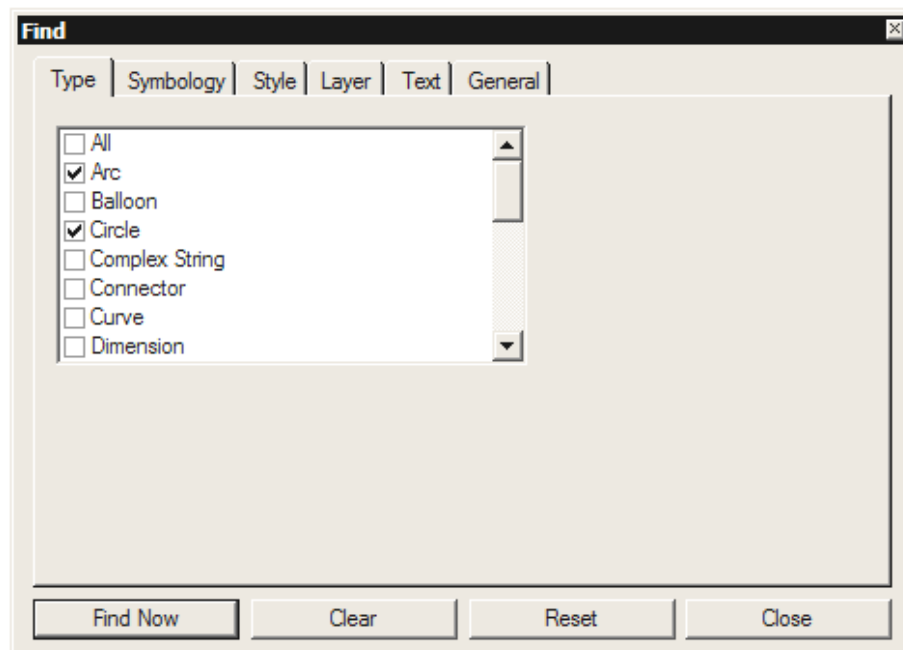
Searches for objects within a SmartSketch document based upon user-defined criteria. The **Find** command can be used to locate geometric objects and symbols that have a specific format or a specific set of defined attributes. Activated by selecting **Find** on the **Edit** menu or press **Ctrl + F** keys to execute the command. **Tools → Customize** can be used to place the **Find** button on a toolbar, the icon is located in the **Edit** category.

First, define the search criteria on the various tabs in the **Find** dialog box, and then execute the search. Items that match the defined criteria appear highlighted on the active **Drawing** sheet.

It is important to note that by default, the option **All** is selected on every tab, (with the exception of the **General** tab) in the **Find** dialog box. This setting may need to be disabled before executing the query.

- **Find Now** - executes the search based on the criteria specified in each tab.
- **Clear** - removes the results of the last search conducted. All items that appeared selected in the SmartSketch drawing are deselected.
- **Reset** - returns the options on each of the **Find** dialog box tabs to their default settings.

Type Tab

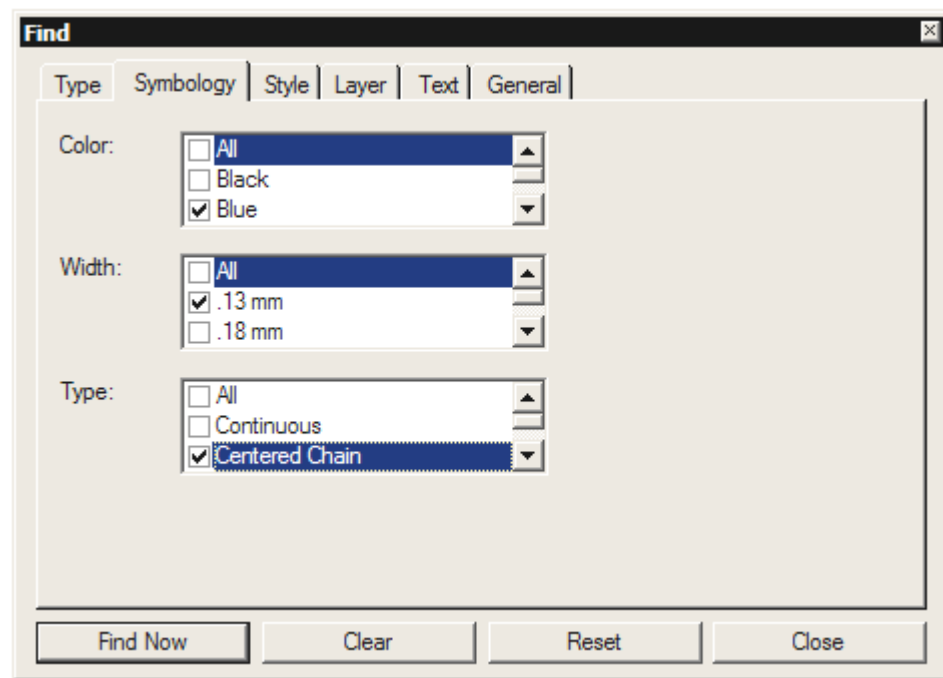


Type

Specifies the type of SmartSketch element, object, or attribute to be located. It is possible to define the parameters of the search to encompass a wide-range of items, including geometric objects, SmartSketch symbols, or a variety of attributes.

Symbology Tab

Sets options for performing a search based on color, line weight, and line type.



Color

Specifies the drawing color used in the search criteria.

Width

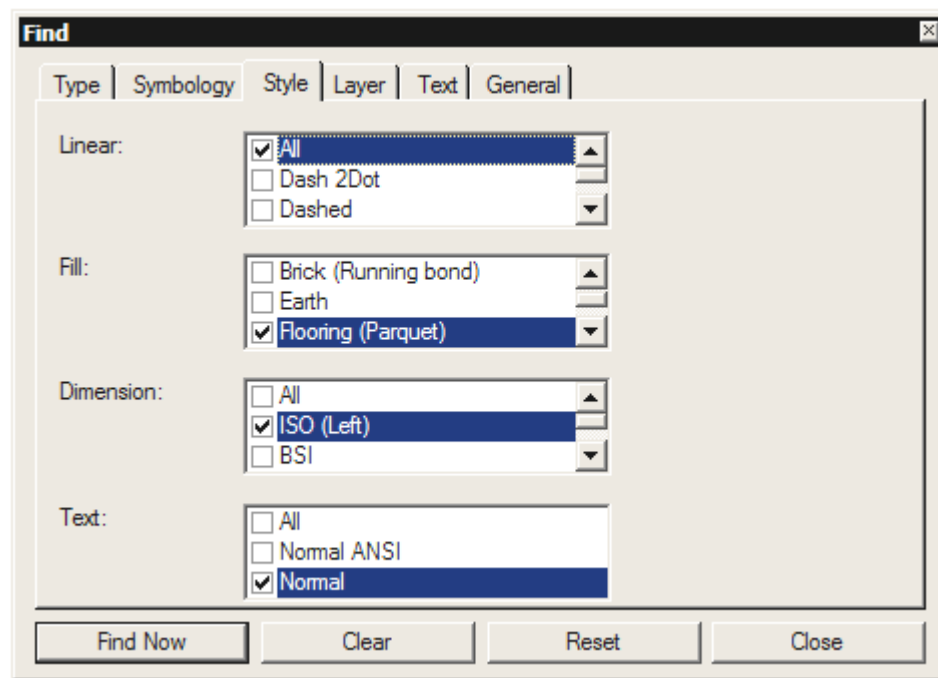
Defines the line width used in the search criteria.

Type

Specifies the drawing line type used in the search criteria.

Style Tab

Defines search criteria based on style attributes.



Linear

Specifies the linestyle used in the search criteria.

Fill

Specifies the type of fill used in the search criteria.

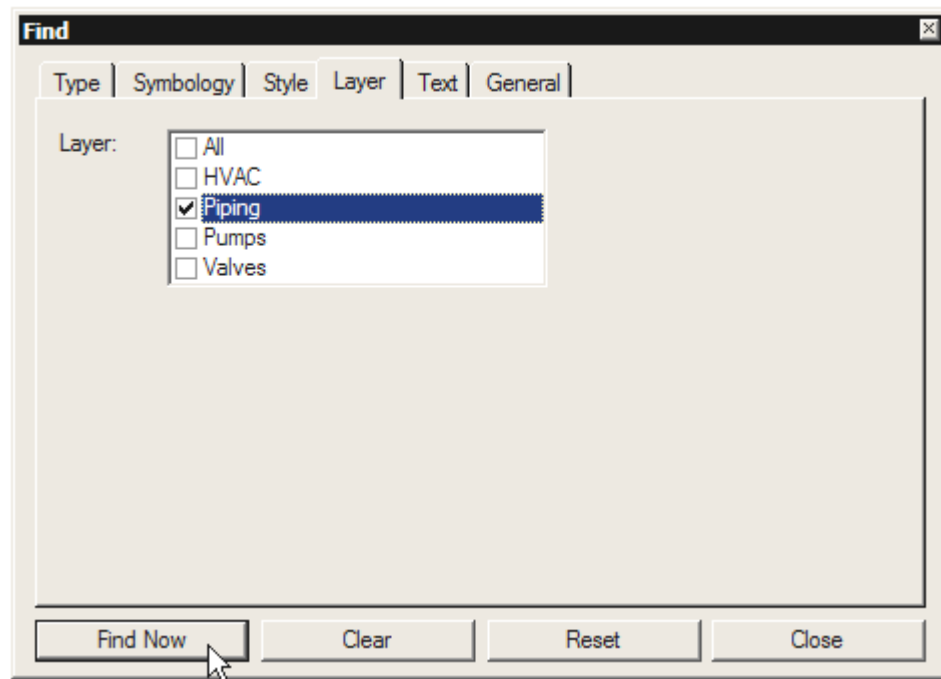
Dimension

Specifies the type of dimension used in the search criteria.

Text

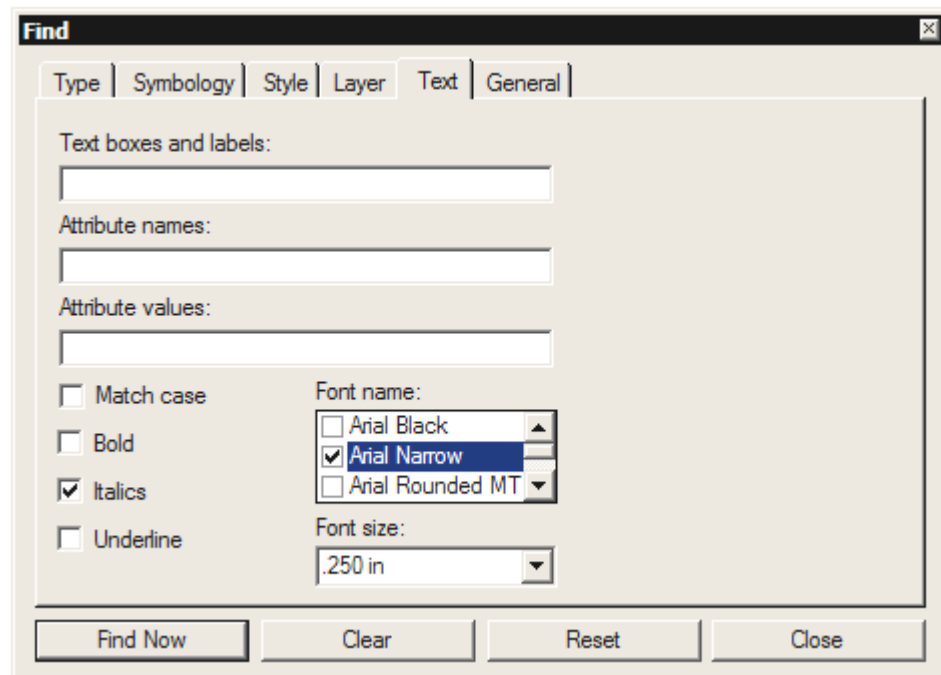
Specifies the type of text used in the search criteria.

Layer Tab



Specifies which layers of the SmartSketch document to conduct the search on. Two options are available by default - **All** layers or the **Default** layer. However, if additional layers have been created in the SmartSketch document, they are also displayed.

Text Tab



Text boxes and labels

Specifies the text string used in the search criteria.

Attribute names

Specifies the attribute name used in the search criteria.

Attribute values

Specifies the attribute value used in the search criteria.

Match case

Limits the search criteria to include only the text with the same capitalization as the **Text boxes and labels** text.

Bold, Italics, and Underline

Limits the search criteria to include only text that is bold, italicized, and /or underlined.

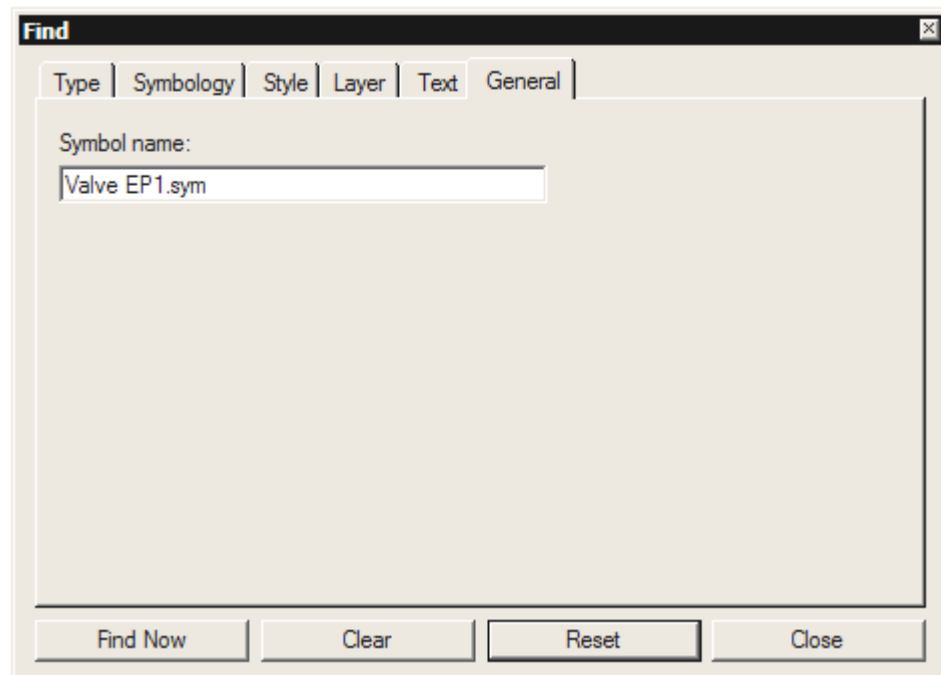
Font name

Limits the search criteria to include only text that matches the font name specified.

Font size

Limits the search criteria to include only text that matches the font size specified.

General Tab



Symbol name

Specifies the name of the symbol to be located when the search executes.

SECTION 8

Relationships

Discuss drawing with relationships and constraints.

Topics

- SmartSketch Drawing Glyphs
- SmartSketch Settings
- Intent and Tolerance Zone
- Applying a Relationship to Existing Geometry

Drawing with Relationships

When drawing, SmartSketch tracks the movement of the mouse cursor and shows a temporary, dynamic display of the element being drawn. This temporary display shows what the new element will look like when clicking at the current position.

SmartSketch gives information about the element being drawn by displaying relationships between the temporary dynamic element and the following:

- Other elements in the drawing
- Horizontal and vertical orientations
- The origin of the element you are drawing

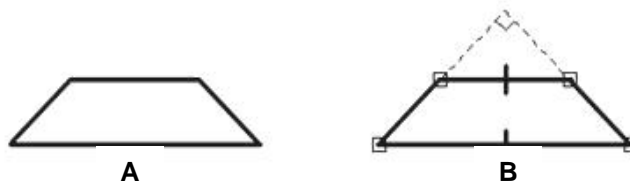
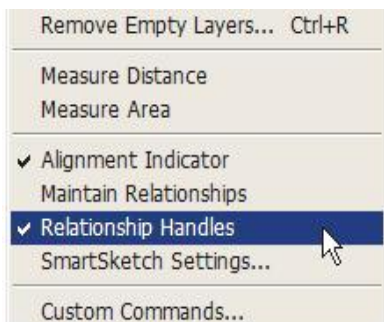
When SmartSketch recognizes a relationship, it displays a relationship indicator at the mouse cursor. As the mouse cursor is moved, SmartSketch will update the indicator to show new relationships. If a relationship indicator is displayed at the cursor when clicking to draw the object, the software can apply that relationship to the element (if **Maintain Relationships** is active). The relationship indicator displayed at the mouse cursor is called a “glyph”.

For example, if the **Horizontal** relationship indicator is displayed when you click to place the second end point of a line, then the line will be exactly horizontal.



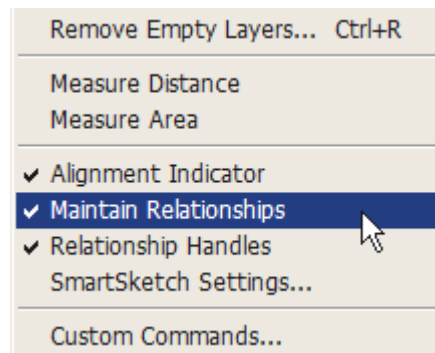
Relationship Handles

When applying maintained relationships, relationship handles appear on the related elements. Relationship handles are symbols that show how elements are related and can be removed by deleting the handles. The relationship handles can be displayed or hidden in drawings with the **Relationship Handles** option on the **Relationship** toolbar or by using the **Tools** menu. **A** is an example of relationship handles turned off, and **B** is an example of relationship handles toggled on.



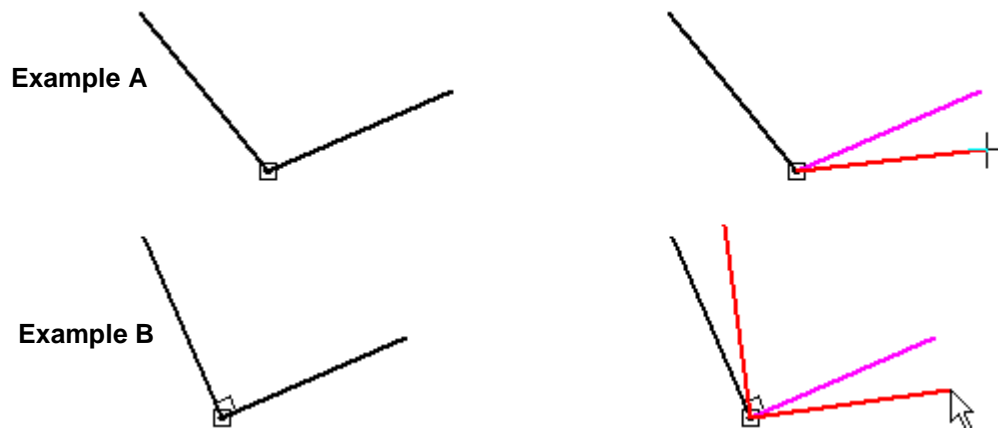
Maintain Relationships

If toggled on, the **Maintain Relationships** option on the **Tools** menu will maintain relationships as you draw. Subsequently, the relationships become part of the design and are maintained until removing them. If **Maintain Relationships** is set when using the commands on the **Relationship** toolbar to add relationships to the drawing, the added relationships will also be maintained.



Relationship Behavior

An element that has no maintained relationships applied to it can be changed in various ways. When there are no maintained relationships between two lines, each line can be moved and changed without affecting the other (**Example A**). However, if applying a perpendicular relationship between the two lines and moving one line, the other line will move with it (**Example B**).

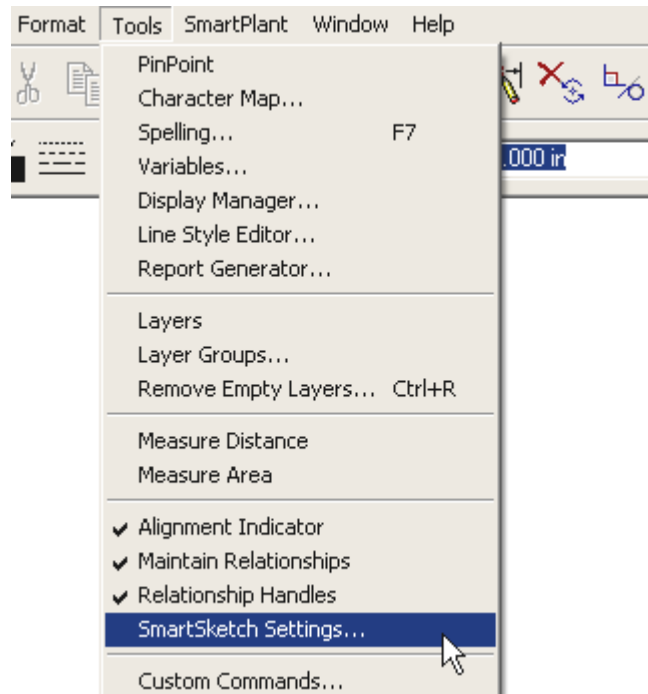


5. Notes:

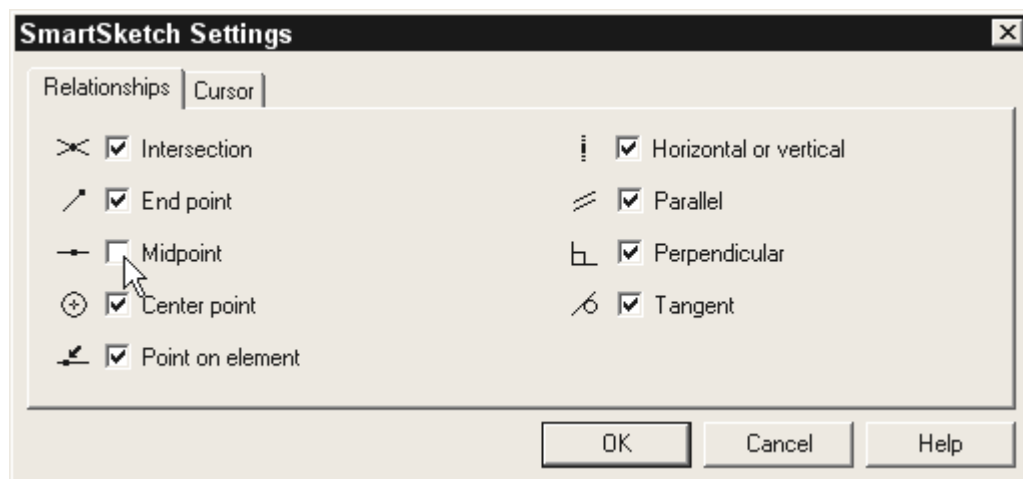
- When applying a relationship between elements, the relationship is maintained when modifying either element.
- If a line and an arc share a tangent relationship, they remain tangent when either element is modified.
- If a line and an arc share a connect relationship, they remain connected when either element is modified.

SmartSketch Settings

Another term for relationship indicators is SmartSketch Settings. Relationships can be applied as you draw by presetting the SmartSketch Settings dialog box, or you can go back and add relationships to objects after they have been drawn. The SmartSketch Settings dialog box is opened with the **SmartSketch Settings...** toggle on the **Tools** menu.



The SmartSketch dialog box customizes the operation of the software in recognizing and placing relationships. You can set pointer behavior and specify which relationships are recognized as you draw.



Relationships Tab



Intersection

Displays when you are at the intersection of two elements.



End point

Displays when you are at the end point of an element.



Midpoint

Displays when you are at the midpoint of an element.



Center point

Displays when you are at the center of a circle or an arc.



Point on element

Displays when you are at any point on an element.



Horizontal or Vertical

If the Horizontal relationship indicator displays when drawing a line, when clicking to place the second end point of the line, the line will be exactly horizontal. If the Vertical relationship indicator displays when drawing a line, when pressing to place the second end point of the line, it will be exactly vertical.



Parallel

Displays when a parallel relationship exists between two elements.



Perpendicular

Displays when a perpendicular relationship is recognized.

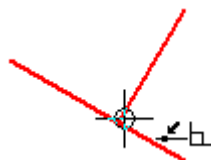


Tangent

Displays when a point of tangency is passed through.

NOTE: To suspend the display and application of relationship indicators, hold down the **<Alt>** key on the keyboard. To reactivate the relationship indicators release the **<Alt>** key.

SmartSketch can only recognize one or two relationships at a time. When SmartSketch recognizes two relationships, it displays both relationship indicators at the cursor.



Relationship Toolbar



The **Relationship** toolbar is activated from the **Main Menu**. You can use the commands on this toolbar to create relationships between two drawing elements.



When adding relationships with the **Relationship** toolbar, click a button on the toolbar and then select the objects that you want to relate. The first object selected will be modified to reflect the relationship.

Connect



Connects two elements or keypoints at one point. This command also connects a keypoint on an element to a position that is selected on another element.

Horizontal/Vertical



Makes a line horizontal or vertical. This command also horizontally or vertically aligns two keypoints.

Parallel



Makes two lines parallel.

Perpendicular



Makes two elements perpendicular. Two lines, a line and an arc, or a line and a circle can be made perpendicular.

Tangent



Makes two elements tangent. Two arcs or an arc and a line can be made tangent.

Colinear



Makes two lines colinear.

Concentric



Makes an arc or circle concentric with another arc or circle.

Symmetric



Makes elements symmetric about an axis. The characteristics of the elements on each side of the axis, such as size and position, are maintained by the symmetric relationship.

Equal



Makes elements or dimension values equal. The length of lines, the radius of arcs and circles, or the values of dimensions can be made equal.

Lock



Controls elements so they cannot be modified.

Relationship Handles



Displays or hides any relationship handles on elements.

Alignment Indicator



Displays or hides alignments as they are crossed over.

Maintain Relationships



Turns on and displays the relationship handles that indicate the designated relationship is being maintained as the user is drawing.

Apply a Relationship to Existing Geometry

By using the **Relationship** toolbar, you can go back to previously placed objects and add relationships or modify existing relationships to those elements. For example, the size of two circles can be made equal with an **Equal** relationship. Select the circle to be changed first then select the circle to be made equal to.



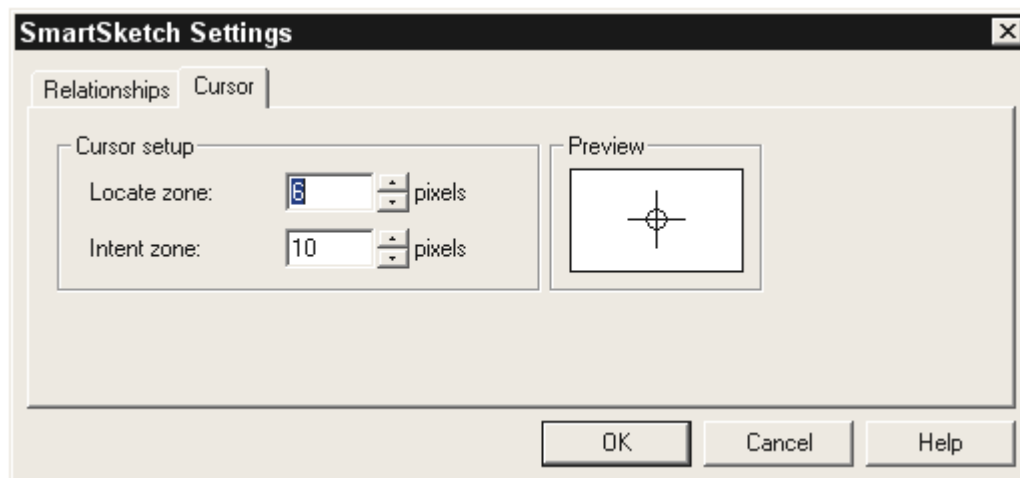
A relationship can also maintain a physical characteristic of an individual element. For example, a line can be made horizontal. The line remains horizontal even if its position and length are changed.



Maintain Relationships does not need to be toggled on to modify the appearance of objects in a drawing using the commands on the **Relationship** toolbar. The objects will modify, but they will not be constrained to maintain the applied relationship.

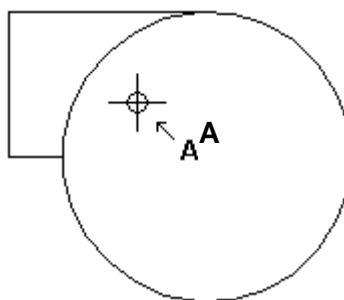
NOTE: Relationships are maintained between elements that are assigned to different layers on a drawing sheet. For example, suppose a line and a circle are tangent and they are assigned to different layers. If the line's layer is displayed and the circle's layer is hidden, you can still see the tangent relationship handle on the line. If the line's layer is hidden as well, the tangent relationship handle is hidden.

Cursor Settings

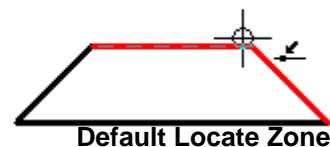
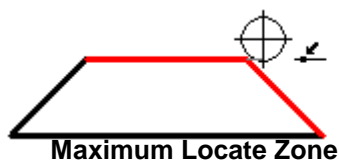


Locate Zone

The mouse cursor does not have to move to an exact position for SmartSketch to recognize a relationship. SmartSketch recognizes relationships for any element within the locate zone of the cursor. The circle around the cursor crosshair **A** or at the end of the cursor arrow indicates the locate zone.



A larger **Locate Zone** will increase the area that the cursor will need to determine relationships. In other words, a larger **Locate Zone** will make it possible for you to be farther away from an object when any applicable relationship indicators are displayed. This can also be disadvantageous when working in a drawing that has a great deal of graphic data.



NOTE: The default setting for the **Locate Zone** is 6 pixels, with a minimum setting of 3 pixels and a maximum setting of 12 pixels.

Intent Zone

Intent zones allow the arc and circle commands to interpret your intentions as you draw.

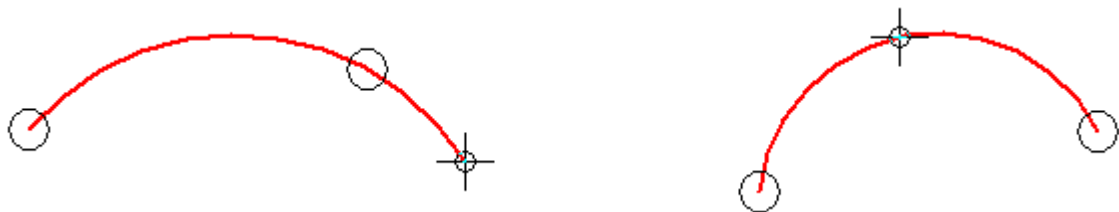
When selecting a starting point to begin drawing certain types of arcs, the software divides the region around the selected position into an intent zone. Moving the mouse cursor over one quadrant of the intent zone on the way to the next location can determine what the software will do next.

To change the active intent zone, move the mouse cursor back over this zone, and then emerge in a different quadrant (e.g. to create either a tangent or perpendicular arc).



NOTE: The default setting for the **Intent Zone** is 10 pixels, with a minimum setting of 3 pixels and a maximum setting of 12 pixels.

With some tools the **Intent Zone** is not quartered, but rather appears as a circle that is used to help you determine if the second point of an arc created using the **Arc by 3 Points** tool will be a point along the curve of the arc or whether it will be 2nd end point for the arc. The Intent Zone will also be displayed when you use the **Circle by 3 Points** command to help you determine the scope of the circle.



Finally, you can use the **Intent Zone** when drawing a line that is tangent or perpendicular to a curved object such as an arc.



SECTION 9

Manipulating Graphic Objects

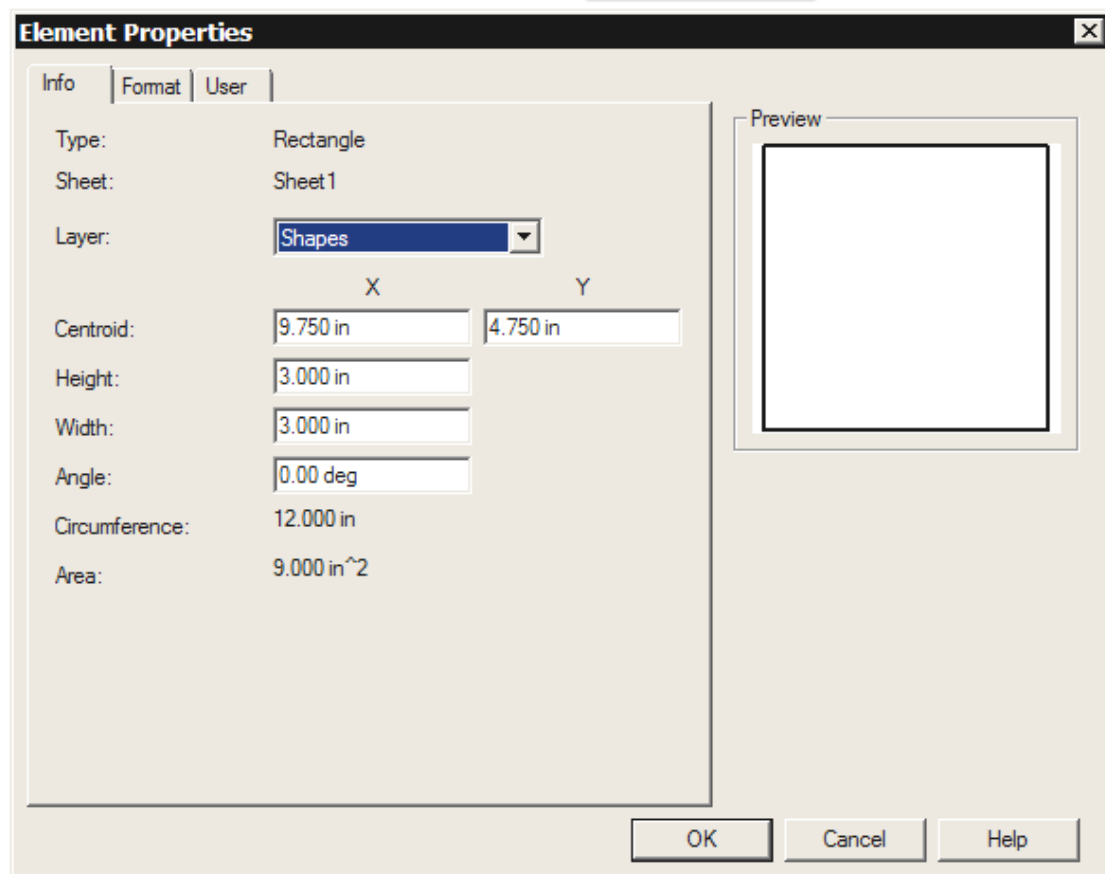
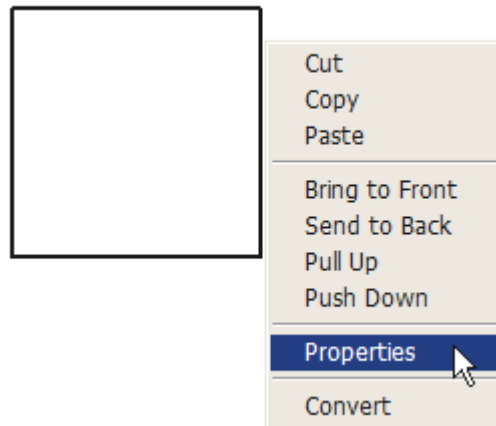
SmartSketch has a variety of tools available for the modification of graphic objects, once they have been added to your drawing.

Topics

- Element Properties Dialog Box
- Format Painter
- Move / Copy
- Nudge
- Offset and Associative Offset
- Rotate
- Mirror
- Align
- Scale
- Rectangular and Circular Patterns
- Trim and Extend tools
- Group and Ungroup

Element Properties

You can get a display of element properties by selecting the element with the **Select Tool**, right-click over the object, and then select **Properties** from the shortcut menu. **Element Properties** allows the user to change object components such as layer, style, color and width. The dialog box can also be used to add attribute data on the **User** tab, as well as review basic data about the object, or change the object's coordinate position, in the document on the **Info** tab.



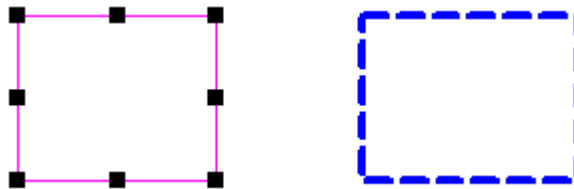
Format Painter



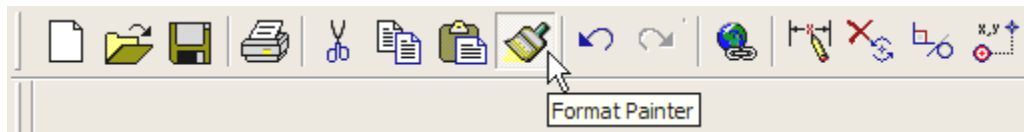
This tool copies properties such as line width, color, line type, font, text size, etc. from selected elements in the drawing or diagram, and applies them to other objects as they are selected. The two objects do not have to be similar. When copying a format between dissimilar elements, like text to geometry, or vice versa, only those common attribute(s) they have (e.g. color) will be applied. However, **Format Painter** will not re-format symbols.

To Use Format Painter

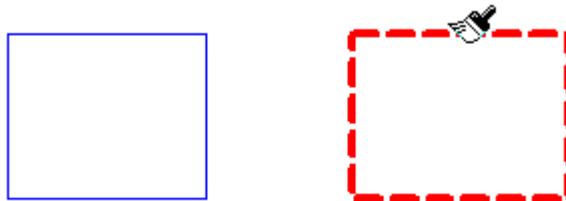
1. Select the object with the formatting you want to copy.



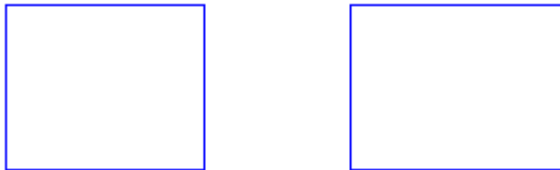
2. Activate the **Format Painter** tool from the **Main Menu**.



3. Select the object(s) to be modified.



4. The change should be reflected immediately. Right-click the mouse when finished.

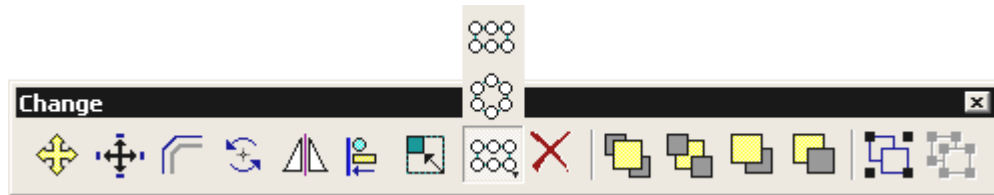


NOTE: If a connector with a line terminator is selected as the object from which to copy, the terminator will be added to any other linear geometry selected with the **Format Painter**. This is the only way to apply terminators to linear geometry that is not a connector.

Change Toolbar



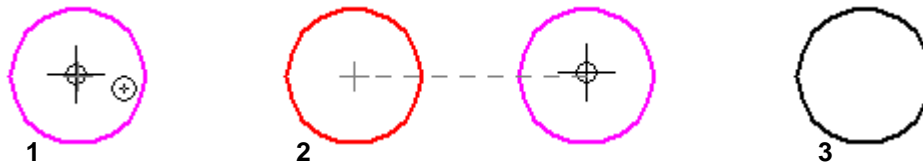
You can use the commands on this toolbar to manipulate objects. Clicking the **Change** command on the **Main Menu** will activate the toolbar.



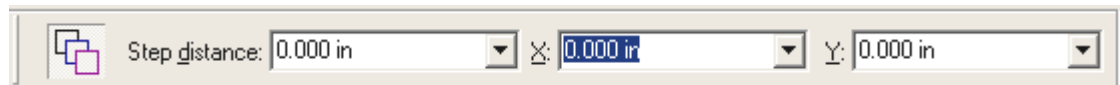
Move / Copy



Moves objects from one location to another. You can move one or more objects at a time, and you can move object groups. Simply select the object(s) to be moved or copied at the “**from**” point (after activating the **Move/Copy** tool), and click to place the object(s) at the “**to**” point.



You can specify the locations by clicking on the drawing sheet or by entering values in the ribbon bar. Objects can be copied by clicking the **Copy** button in the ribbon bar.



↑
Copy On/Off toggle

Step distance

Increments or decrements the value displayed in the **X:** and **Y:** input fields. For example, typing a step value of 0.25 and moving the pointer away from the “**from**” point along the x-axis would increment the distance from 0.25 to 0.5, 0.75, and so forth in the **X:** field.

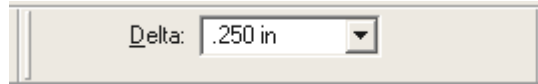
X: and Y:

Sets value(s) for the **X** and/ or **Y** coordinate. You can use each option by itself or with the other option.

Nudge



Moves an object or a select set in small increments. You use the **Delta:** box on the **Nudge** ribbon bar to define the size of the increment, and then use the arrow keys to bump or "nudge" the object or select set in the corresponding direction (left, right, up, and down).



Offset



Draws an offset copy of an element or a set of contiguous elements. This command copies elements while maintaining characteristics such as the angle of lines and the center point of arcs and circles.

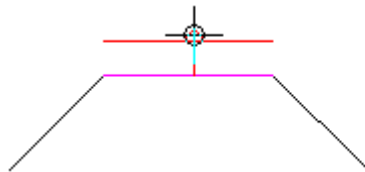
- Offsetting outside the perimeter of the original object creates a larger object.
- Offsetting inside the perimeter of the original object creates a smaller element.



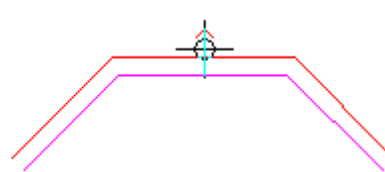
Select Chain On/Off toggle

Select Chain

- If it is not active, **Offset** copies only the selected object, (this is the default setting).
- If on, this option will offset a chain of all continuous objects.



Select Chain Off



Select Chain On

Step Distance

Sets the distance from the base object to the offset copy.

Cumulative Offset:

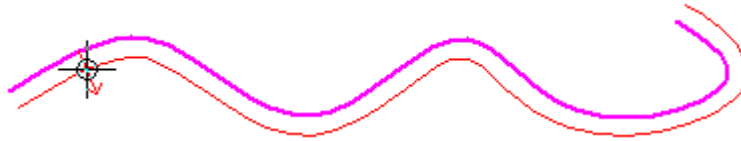
Displays or sets the total distance of the current offset graphic object from the original graphic object.

6. **Important:** The **Associative Offset** command is available *only* if you use **Customize** on the **Tools** menu to place it on a toolbar or menu. On the **Toolbars** tab of the **Customize** dialog box, select **Manipulation** category to display the **Associative Offset** button.

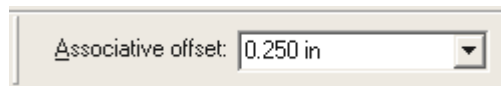
Associative Offset



Draws a copy of a curve and applies an offset relation. The **Associative Offset** tool copies the original curve geometry at a specified distance.



Offsetting outside the perimeter of the original curve creates a larger object. Offsetting inside the perimeter of the original curve creates a smaller object.



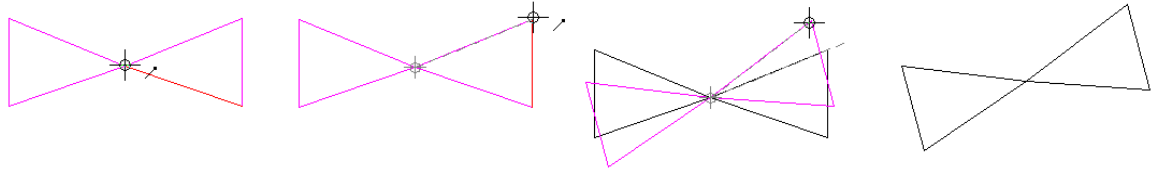
Associative offset:

Sets the distance from the base object to the offset copy.

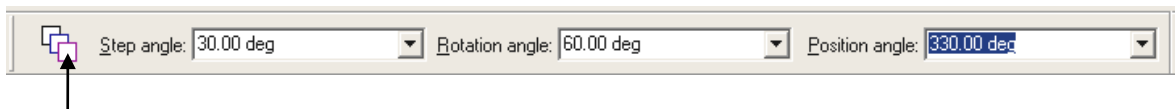
Rotate



The **Rotate** command rotates one or more objects a precise distance or angle about a specified point.



The first point will define the center of rotation. The second point will define the point you want to rotate from and the final point will complete the rotation at the angle you define either dynamically or through the values keyed in the ribbon bar.



Copy button

If toggled on, it will create a copy of the object(s) that you are rotating.

Step Angle

Specifies the rotation step angle. The step angle specifies the increments, in degrees, that an object rotates from a temporary axis. The temporary axis is the line between the object's center of rotation and the point that you rotate the element from.

You can enter increments in the **Step Angle** box if you want to dynamically view the results of the rotation. For example, if the box is set to 30.0, the rotation is displayed in 30-degree increments. The default setting is 0 degrees so that the rotation is fully dynamic.

Rotation Angle

Defines the rotation angle. The rotation angle is the angle between the point that you rotate from and the point that you rotate to.

Position Angle

Displays the angle between the horizontal axis through the center of rotation and the point that you rotate from. Then, if you rotate the object, the position angle changes to become the angle between the horizontal axis and the point that you rotate to. The position angle is always a positive value that is measured counterclockwise from the axis.

After rotating an object or elements, the **Rotate** command stays active. This allows selected objects to be rotated in new locations until choosing another command.

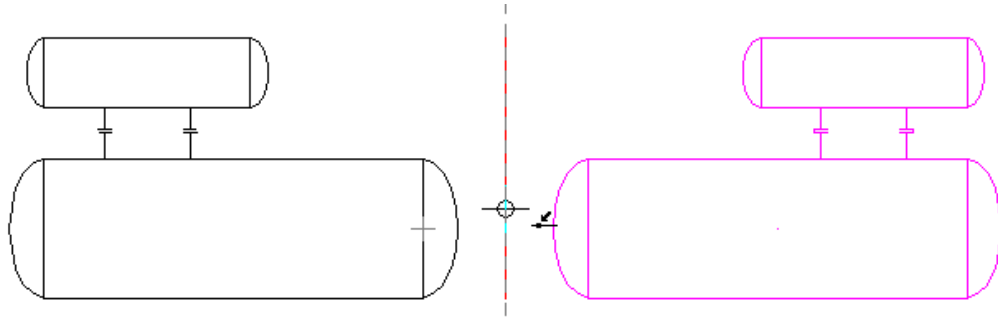
When using the **Rotation Angle** box or the **Position Angle** box and not specifying a from point, the from point is the center of the selected object.

Mirror



To mirror about a defined mirror axis, select one or more objects. Click in free space or press a keypoint with the left-mouse button. Move the cursor until the mirror axis and the mirrored objects are in the desired position, then click to complete the process.

Objects can be mirrored and 'mirror copied' by defining a mirror axis or by using a linear object as a mirror axis. The linear object does not have to be part of the selection set.

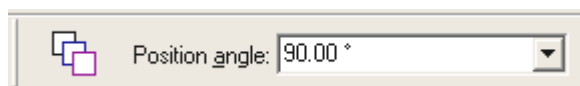
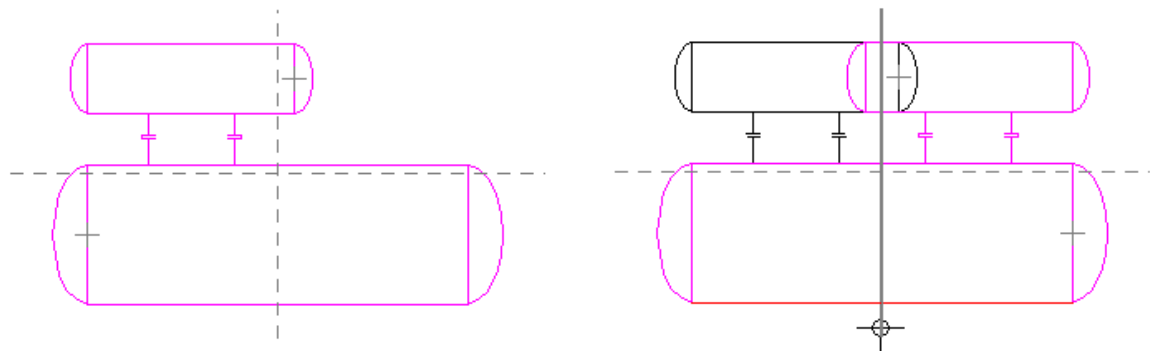


Position the cursor over the linear object that will be used as the mirror axis. The mirrored objects are displayed dynamically on the other side of the mirror axis. Position the mouse cursor so that the mirrored objects are where you want them, then click the left-mouse button. Press the right-mouse button to end the **Mirror** command.

View manipulation commands, such as **Zoom** and **Pan**, can be used with the **Mirror** command. When you finish using a view manipulation command, the software returns to the **Mirror** command at the point where the view command was chosen.

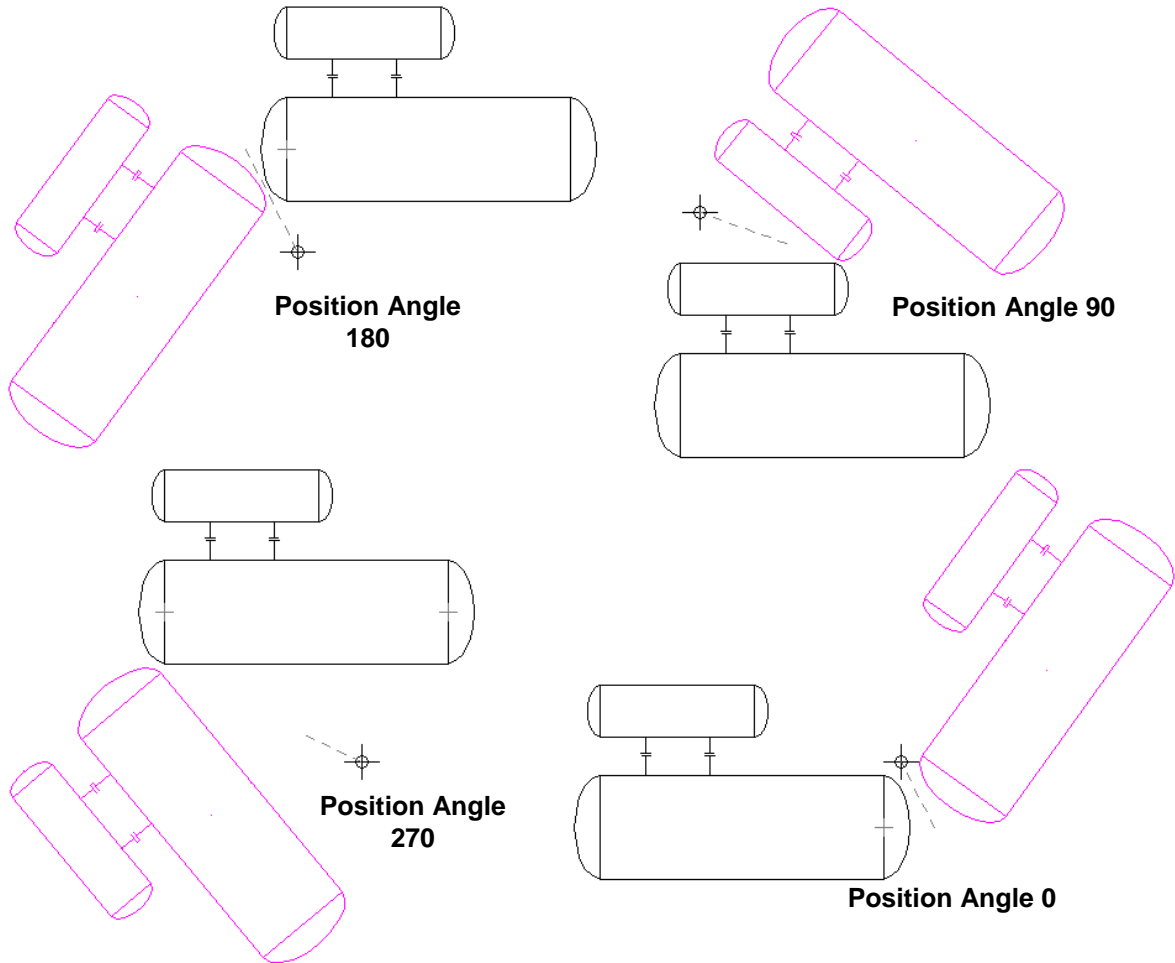
When mirroring and copying objects with the **Maintain Relationships** toggle set, relationships between objects within the select set are maintained if the relationships still apply after objects have been moved or copied.

The **Mirror** command can be selected before choosing the objects to mirror. The **Mirror** command allows you to flip objects in the select set. When the software displays axes through the center of the select set, choose a horizontal or vertical mirror axis. In the following example, the vertical axis was selected. The object essentially will flip without changing its coordinate position on the drawing sheet.



Position Angle

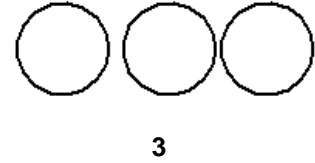
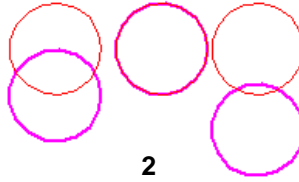
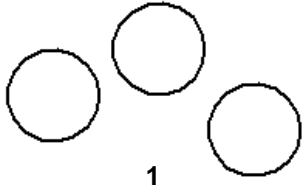
The **Mirror** ribbon bar sets the angle of the mirror axis. The origin of the angle measurement is the point you clicked for the beginning of the mirror axis. Setting the position angle to **0** extends the axis horizontally to the right of the screen, **90** extends the axis vertically to the top, **180** extends the axis horizontally to the left, and **270** extends the axis vertically to the bottom.



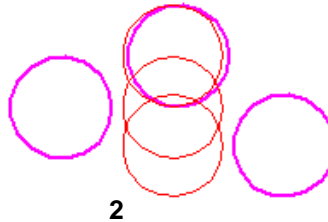
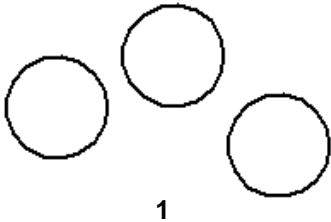
Align



Alignment involves lining up two or more objects vertically or horizontally, either by their edges or their centers.



Use care when aligning objects along the Vertical or Horizontal axis, you may unintentionally stack objects directly on top of each other.



This tool will align two or more objects in a select set along the edge or axis chosen in the ribbon bar. Alignment features only line up objects; they do not distribute an equal amount of space between aligned objects.



Align Left



Aligns the left side of objects in a select set.

Align Right



Aligns the right side of objects in a select set.

Align about Vertical



Aligns the objects in a select set along their vertical centers.

Align about Horizontal



Aligns the objects in a select set along their horizontal centers.

Align Top



Aligns the tops of objects in a select set.

Align Bottom

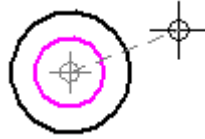


Aligns the bottoms of objects in a select set.

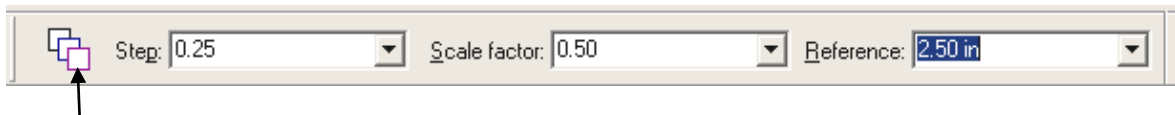
Scale



The **Scale** command reduces or enlarges selected objects by a scale factor that you define. The scale factor is the same along the x and y-axes. Scalable objects include framed objects, such as text boxes.



Click where you want the scale origin to be. The software displays a crosshair at the scale origin, and dynamically displays a line between the scale origin and the mouse cursor. Move the cursor until the objects are the desired size, then click the left button to finish the operation.



Copy button

If toggled on, it will create a copy of the object(s) that you are scaling.

Step

Specifies the step value for the **scale factor**. The step value causes the **scale factor** to decrease or increase in increments as you move the mouse cursor toward or away from the scale origin. For example, if you set the step value to 0.25, the scale increases in increments of 25% as you move the mouse cursor away from the scale origin.

Scale Factor

Specifies how much the software reduces or enlarges the element. A scale factor between zero and one reduces; a scale factor greater than one enlarges.

Reference

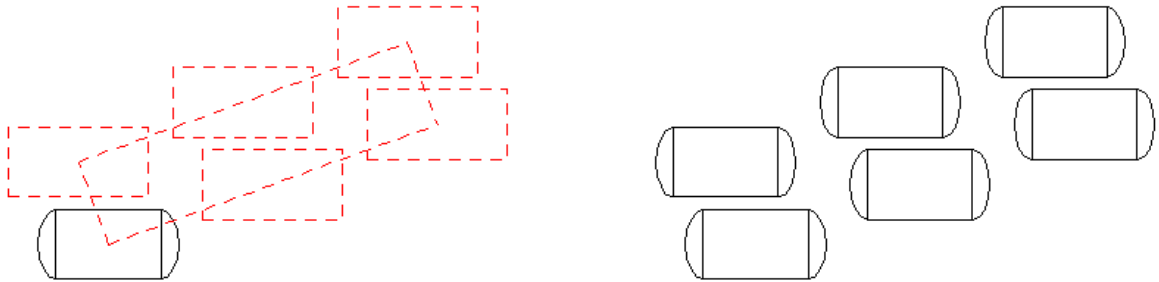
Specifies how long the dynamic line from the scale origin to the mouse cursor has to be in order to achieve a scale factor of 1.

For example, if you set **Reference** to 1, for every inch you move the mouse cursor away from the scale origin, the scale factor increases by one. If you set **Reference** to 2, for every two inches you move the mouse cursor away from the scale origin, the **scale factor** increases by one.

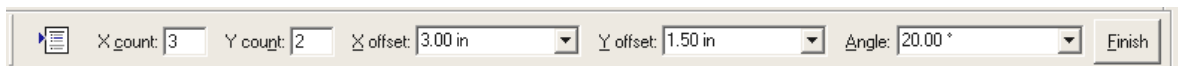
Rectangular Pattern



The **Rectangular Pattern** tool copies selected graphic objects in a rectangular pattern on the drawing sheet.



To create a rectangular pattern select one or more objects to pattern, set the values in the ribbon bar and click the **Finish** button on the same ribbon bar.



Rectangular Pattern Options (see next page)

X count

Sets the number of elements along the x-axis (columns) of the pattern.

Y count

Sets the number of elements along the y-axis (rows) of the pattern.

X offset

Sets the distance between adjacent rows in the pattern, measured along the x-axis.

Y offset

Sets the distance between adjacent rows in the pattern, measured along the y-axis.

Angle

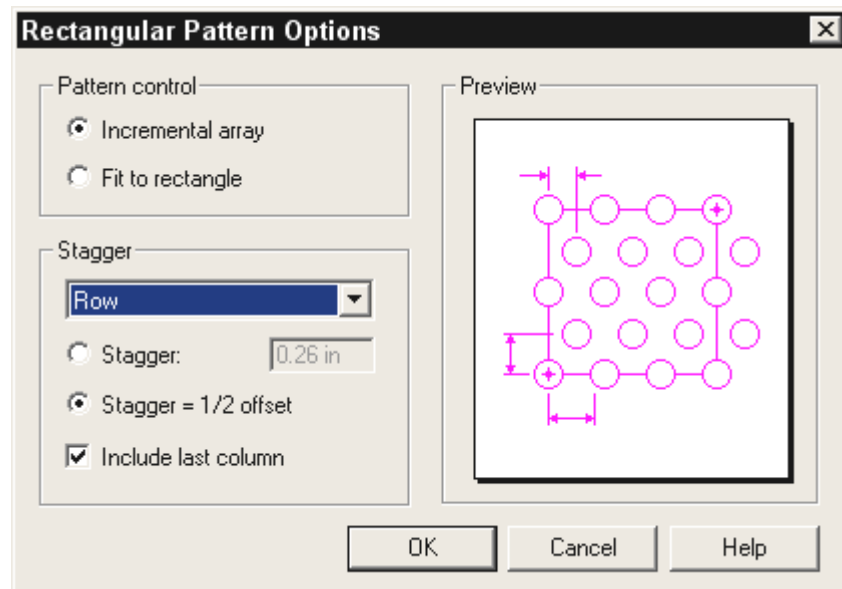
Sets the rotation angle of the pattern.

Finish

When you click this button, the pattern is created on the drawing sheet and you can select other objects.



Rectangular Pattern Options Dialog Box



Pattern Control

Specifies how the pattern is constructed.

Incremental array

Draws a pattern with a set offset between members.

Fit to rectangle

Draws a pattern by evenly spacing members along the x and y-axes of the pattern.

Stagger

Controls whether pattern members are arranged in a straight matrix, or whether every other row or column is offset from its default position. Options are **None**, for a straight matrix, **Rows**, to offset alternate pattern rows, and **Columns**, to offset alternate pattern columns.

Stagger

Sets the row or column stagger distance to the specified distance.

Stagger = 1/2 offset

Sets the row or column stagger distance to half the **X** offset or **Y** offset value.

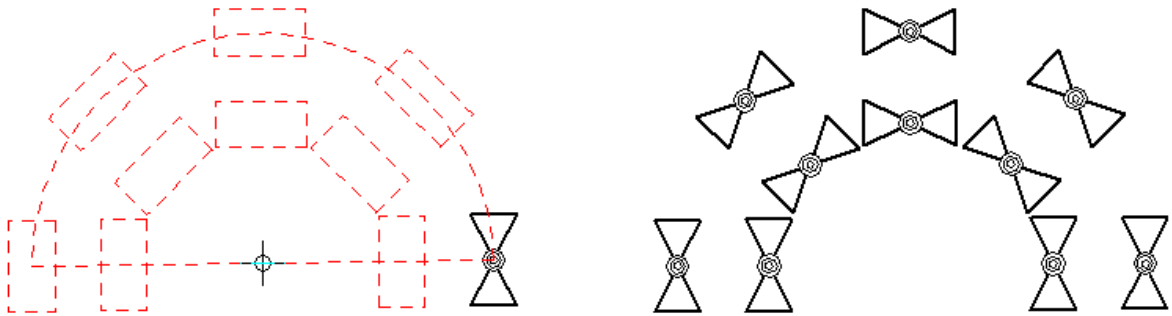
Include last column

Controls whether to include the last staggered column in the pattern or to exclude the last column.

Circular Pattern



The **Circular Pattern** tool arranges selected objects in a circular pattern on the drawing sheet.



To create a circular pattern select one or more objects to pattern, set the desired values in the ribbon bar and click the **Finish** button on the same ribbon bar.



Circular Pattern Options (see next page)

Count

Sets the number of copies in the pattern.

Rows

Sets the number of arc shaped rows in the pattern. This option is available only when the **Rows** option on the **Circular Pattern Options** dialog box is set to **Multiple Inward** or **Multiple Outward**.

Angle

Sets the angle of the pattern. This box sets the sweep angle between pattern objects.

Row Spacing

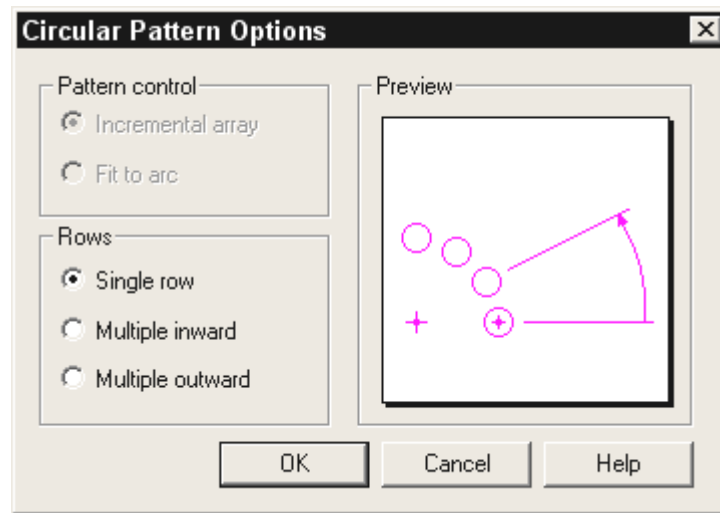
Sets the distance between adjacent arc-shaped rows in the pattern, as measured from the bottom of one row to the bottom of the next. The bottom is the side closest to the center point of the circular pattern.

Finish

Completes the pattern. When you click this button, the pattern is created on the drawing sheet and you can select other objects.



Circular Pattern Options Dialog Box



Pattern Control

Specifies how the pattern is constructed.

Incremental Array

Draws a pattern with a set sweep angle between members.

Fit To Arc

Draws a pattern by evenly spacing members along an arc.

Rows

Controls how pattern rows are arranged.

Single Row

Draws a single row of pattern members.

Multiple Inward

Draws multiple rows, each one closer to the pattern center than the last.

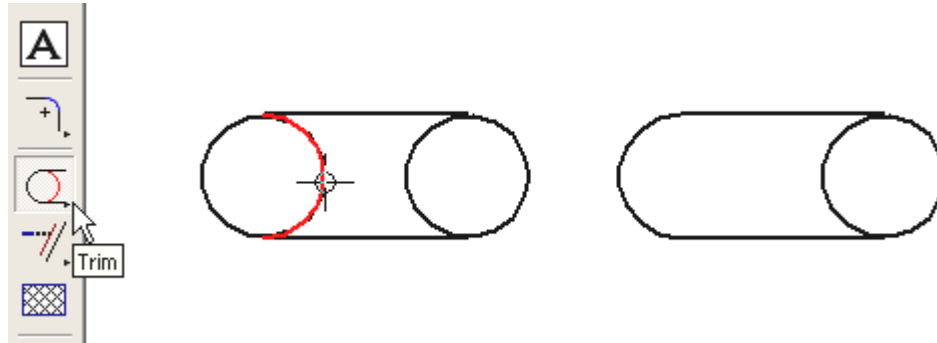
Multiple Outward

Draws multiple rows, each one farther away from the pattern center than the last.

Trim



The **Trim** command trims open and closed objects to the closest intersection in both directions. To trim one object at a time, select each object to trim. Only the portion of the object that will be trimmed will be highlighted.



To trim more than one object at the same time, drag the cursor over the elements. When releasing the cursor button, the objects are trimmed.

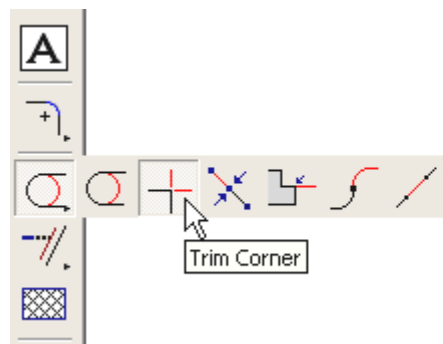


If trimming an object that does not intersect any other elements, the command trims the entire object, effectively deleting the object.

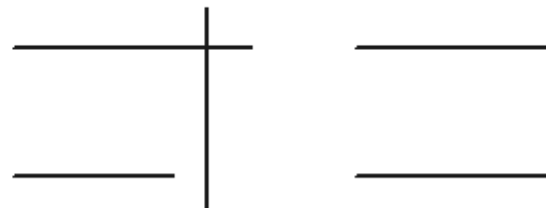
Trim Corner



The Trim Corner command creates a corner by extending and/or trimming two selected open geometric objects. You may use either the “**click**” method or “**drag**” method to trim or extend a corner. When using the drag method the parts of the object over which you dragged the cursor remain.



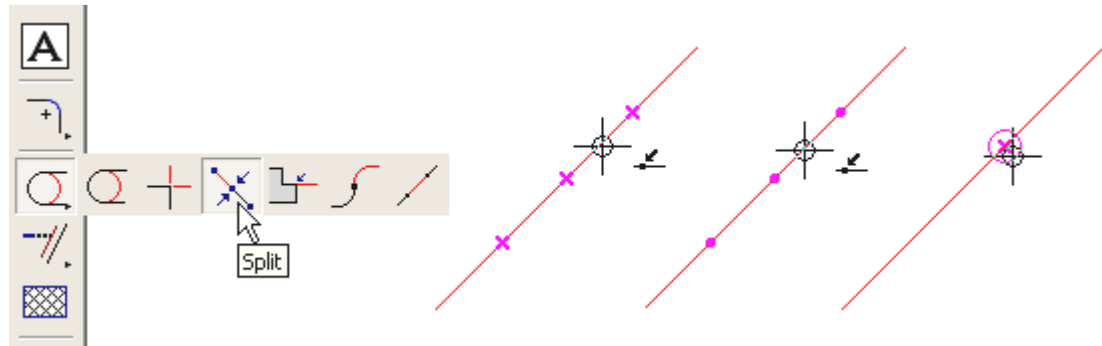
Only one corner can be drawn at a time by dragging the cursor.



Split

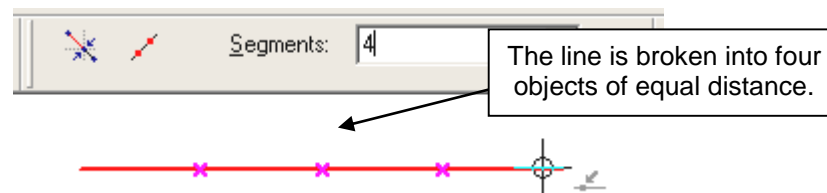


Provides several modes for splitting graphic objects. (A graphic object is any primitive geometric shape, such as a line, circle, or arc.) The object can either be broken into several segments, or simply mark the object with equidistant points.



Breaking an object into equal pieces

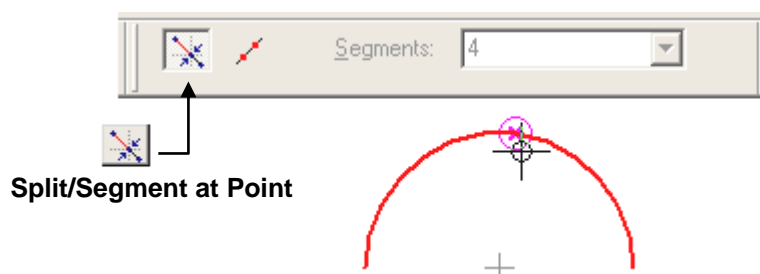
The **Split** mode is enabled by default whenever the **Split** command is selected. From the **Draw** toolbar, select the **Split** tool. On the **Split** ribbon, enter the number of breaks in the **Segments** field – you may enter a value from 1 to 100. Select a graphic object in the SmartSketch drawing.



As long as the **Split** command is active, you can continue splitting graphic objects by selecting them on the Drawing sheet. Press **Esc** or right-click an empty space on the active Drawing sheet to exit the **Split** command.

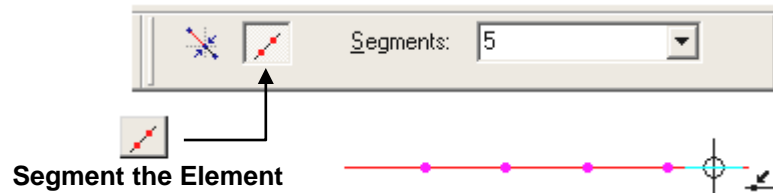
To split the selected object at a specific point

On the **Split** ribbon, click the **Split/Segment at Point** icon. Select a graphic object in the SmartSketch drawing to divide the object at a specific point.

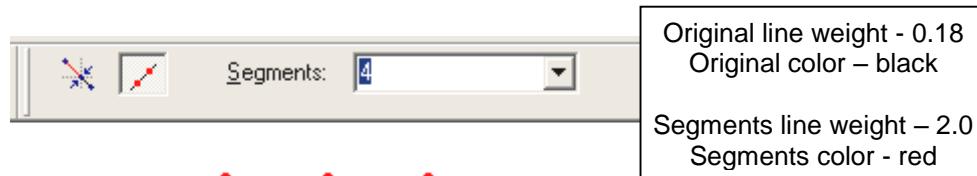


To place points equally spaced along an object

Activate the **Split** tool, and on the ribbon bar, click **Segment the Element** to enable **Segment** mode. Then, enter the number of segments in the **Segments** field. Select a graphic element in the SmartSketch drawing.



When this command is active, objects will not be broken up into pieces, but rather will place points at equal positions along the object. It is possible to see the points and use them when drawing. To do this, change the **Line Color** setting to contrast with the element you will be segmenting, and be sure to increase the **Line Weight**. When you complete the steps to segment the graphic object, you will see points along the element.

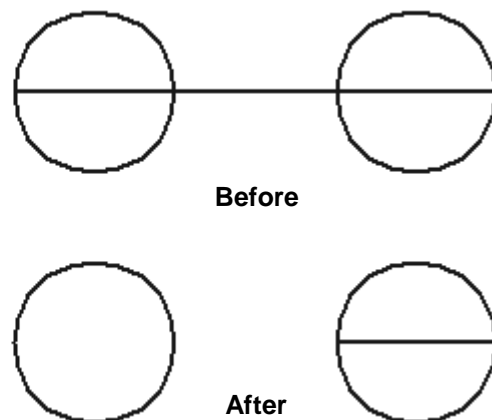
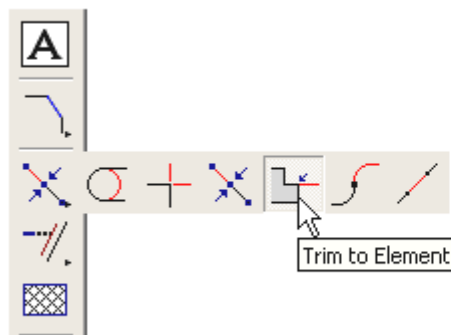


NOTE: **Segment the Element** may be toggled on simultaneously with **Split/Segment at Point**. The result will be an element that is not split into two pieces, but a point will be placed at the location indicated along the graphic object.

Trim to Element Command



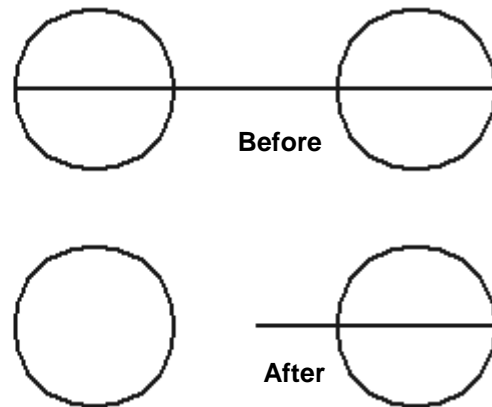
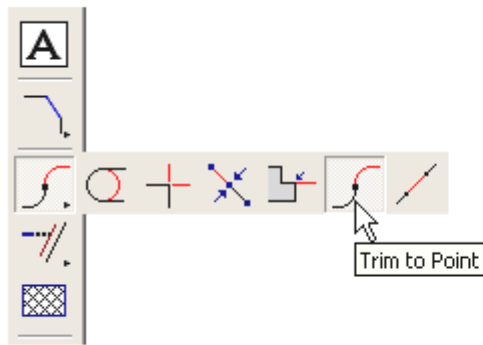
Trims open and closed elements to another selected object. Select the object to be trimmed and then select the object to be trimmed to.



Trim to Point Command



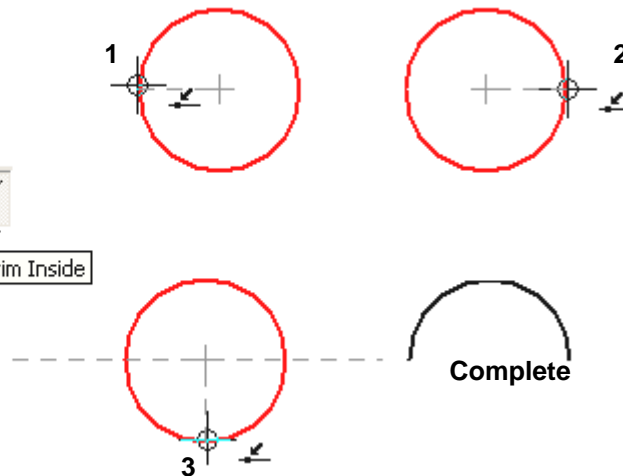
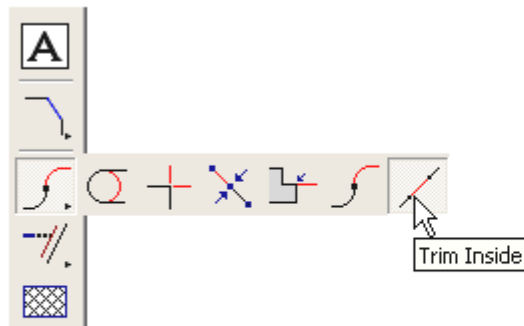
Trims within the selected element (partial delete). This command requires three points of reference. The first point determines a “start” point of the trim. The second point determines the “end” point of the trim. A third and final click determines which side of the cutting plane will be trimmed.



Trim Inside Command



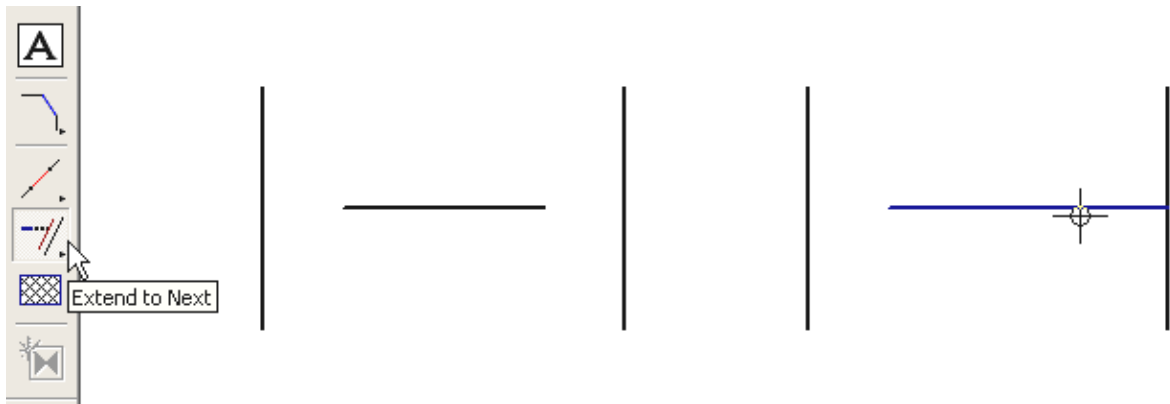
Trims within the selected element (partial delete). This command requires three points of reference. The first point determines a “start” point of the trim. The second point determines the “end” point of the trim. A third and final click determines which side of the cutting plane will be trimmed.



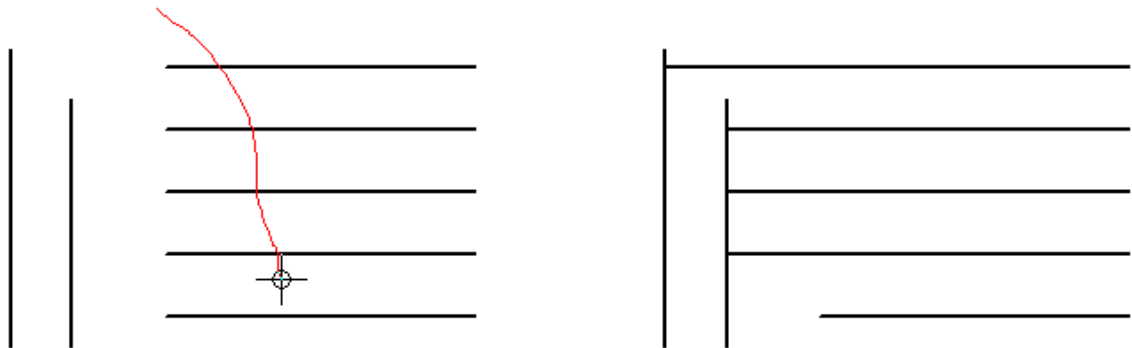
Extend to Next



The **Extend to Next** command extends one or more open elements until they intersect with the nearest object in the active window. The extension direction is determined by the position where you select the element to extend. For example, if you select a horizontal line to the right of its midpoint, the line extends to the right, then simply click the left-mouse button to accept the extension.



To extend more than one object at the same time, drag the cursor over the objects near the end to be extended. When releasing the cursor button, all the highlighted elements are extended.



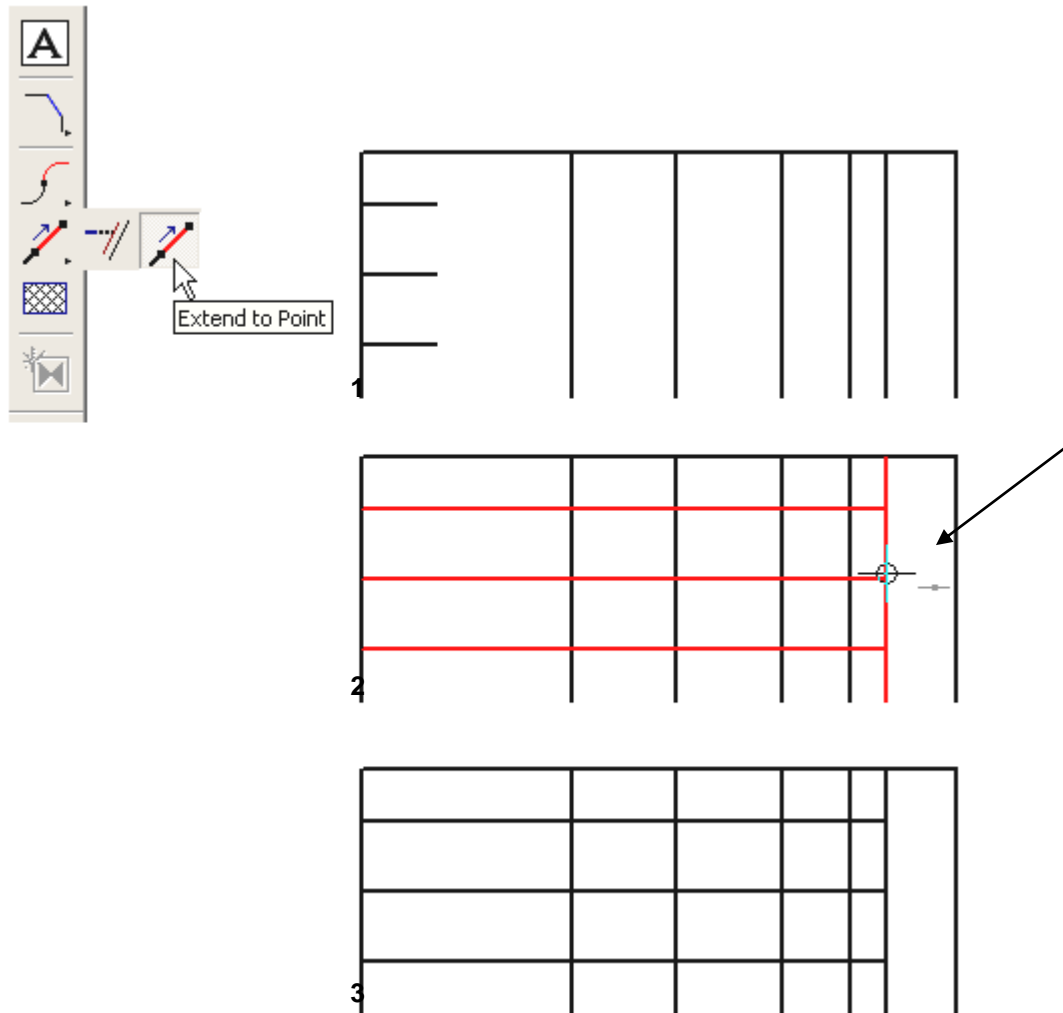
If an object does not extend in the desired direction, move the cursor closer to the end of the element to be extended.

Extend to Point



Extends one or more open geometric objects to a point in space or to another object. The extension direction is determined by the position where you select the object to extend.

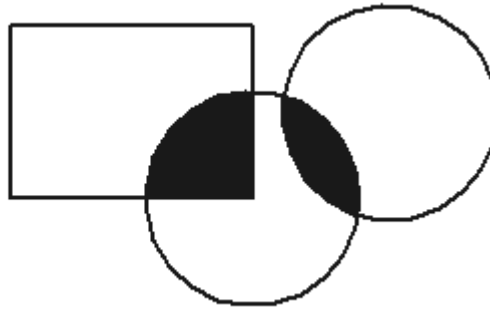
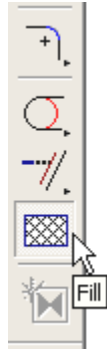
Select the item(s) to be extended with the **Select Tool** and then click the **Extend to Point** command. Click on the point that the object(s) will be extended to, and to complete the operation, right-click the mouse button in a clear area of the drawing space.



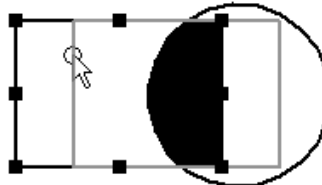
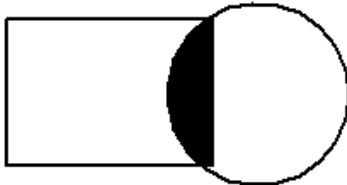
Fill



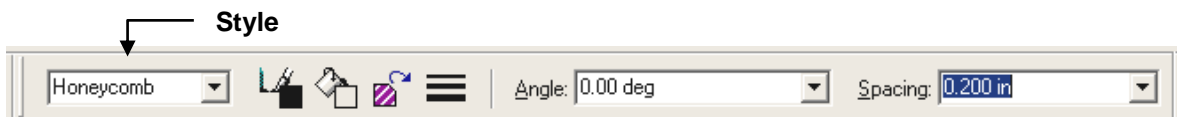
A boundary can be filled with a pattern or a solid color using the **Fill** command on the **Draw** toolbar. Click inside one or more closed boundaries that you want to fill.



A fill can exist only inside a closed boundary. The boundary can be made up of more than one object. A fill is associative, which means if you change the boundary, the fill changes to conform to the new boundary area.



Fill Ribbon Bar



Style

Lists and applies the available styles.



Pattern Color

Applies a pattern line color for pattern fills. If selecting **None**, the background will be transparent. Filled elements cover other objects when they overlap. Select the **More** option to define custom colors with the **Colors** dialog box.



Solid Color

Applies a system color to set the background color for the fill. If selecting **None**, the background will be transparent. Filled elements cover other objects when they overlap. Select the **More** option to define custom colors with the **Colors** dialog box.



Redo Fill

Re-applies a fill to a filled area when the boundary changes. A fill can become disabled, and change color, if its boundary is modified by drawing another object or moving part of the existing boundary. If clicking **Redo Fill**, the area surrounding the fill handle is filled again. See the example below:



Line Width

Applies a line width to pattern fills.

Angle

Sets the angle of the fill in the active unit. Zero degrees is horizontal along the x axis, and the angle increases in a counterclockwise direction from zero on the positive side of the x axis. If typing a negative value, the software displays the equivalent positive value.

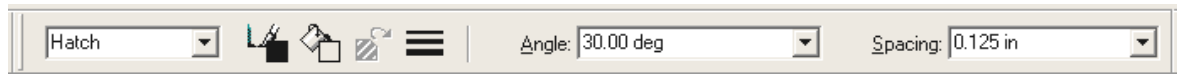
Spacing

Adjusts the spacing of the pattern lines in a fill.

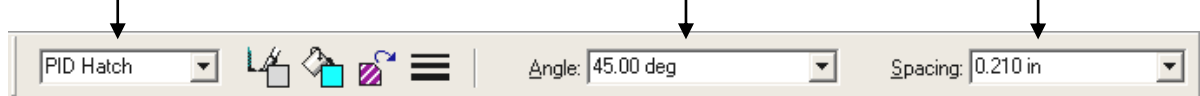
Create a Fill Style

You cannot modify an existing fill style, but you can create a new fill style by typing a new name in the **Style** list box on the **Fill** ribbon bar. The new style uses the settings on the ribbon bar as the formats for the style.

Existing Fill Type



New Name and New Settings

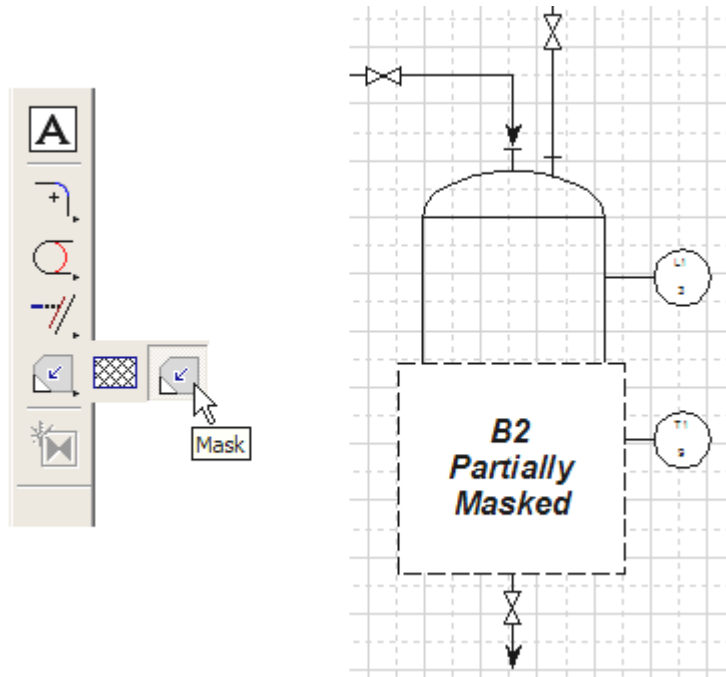


If you want to copy a fill style from one document to another, just create a fill, use the style in the first document, and then copy and paste the fill to a new document. The fill style appears in a dropdown list on the **Fill** ribbon bar in the new document.

Mask

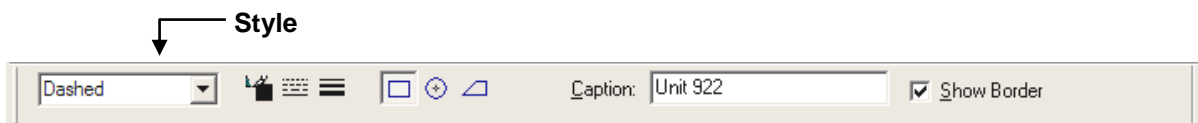


Masks out underlying graphics without deleting or trimming. SmartSketch provides a lot of flexibility for creating a mask, such as shape, line style, with or without a border or label (placed as a label in the center of the mask area). The example below shows a rectangular mask, border shown with dashed line style, with a caption.



Mask Ribbon Bar

Determines the shape, style and placement mode of the mask boundary object.



Style

Sets the line style for the mask.



Rectangular Mask

Restricts the placement mode of the mask to be rectangular.



Circular Mask

Restricts the placement mode of the mask to be circular.



Polygonal Mask

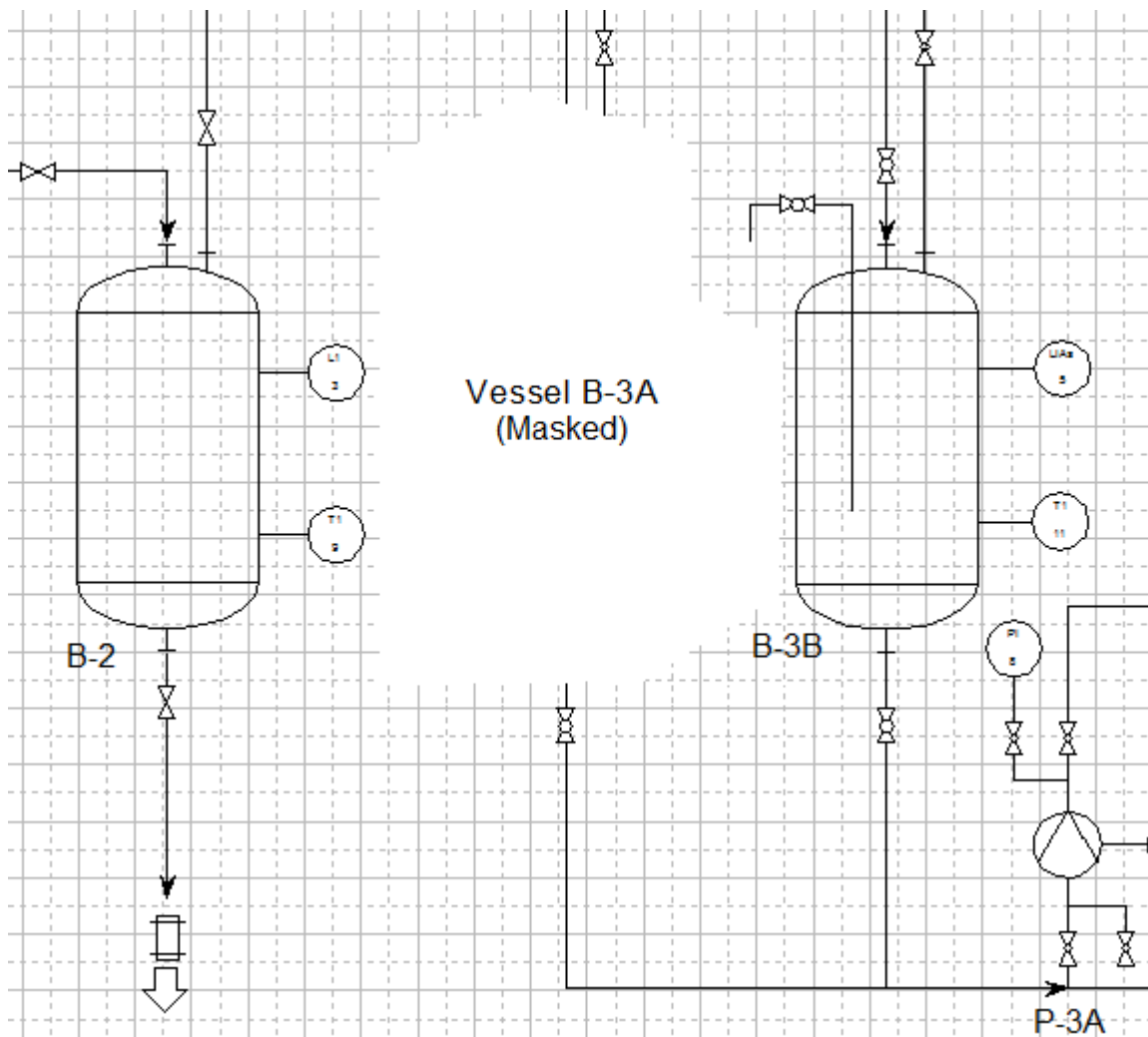
Restricts the placement mode of the mask to polygonal.

Caption

Specify the text to be placed as a label for the mask. If the Caption box is blank, a label will not be generated when you finish drawing the mask.

Show Border

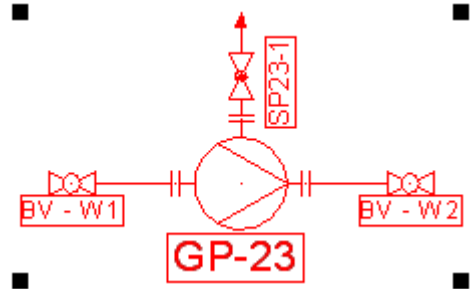
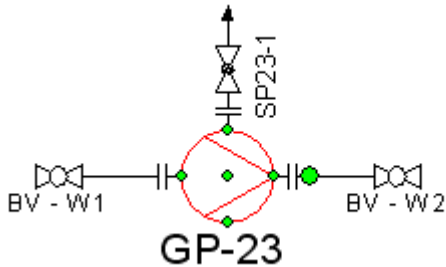
Specifies whether the border of the mask is displayed. If **Show Border** is unchecked, the color setting is ignored and the border object color is set to "blank".



Group



The **Group** command is located on the **Change** toolbar and is used to bind objects so that you can locate, select, and manipulate them as a unit. To group graphic objects on the drawing sheet use the **Select Tool** to designate the objects to be grouped together by creating a select set. Complete the operation by clicking the **Group** command.



In the illustration above, the figure on the left displays one item selected before the objects have been grouped. The figure on the right displays a group. The black handles display at the extreme corners of the grouped objects. Some properties of groups are:

- An object can be a member of only **one** group.
- Grouped objects must reside on the same drawing sheet and be in the same document.
- Any locatable object can be grouped.
- Any combination of graphic object types can be grouped.

Ungroup





Once objects are grouped, you can use the **Ungroup** command from the **Change** toolbar to ungroup all of the elements at the same time.

Selecting Groups and Members of Groups

Once you select a group, you can use a command or ribbon bar option on all of its members. For example, when you move, copy, or delete a group, all of its members are moved, copied, or deleted. You can move or apply relationships to a member of a group and the changes do not affect the position of the other members of the group.

Groups: Using the Select Tool Ribbon Bar

The options on the **Select Tool** ribbon bar allow you to select an entire group or individual

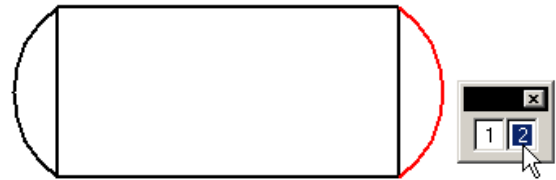
members of a group.  **Top Down** will select the individual members of a group.  **Bottom Up** will select the group as a whole.



Groups: Using PickQuick

You can also use **PickQuick** to select one or more group members or an entire group. If you use **PickQuick** to select a member of a group, you can perform tasks on the member individually, without affecting other members.

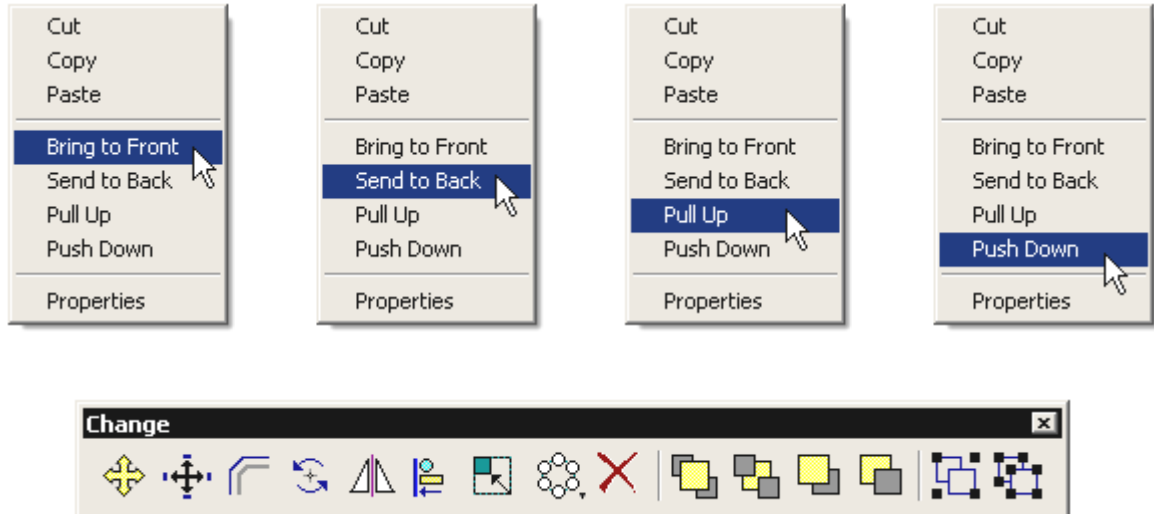
1. Use **PickQuick** to select the object you want to remove from the group.



2. From the **Change** toolbar, click the **Ungroup**  command. The object is removed from the group.

Display Priority Commands

The display priority commands can be selected from the **Change** toolbar or by clicking the right-mouse button over an object and selecting the command from the shortcut menu.



Bring to Front

Moves one or more selected objects or object groups to the front of the display order.



Send to Back

Moves one or more selected objects or object groups to the back of the display order.



Pull Up

Moves one or more selected objects or groups up one position in the display order.



Push Down

Moves one or more selected objects or groups down one position in the display order.

SECTION 10

Dimensioning and Measurement Tools

This lesson will cover tools that are used to dimension and measure graphic objects in SmartSketch.

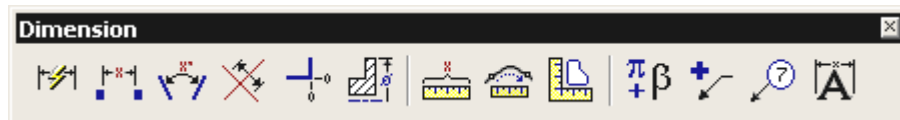
Topics

- Dimension Toolbar
- SmartDimension
- Distance Between
- Angle Dimension
- Coordinate Dimension
- Dimension Text
- Dimension Formats
- Driving and Driven Dimensions
- Measure Distance
- Measure Distance Along
- Measure Area
- Custom Dimension Styles

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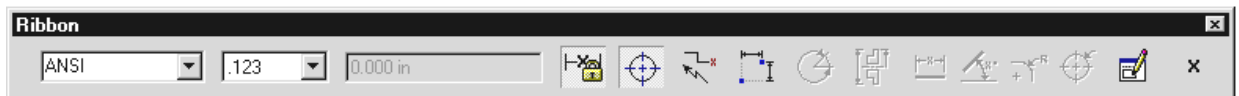
Dimension Toolbar

To dimension elements, choose one of the dimension commands from the **Dimension** toolbar, and then select the elements or keypoint to be dimensioned.



As dimensions are being placed, the software shows a temporary, dynamic display of the dimension. This temporary display shows what the new dimension will look like if the current mouse cursor position is selected. The dimension orientation changes depending on where the mouse cursor is moved.

Dimension commands on the **Dimension** toolbar have a ribbon bar that sets options for placing the dimension. When selecting a dimension in the drawing, the same ribbon bar options appear. The options can be used to change the selected dimension to a new value or modify its graphical appearance.

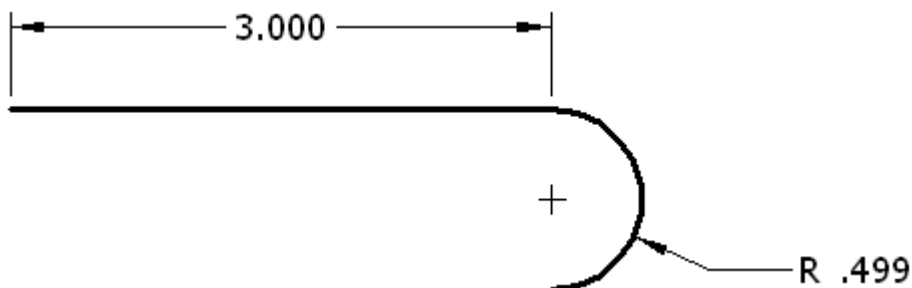


SmartDimension



SmartDimension is a dynamic dimensioning tool that is designed to place a dimension by clicking on the basic graphic object to initiate a dimension, and placing a second point to complete the placement of the dimension. SmartDimension will only dimension the following objects:

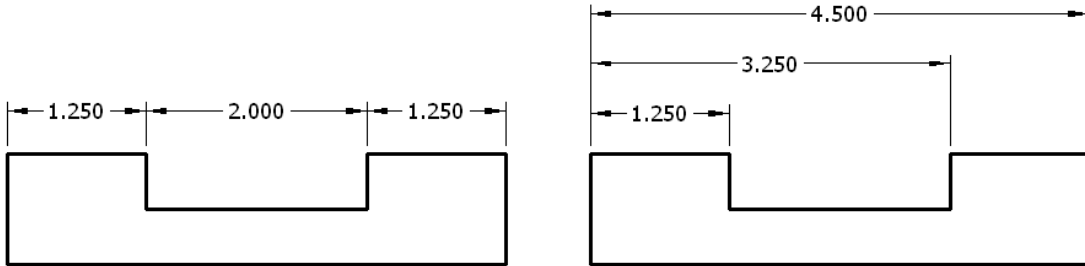
- Length and angle of a line
- Radius and diameter of a circle
- Length, angle, radius, and diameter of an arc
- Radius of an ellipse or curve



Distance Between



The **Distance Between** command places a linear dimension that measures the distance between objects or key points. You can place linear dimensions in stacked or chained dimension groups.



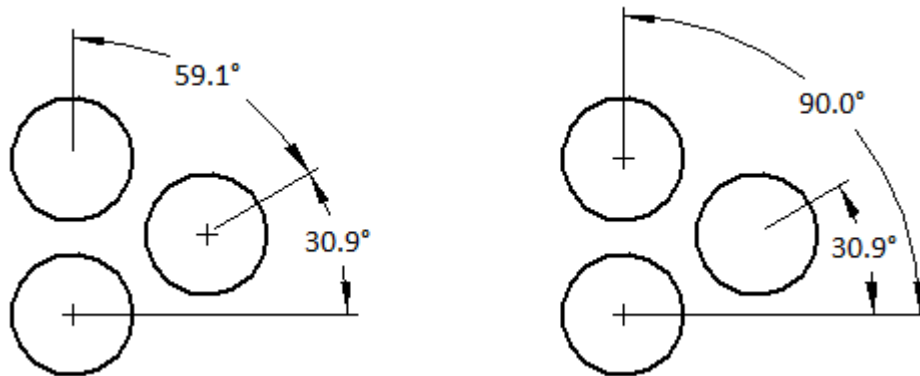
Move the mouse cursor where the dimension is to be placed. The dimension dynamically follows the movement of the mouse cursor. Right-click to finish the command.



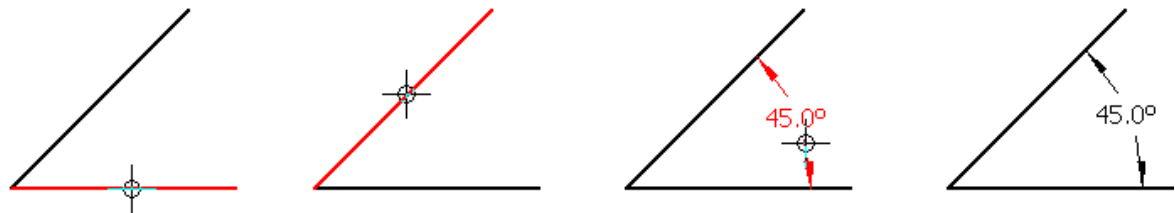
Angle Between



Places a dimension that measures the angle between elements or key points. You can place angular dimensions in stacked or chained dimension groups.



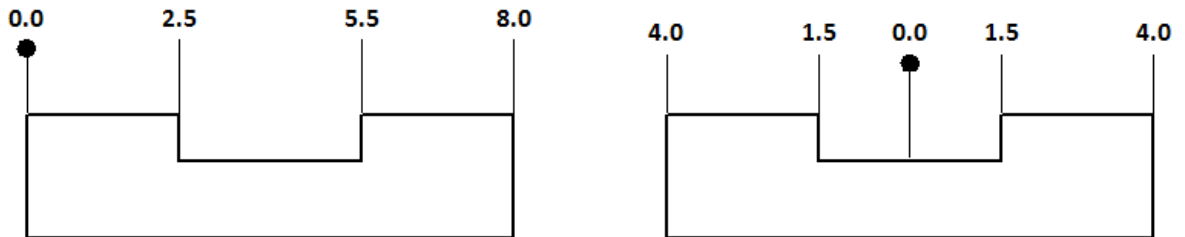
Move the mouse cursor where the dimension is to be placed. After defining two points the dimension dynamically follows the movement of the mouse cursor. Click to place the dimension. Right-click to complete the command.



Coordinate Dimension



Places a dimension that measures the distance from a common origin to one or more key points or objects. The coordinate dimensions that refer to the common origin are members of a coordinate dimension group.



You can place coordinate dimensions in any order and on either side of the origin with respect to the dimension axis. You can also add additional coordinate dimensions to existing coordinate dimension groups.

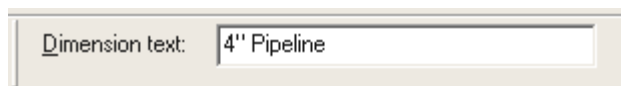
Dimension Text



Overrides a driven dimensional value with a text string.

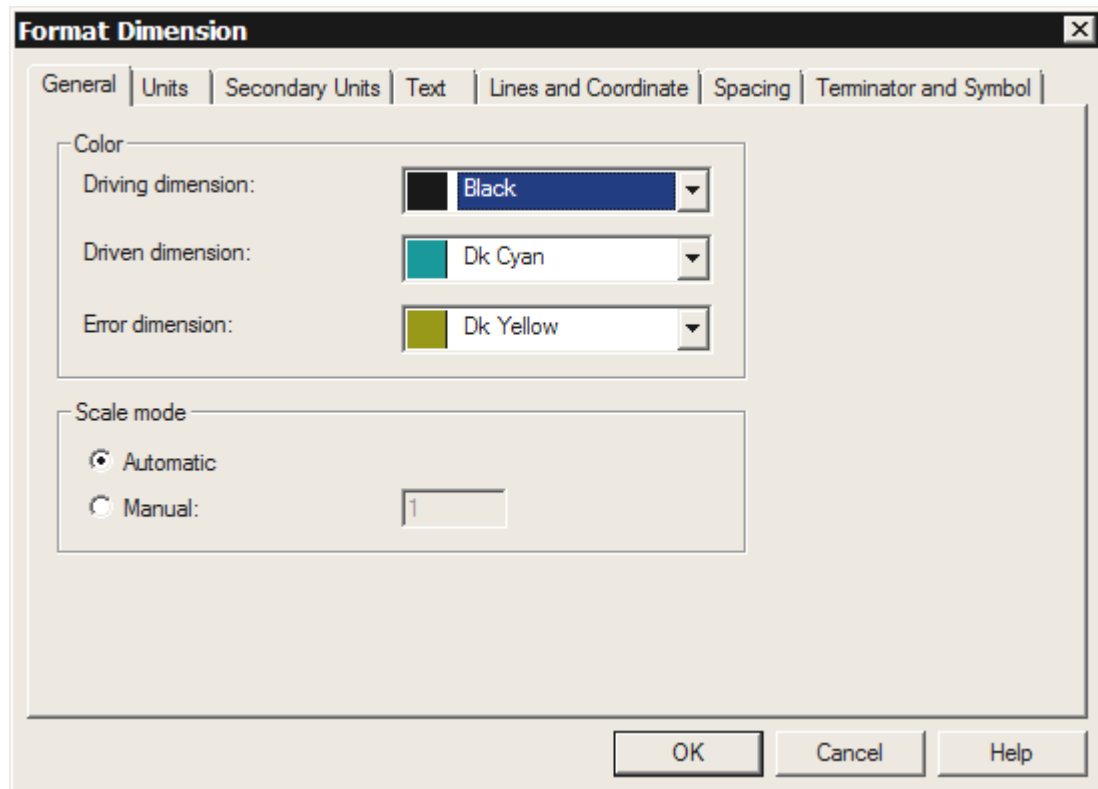


Click dimension text that you want to change. Key in new text string and press Enter to complete the command.



Dimension Format

The **Format Dimension** dialog box allows you to establish, or override active settings for dimensions. The dialog box allows the dimension unit, text size, font, and color of the dimension to be changed. The lines and terminator of the dimensions can also be altered to various preferences.



Color

Sets colors for the different dimension types: **Driving, Driven, or Error.** Error dimensions are dimensions that are not attached to elements. Error dimensions sometimes occur when inserting foreign data into a document.

Scale Mode

Sets the scale mode to automatic or manual.

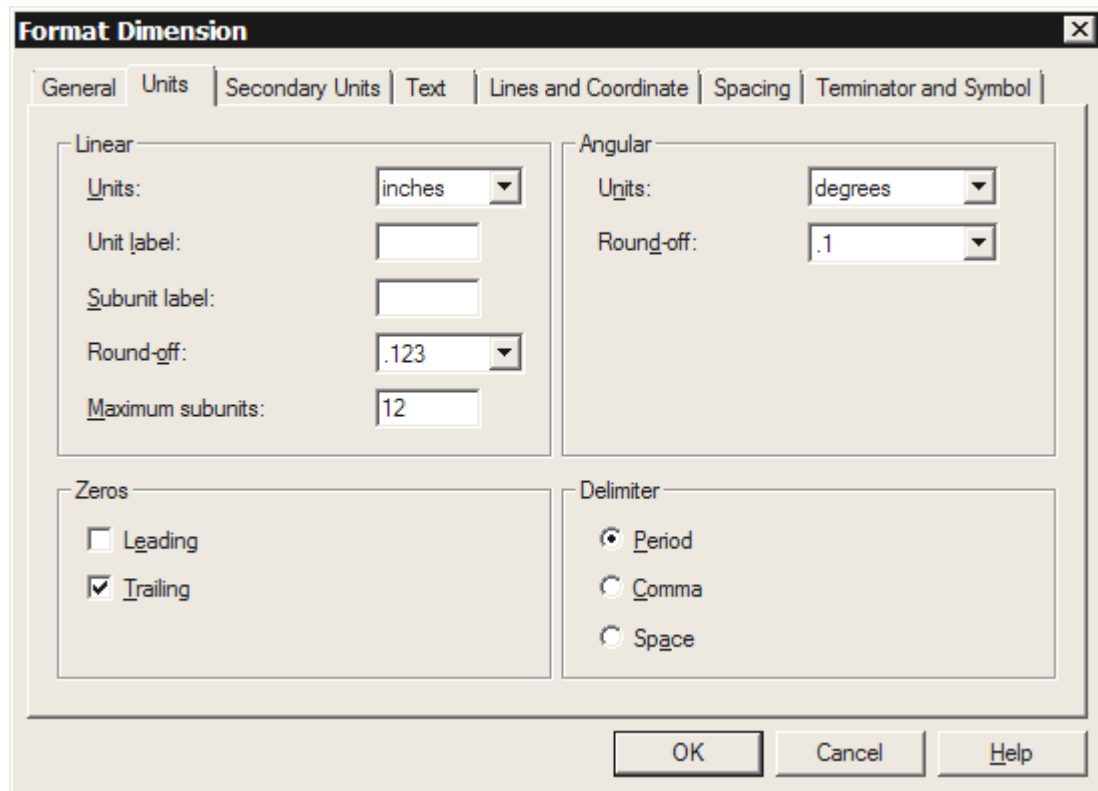
Automatic

Determines a correct dimensional value based on the scale set in the drawing view.

Manual

Scales a dimensional value. The scale value determines the dimensional value displayed.

Units Tab



The image shows the 'Format Dimension' dialog box with the 'Units' tab selected. The dialog has a title bar with a close button. Below the title bar are tabs: 'General', 'Units' (selected), 'Secondary Units', 'Text', 'Lines and Coordinate', 'Spacing', and 'Terminator and Symbol'. The 'Units' tab is divided into four sections: 'Linear', 'Angular', 'Zeros', and 'Delimiter'. The 'Linear' section has fields for 'Units' (set to 'inches'), 'Unit label' (empty), 'Subunit label' (empty), 'Round-off' (set to '.123'), and 'Maximum subunits' (set to '12'). The 'Angular' section has fields for 'Units' (set to 'degrees') and 'Round-off' (set to '.1'). The 'Zeros' section has checkboxes for 'Leading' (unchecked) and 'Trailing' (checked). The 'Delimiter' section has radio buttons for 'Period' (selected), 'Comma', and 'Space'. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

Section	Property	Value
Linear	Units	inches
	Unit label	
	Subunit label	
	Round-off	.123
	Maximum subunits	12
Angular	Units	degrees
	Round-off	.1
Zeros	Leading	<input type="checkbox"/>
	Trailing	<input checked="" type="checkbox"/>
Delimiter	Period	<input checked="" type="radio"/>
	Comma	<input type="radio"/>
	Space	<input type="radio"/>

Linear

Specifies the unit settings for a linear dimension.

Units

Sets the dimension units. This can differ from the Units property of the file.

Unit label

Sets the unit label for the dimension. You may enter any upper or lower case characters as required to be applied on the dimension (e.g. 'in' or 'In' or 'IN').

Subunit label

Sets the sub unit label to your choice of characters if you pick a Units option that includes a subunit display.

Round-off

Sets the round off for the value. This control is sensitive to the unit setting (decimal or fractional) and contains values as appropriate for the unit. It is also sensitive to the dimension being placed and contains values as appropriate for the dimension.

Maximum subunits

Sets the maximum subunits allowed, which typically applies to feet and inches. A value up to 18 can be entered. For example, if you type in 18, any dimension that is greater than one foot but less than a foot and a half shows in inches (e.g. "15 in" and not "1 ft and 3 in").

Angular

Specifies the unit settings for angular dimensions: **Degrees**, **Deg-Min-Sec**, or **Radians**.

Units

Sets the dimension units. This can differ from the **File > Property** units of the document.

Round-off

Sets the round off for the value. This control is sensitive to the unit setting (decimal or fractional) and contains values as appropriate for the unit. It is also sensitive to the dimension being placed and contains values as appropriate for the dimension.

Zeros

Determines if a zero is placed on the left or right of the decimal in a dimension.

Leading

Places a zero to the left of the decimal point if there are not any numbers to the left.

Trailing

Places zeroes to the right of the decimal point. The number of zeros placed is based on the active setting for Round-Off. For example, if the dimensional value is .5, and the round-off setting is .1234, the dimensional value appears as .5000.

Delimiter

Specifies the decimal delimiter for a dimension.

Period

Sets a period as the decimal delimiter.

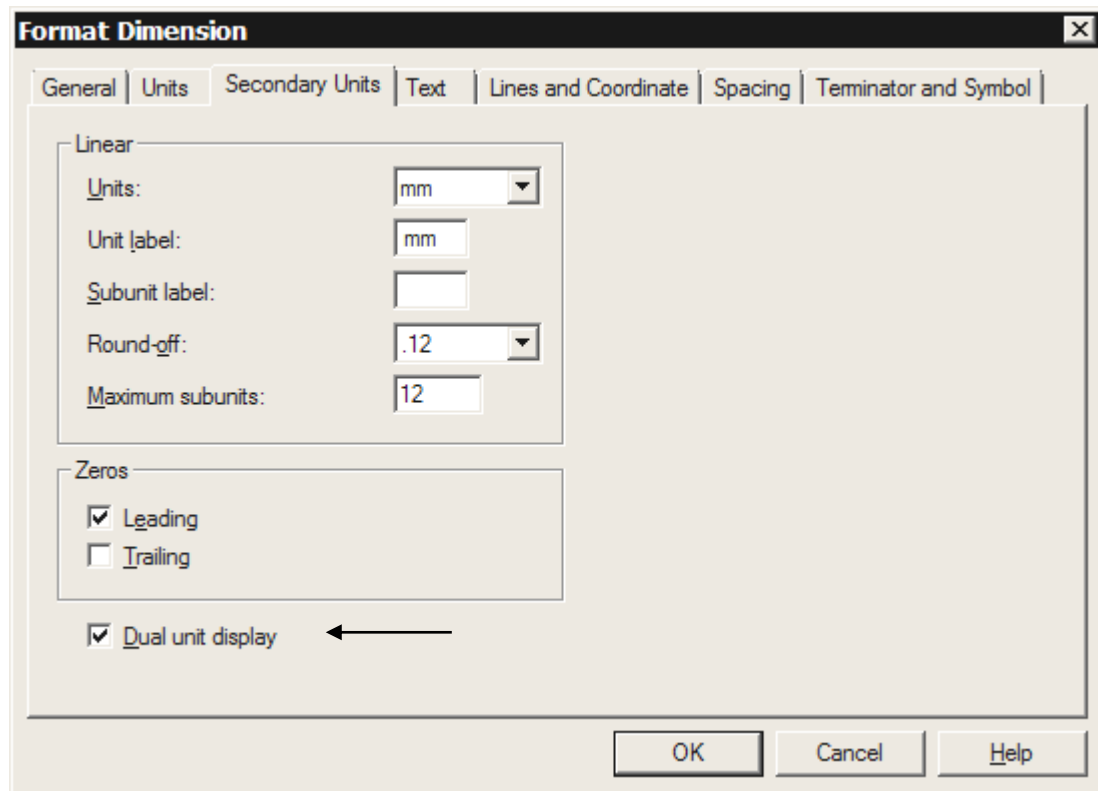
Comma

Sets a comma as the decimal delimiter.

Space

Sets a space as the decimal delimiter.

Secondary Units Tab



The image shows the 'Format Dimension' dialog box with the 'Secondary Units' tab selected. The dialog has several tabs: General, Units, Secondary Units, Text, Lines and Coordinate, Spacing, and Terminator and Symbol. The 'Secondary Units' tab contains two main sections: 'Linear' and 'Zeros'. The 'Linear' section has five fields: 'Units' (a dropdown menu showing 'mm'), 'Unit label' (a text box showing 'mm'), 'Subunit label' (an empty text box), 'Round-off' (a dropdown menu showing '.12'), and 'Maximum subunits' (a text box showing '12'). The 'Zeros' section has two checkboxes: 'Leading' (checked) and 'Trailing' (unchecked). Below these sections is a checkbox for 'Dual unit display' which is checked, with an arrow pointing to it from the left. At the bottom right of the dialog are three buttons: 'OK', 'Cancel', and 'Help'.

Linear

Specifies the unit settings for a linear dimension.

Units

Sets the secondary units in drawings set up for dual unit display. For example, the primary unit could be inches, while the secondary unit could be millimeters. When you place the dimension, it displays both units. Converting the primary unit values derives the secondary unit.

Unit label

Sets the secondary units label.

Subunit label

Sets the secondary subunit label.

Round-off

Sets the round-off value for secondary units.

Maximum subunits

Sets the maximum subunits used for secondary subunits.

Zeroes

Specifies whether a zero is on the left or right of the decimal in a dimension.

Leading

Places a zero to the left of the decimal point if there are no significant digits to the left (e.g. '0.54' instead of '.54')

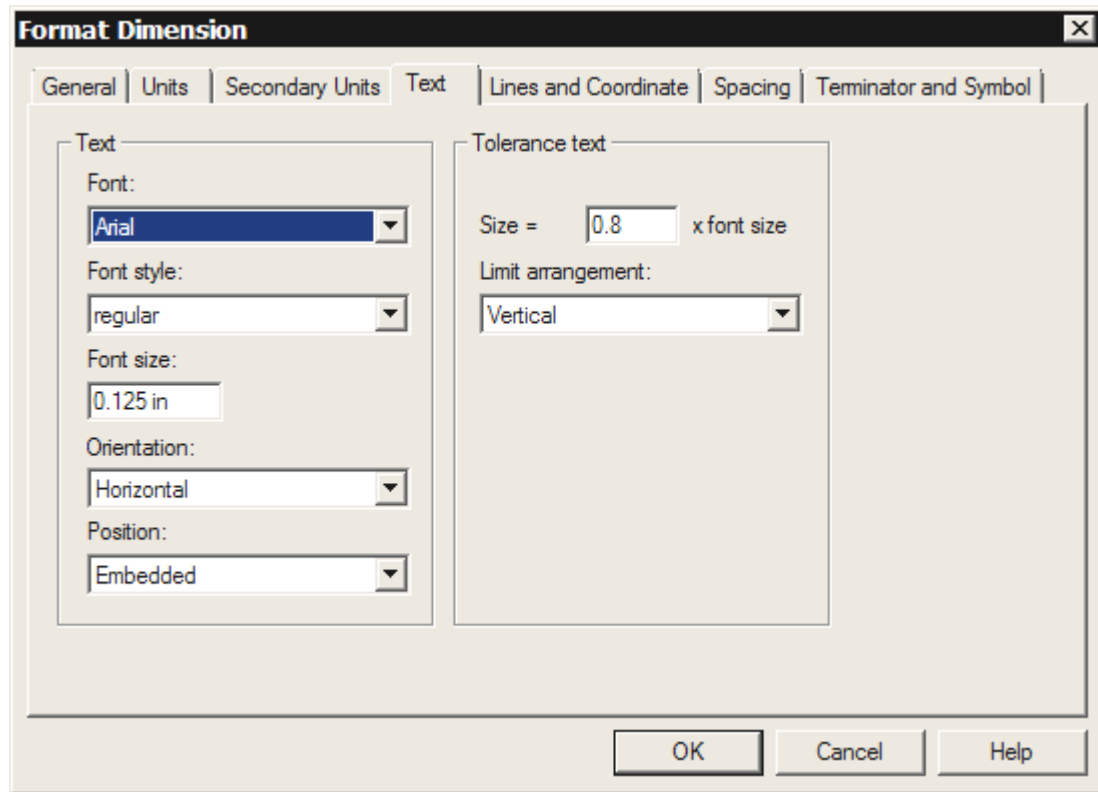
Trailing

Places zeros to the right of the decimal point. The number of zeros placed is based on the active setting for Round-Off. For example, if the dimensional value is exactly .5, and the round-off setting is .1234, the dimensional value adds three zeros to display it as .5000.

Dual Unit Display

This check box must be set to display secondary units for dimensions in drawings; otherwise it only places the primary dimensional value. For example, the primary unit might be inches, with the secondary units displayed as millimeters. When you place the dimension, it displays both units – inches over millimeters as a single dimension object.

Text Tab



Text

Sets options for the text within a dimension.

Font

Sets the font used for the dimension text.

Font style

Enters the font style to be used for the text within a dimension (Bold, etc.)

Font size

Enters the font size to be used for the text within a dimension. By default this should be in true paper size, regardless of the scale setting for the sheet.

Orientation

Sets the orientation of the dimension text with respect to the dimension line. The orientation can be set to horizontal, vertical, perpendicular, or parallel.

Position

Sets the position where text appears in relation to the base line. The base line is an imaginary horizontal line directly beneath a line of text.

Tolerance Text

Sets options for text in certain types of dimensions that have related tolerances. You can choose these from the **Dimension Type** button on the **Dimension** ribbon bar.

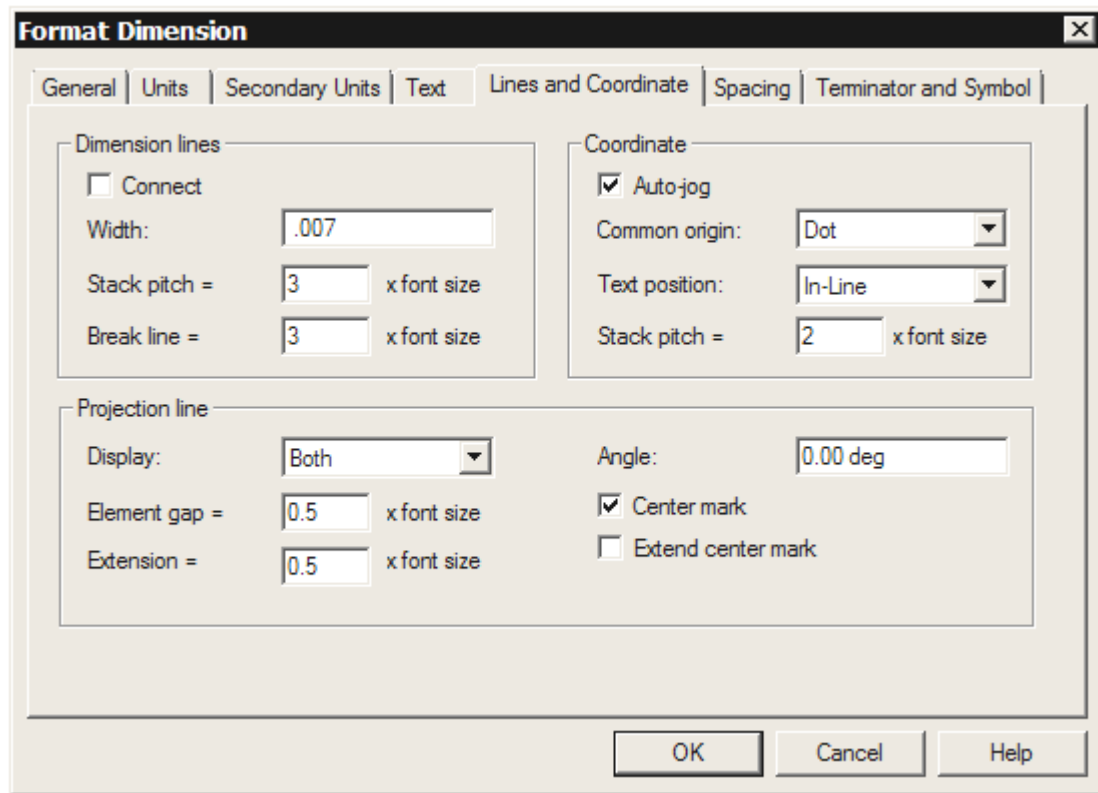
Size

Sets the size of the text for tolerance text. The value is a ratio of the dimension text size. For example, if you enter .5, the size of the tolerance text will be half the size of the dimension text.

Limit arrangement

Sets the text arrangement on limit dimensions as either Horizontal or Vertical.

Lines and Coordinates Tab



The image shows the 'Format Dimension' dialog box with the 'Lines and Coordinate' tab selected. The dialog has several tabs: General, Units, Secondary Units, Text, Lines and Coordinate (active), Spacing, and Terminator and Symbol. The 'Lines and Coordinate' tab contains three main sections: 'Dimension lines', 'Coordinate', and 'Projection line'. The 'Dimension lines' section has a 'Connect' checkbox (unchecked), a 'Width' field set to '.007', and 'Stack pitch' and 'Break line' fields both set to '3' with a multiplier 'x font size'. The 'Coordinate' section has an 'Auto-jog' checkbox (checked), a 'Common origin' dropdown set to 'Dot', a 'Text position' dropdown set to 'In-Line', and a 'Stack pitch' field set to '2' with a multiplier 'x font size'. The 'Projection line' section has a 'Display' dropdown set to 'Both', an 'Angle' field set to '0.00 deg', and 'Element gap' and 'Extension' fields both set to '0.5' with a multiplier 'x font size'. There are also checkboxes for 'Center mark' (checked) and 'Extend center mark' (unchecked). At the bottom are 'OK', 'Cancel', and 'Help' buttons.

Section	Property	Value
Dimension lines	Connect	<input type="checkbox"/>
	Width	.007
	Stack pitch	3 x font size
	Break line	3 x font size
Coordinate	Auto-jog	<input checked="" type="checkbox"/>
	Common origin	Dot
	Text position	In-Line
	Stack pitch	2 x font size
Projection line	Display	Both
	Angle	0.00 deg
	Element gap	0.5 x font size
	Extension	0.5 x font size
	Center mark	<input checked="" type="checkbox"/>
	Extend center mark	<input type="checkbox"/>

Dimension Lines

Connect

Controls whether the dimension line extends between both terminators when you place the dimension text and terminators outside the projection lines.

Width

Sets the width of the dimension line.

Stack Pitch

Sets the distance between stacked dimensions. The value is a multiple of the dimension text size.

Break Line

Sets the size of the break line for the dimension line. This value is a multiple of the text size.

Coordinate

Sets options for the **Coordinate Dimension** tool.

Auto-Jog

Toggles the jog control on the **Dimension** ribbon bar. You can use this option only when placing a coordinate dimension. If you set **Auto-Jog** and the distance between two dimensions is less than the value set for **Stack Pitch**, then you can place the dimension with a jog in the projection line.

Common origin

Sets the symbol type for the common origin on coordinate dimensions. You can set the symbol type to dot, circle, or none.

Text position

Positions the text in a coordinate dimension as above or in-line.

Stack pitch

Sets the distance between stacked dimensions as a multiple of the dimension text size.

Projection Line

Display

Controls the display of projection lines on linear dimensions. You can set the display to none (neither visible), origin, measurement, or both the origin and measurement.

Element gap

Sets how far the projection line is set off from the element you are dimensioning. This value is a multiple of the dimension text size.

Extension

Sets how far the dimension line extends beyond the dimension. This value is a multiple of the dimension text size.

Angle

This is a very useful control to set the slant angle of the projection lines on dimensions for objects drawn with the isometric tools.

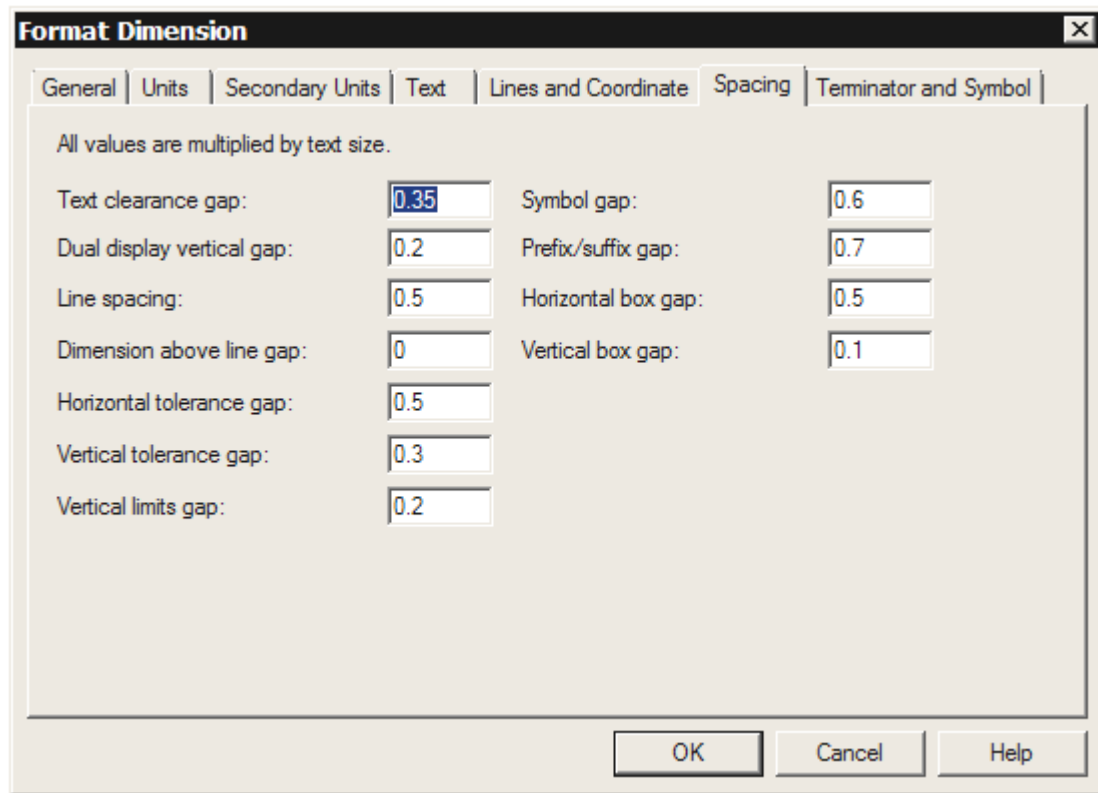
Center mark

Places a centerline automatically as you dimension arcs or circles.

Extend center mark

Displays projection lines on centerlines.

Spacing Tab



The image shows the 'Format Dimension' dialog box with the 'Spacing' tab selected. The dialog has a title bar with a close button (X). Below the title bar are several tabs: 'General', 'Units', 'Secondary Units', 'Text', 'Lines and Coordinate', 'Spacing' (which is active), and 'Terminator and Symbol'. The main area of the dialog contains a note: 'All values are multiplied by text size.' Below this note are several input fields arranged in two columns. The left column contains: 'Text clearance gap:' with a value of 0.35, 'Dual display vertical gap:' with a value of 0.2, 'Line spacing:' with a value of 0.5, 'Dimension above line gap:' with a value of 0, 'Horizontal tolerance gap:' with a value of 0.5, 'Vertical tolerance gap:' with a value of 0.3, and 'Vertical limits gap:' with a value of 0.2. The right column contains: 'Symbol gap:' with a value of 0.6, 'Prefix/suffix gap:' with a value of 0.7, 'Horizontal box gap:' with a value of 0.5, and 'Vertical box gap:' with a value of 0.1. At the bottom right of the dialog are three buttons: 'OK', 'Cancel', and 'Help'.

Property	Value
Text clearance gap:	0.35
Dual display vertical gap:	0.2
Line spacing:	0.5
Dimension above line gap:	0
Horizontal tolerance gap:	0.5
Vertical tolerance gap:	0.3
Vertical limits gap:	0.2
Symbol gap:	0.6
Prefix/suffix gap:	0.7
Horizontal box gap:	0.5
Vertical box gap:	0.1

Text clearance gap

Sets the space between the text and the dimension line.

Dual display vertical gap

Sets the space between the primary and secondary units when dual unit display is active. You can set the **Dual Unit Display** option on the **Secondary Units** tab so that dimensions display both units.

Line spacing

Sets the amount of space between the superfix or subfix and the dimension text.

Dimension above line gap

Sets the space between the dimension text and the dimension line.

Horizontal tolerance gap

Sets the space between the dimensional value and the tolerance on dimensions.

Vertical tolerance gap

Sets the space between the upper and lower tolerance value on dimensions.

Vertical limits gap

Sets the space between the upper and lower dimensional values on limit dimensions.

Symbol gap

Sets the space between the symbol and the dimension line. You can also set the space between the symbol and the dimension text.

Prefix/suffix gap

Sets the amount of space between a prefix or suffix and the dimension text.

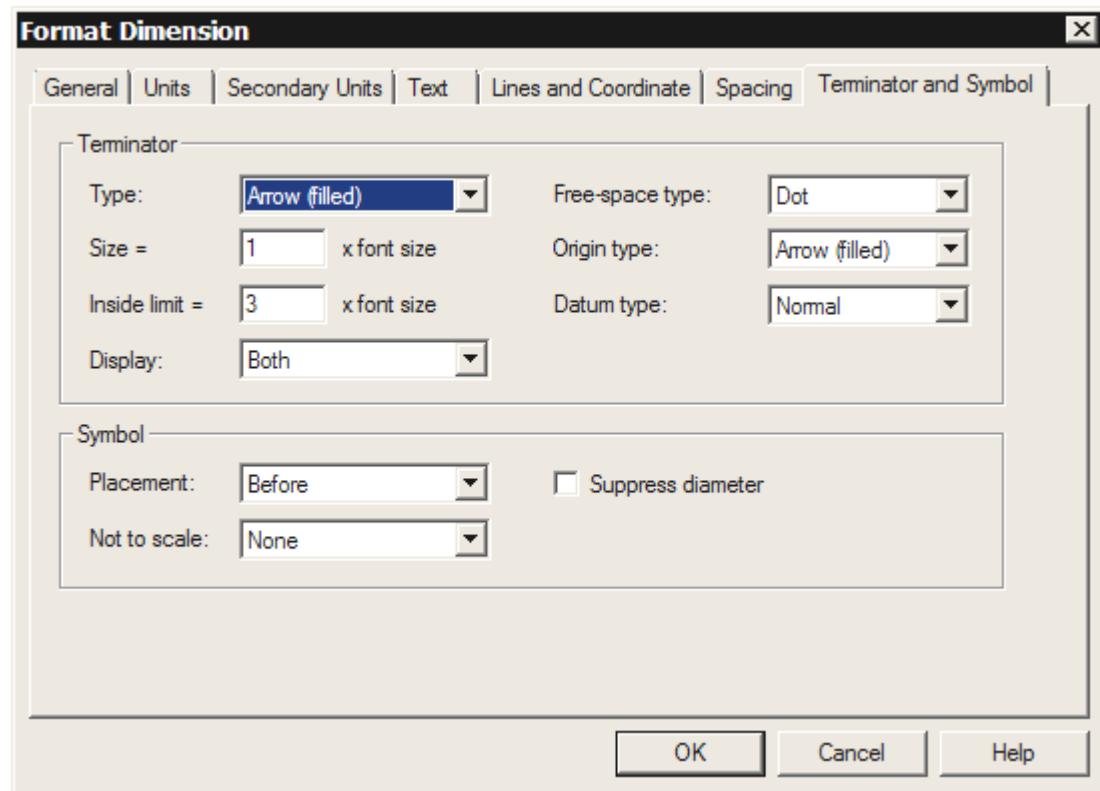
Horizontal box gap

Sets the space between the dimension text and the horizontal edges of the box on dimensions.

Vertical box gap

Sets the space between the dimension text and the vertical edges of the box on dimensions.

Terminator and Symbols Tab



Terminator

Specifies settings for the terminators on a dimension.

Type

Sets the terminator type for all terminators.

Size

Sets the size of the terminator. The value is a multiple of the dimension text size.

Inside limit

Sets the inside limit for the terminator. The value is a multiple of the dimension text size. This setting controls when the terminators are automatically toggled to the outside of the projection lines.

Display

Specifies which end of the terminator is displayed or if both ends are displayed. You can set the display to none (off), origin, measurement, or origin and measurement.

Free space type

Sets the terminator type for a dimension whose terminator end is placed in free space.

Origin type

Sets the terminator type used on the origin of a linear dimension.

Datum type

Sets the terminator type for datum frames. If you set the option to normal, the datum frame uses the active terminator type for dimensions. If you set the option to anchor, the datum frame uses an anchor terminator.

Symbol

Specifies settings for a symbol used in a dimension.

Placement

Sets the placement position for the symbol on diameter and radial dimensions and linear dimensions for an arc. You can place the symbol before or after the dimension. You can also hide the symbol.

Not to scale

Displays an underline, zig-zag, or no indicator on driven dimensions with overridden values. You can use the zig-zag option only on linear dimensions. You can override a driven dimension value by typing a new value in the Edit Value box on a dimension ribbon bar.

Suppress diameter

Suppresses the diameter symbol on diameter dimensions.

Driving and Driven Dimensions

Driving Dimension

A dimension that exercises control over the size or location of the element it is associated with is known as a **Driving Dimension**. Modifying the dimension changes the drawing objects.

To place a driving dimension, you must set **Maintain Relationships** on the **Tools** menu, select an element, and then click a dimension command on the **Dimension** toolbar; the dimension that you place by default should be a driving dimension.

Driven Dimension

A dimension with a value that depends on the value of other dimensions or elements. If the element changes, the dimensional value updates.



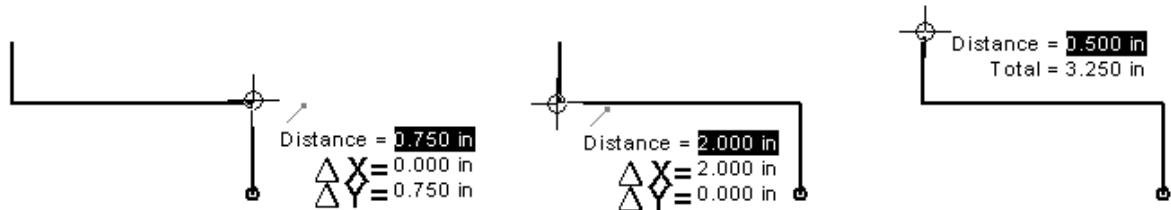
To change a driving dimension to a driven dimension, select a driving dimension and click the **Driving/Driven** button. Or this method can toggle it the other way – Driving to Driven if there is a need to allow the elements to control the dimension.

To set the colors for driving and driven dimensions, select the dimension and click **Properties** on the shortcut menu. Then set the options you want on the **Properties** dialog box.

Measure Distance



The **Measure Distance** command measures the distance between points on elements and points in free space. The first two points define a “from” location and a “to” location for the measurement. Additional points will increase the **Total** (or cumulative) distance measurement. The **Distance** display will reflect the measurement between the last two points placed. The delta values are the distances, as measured along the X and Y-axes.

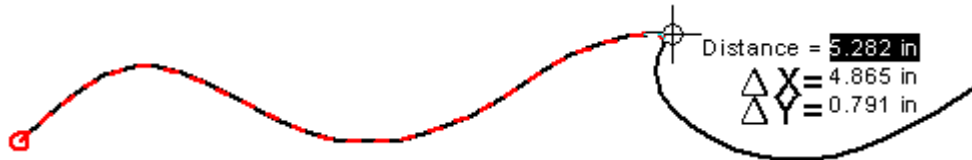


When you are in the middle of a task, you can use the **Measure Distance** command at any time.

Measure Distance Along



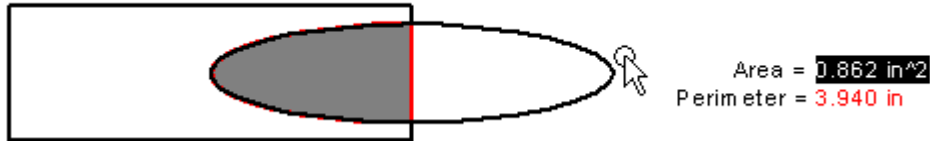
Measures the distance along a linear object based on two points. For closed shapes (curves, circles, ellipses), the measurement will default to a clockwise direction. You can hold the **Ctrl** key to change the measurement direction to counterclockwise. Left-mouse click to start the measurement, and a second click will end the command.



Measure Area



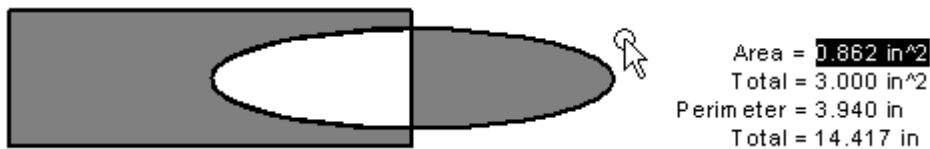
The **Measure Area** command measures the area inside one or more closed boundaries. Click inside a closed boundary to measure its area. The area measurement for the closed boundary will appear next to the pointer.



The **Area** value displays the most recently selected area of the boundary in the current units.

The measured area appears as a gray fill.

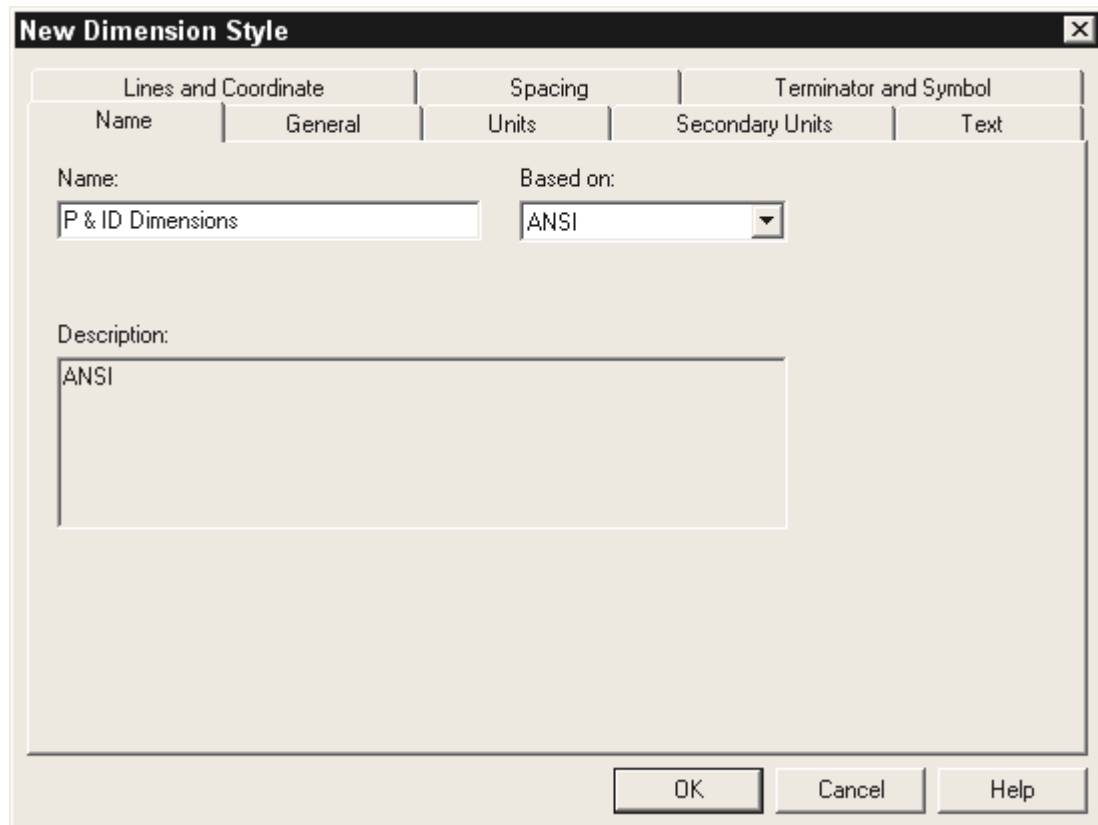
The **Total** value displays the total area of all selected boundaries in the current document units. You may press the **Shift** or the **Ctrl** keys while clicking to add or subtract areas from your measurement.



7. **Important:** You can press the **<Tab>** key to move between the distance and total distance. You can also press **Ctrl + C** to copy the value to the Clipboard. You can then press **Ctrl + V** to paste the value into a text box or ribbon bar field.

Custom Dimension Styles

For more recurrent format settings, remember that you can create unique dimension styles with **Style** on the **Format** menu. Set dimension in the **Style** dialog box and click **New** or **Modify**. The settings options available when using the **New Dimension Style** dialog box are the same settings that are found on the **Format> Dimension** dialog box, with the exception of the **Name** tab (see below).



Name Tab

Names a style when you create or modify a style. This tab appears only when you click **New** or **Modify** on the **Styles** dialog box.

Name

Style names can contain up to 253 characters (including aliases and separators) and can include any combination of characters and spaces, except the backslash character (\), semicolon (;), and braces ({ }). Style names are case sensitive.

Based On

Displays the name of the style that the current or new style is based on.

Description

Displays a description of the formatting options.

SECTION 11

Text and Annotation Tools

An essential part of the drawing process is adding text, notes, and annotations. This information can be added quickly and easily with the text and annotation tools.

Topics

- Placing Text Boxes
- Text Styles
- Text Labels
- Balloons
- Leaders
- Detail View
- Custom Text Style

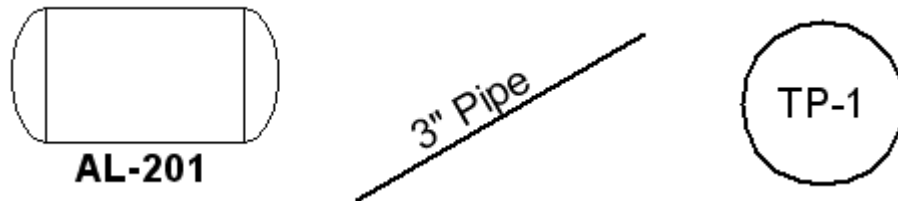
Text and Annotations

An essential part of the drawing process is adding text, notes, and annotations. Annotations are text and graphics that give information about a drawing. You can add text to documents quickly and easily with the Text Box, Text Label, and Balloon tools.

Text Boxes



Text Labels



Balloons



Fonts

The engineering fonts delivered with the software contain industry-specific fonts, special characters, and symbols that you can use to annotate engineering drawings. These fonts include degree symbols, diameter symbols, and other special characters and symbols that are not usually included in a typical word processing package.

Your choice of font should be based on the industry for which you are creating engineering drawings.

The software provides TrueType® fonts; with TrueType fonts, what you see on the screen is what appears on the printed page. The screen display of the document closely matches the printed document.

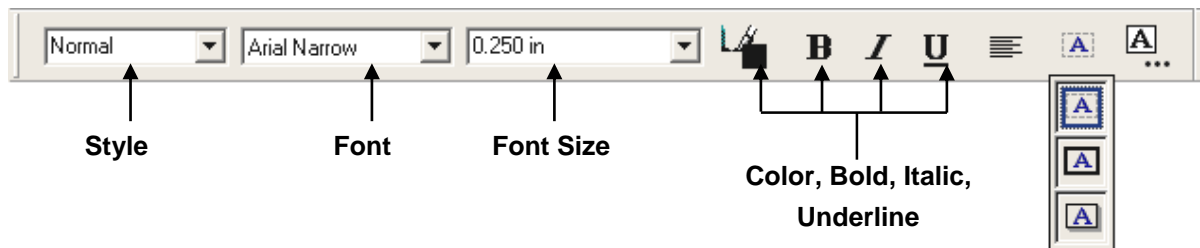
When you open a **.dwg** or **.dgn** document, the software provides default mapping to determine which TrueType font will be assigned to which AutoCAD® or MicroStation® font. You can make changes to this mapping to define your own font assignments. You can change the mapping with the **Options** command on the **Tools** menu. On the **Foreign Data** tab, you can opt to set the appropriate **Options** button (import or export).

Text Box



The **Text Box** command is used to place a text box in a document. A text box can be a single line of text, multiple lines of text, text boxes with a defined width, or text within a border. Text boxes differ from the other annotations in that they are always non-associative.

The **Text Box** ribbon bar is used to format a text box. Modification can be made to a text box after placement by selecting it and changing the desired setting on the ribbon bar.



Style

Lists available font styles.

Font

Lists and applies the available fonts.

Font Size

Applies a text size.

Font Settings

Sets the color of the font as well as determining if it is **Bold**, *Italicized*, or Underlined.



Paragraph Alignment

Positions the paragraph to the **left**, **center**, or **right** of the text box in edit mode.



Border

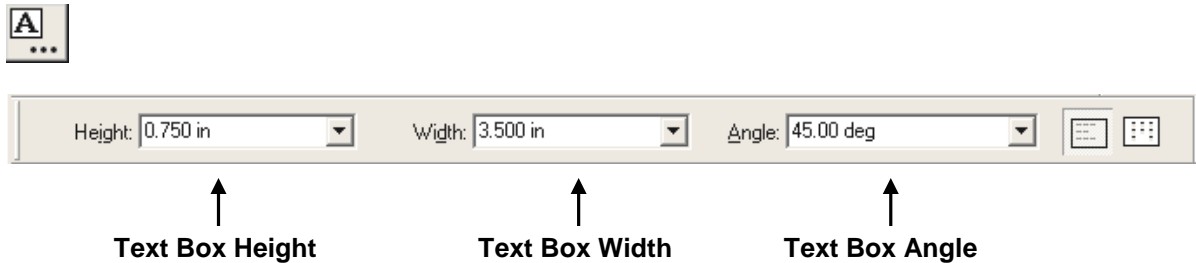
Permits three options for displaying borders around the text boxes. These options allow you to display **no border**, **display a border**, or **display a border with a shadow**.



More...

Displays more options on the **Text Extended** ribbon bar.

More...



Text Box Height and Width

Settings are affected by the method of placement.

- If you select text in the text box, **Height** and **Width** are not available
- If you select a text box and **Height** is not available, the text is driving **Height** with the **Auto** option (found in **Text Box Properties**). The same is true for **Width**.
- If you place a text box with a single point, both **Height** and **Width** are not available. If you drag a text box, **Width** is not available.



Horizontal Text Orientation

Specifies that the text is oriented horizontally in the document.



Vertical Text Orientation

Specifies that the text is oriented vertically in the document.

Placing Text

There are basically two methods that can be used to place text. One technique is the “**click**” method and the other is the “**drag**” method.

“Click” placement method

To place text using the “**click**” method, first you must set the desired text settings in the **Text Box** ribbon bar or **Format >Text Box** dialog box. Once this is completed, click a point at the point of placement in the drawing to activate a blinking cursor. After typing the text, click the right-mouse button to terminate the command.

When you place text by this method, the height and width of the text box are set to Auto mode. The size of the text box increases automatically to contain all of the text. The text does not wrap.

“Drag” placement method

As with the “**click**” method, it is recommended that you first establish the desired text box settings in the **Text Box** ribbon bar or **Format >Text Box** dialog box. Click and drag to define the location and size of the text box. Begin typing text, right-click the mouse when you are finished.

When you place text by this method, the text box width measures exactly between the two points. The text will automatically wrap based on the defined width of the text box. The height of the text box measures at least the height between the two points. If necessary, the height of the text box increases to fit all of the text.

Text Box Handles

The text box handles are used to manipulate the origin or the size of the text box while maintaining the relationship between the origin and justification.



Handles on text boxes appear as an X square, a hollow square, or a solid square.

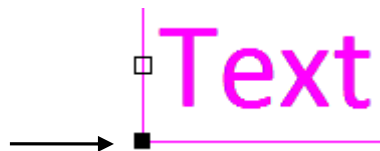
- An **X** handle indicates the origin of the text box. The origin is defined by the horizontal and vertical justification settings. Dragging an **X** handle moves the text box about its origin.



- The hollow handles are used to move the text box. The origin point moves with the textbox.

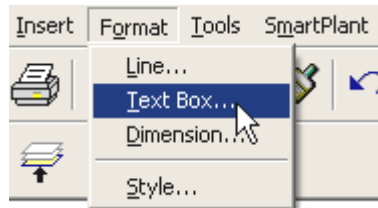


- The solid handles are used to resize the text box. The origin point remains in its current location while the text box is resized.

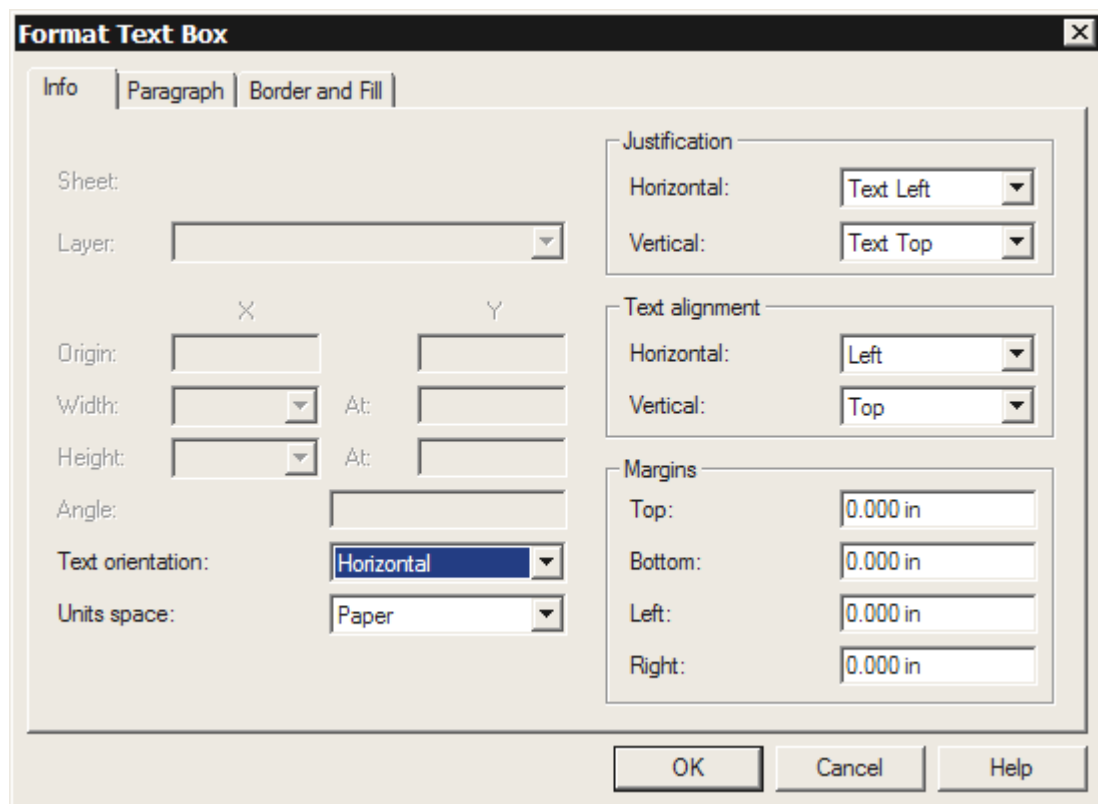


Text Box Format

The **Format Text Box** dialog box allows you to establish, or override active settings for text. The dialog box allows you to determine text size, font, color, and justification of the text to be changed. The margin and border settings of the text box can also be altered to various preferences



Info Tab



You will note the first six settings are disabled on the **Info** tab. These are available when you view the **Text Box Properties** dialog box, which will be discussed later in the lesson.

Text Orientation

Sets the orientation of the text either horizontally or vertically.

Units Space

Sets the text box units to either paper units or to world units.

Paper

Paper units represent the actual text size on a sheet of paper. They are not affected by the sheet scale settings. Paper unit values are defined by the **File>Properties, Units** setting.

World

World units indicate the text is scaled based on your **Sheet Setup** scale settings.

Justification

Specifies the placement relationship between the origin and the shape (text box) according to the horizontal and vertical components.

The inside text area is an area that the margins of the text box define. That is, margins are offset inside the perimeter of the text box.

Horizontal

Specifies horizontal placement at six options based on shape (text box), text area, and the text within the text box at **Left**, **Center**, and **Right** positions.

Vertical

Specifies vertical placement at 10 options based on shape (text box), text area, and the text within the text box at **Top**, **Center** and **Bottom** positions.

Text Alignment

Specifies the placement of formatted text within the shape (text box). You must define horizontal and vertical components together.

Horizontal

Specifies how each paragraph is aligned within the block of formatted text, (**Left**, **Center** and **Right**).

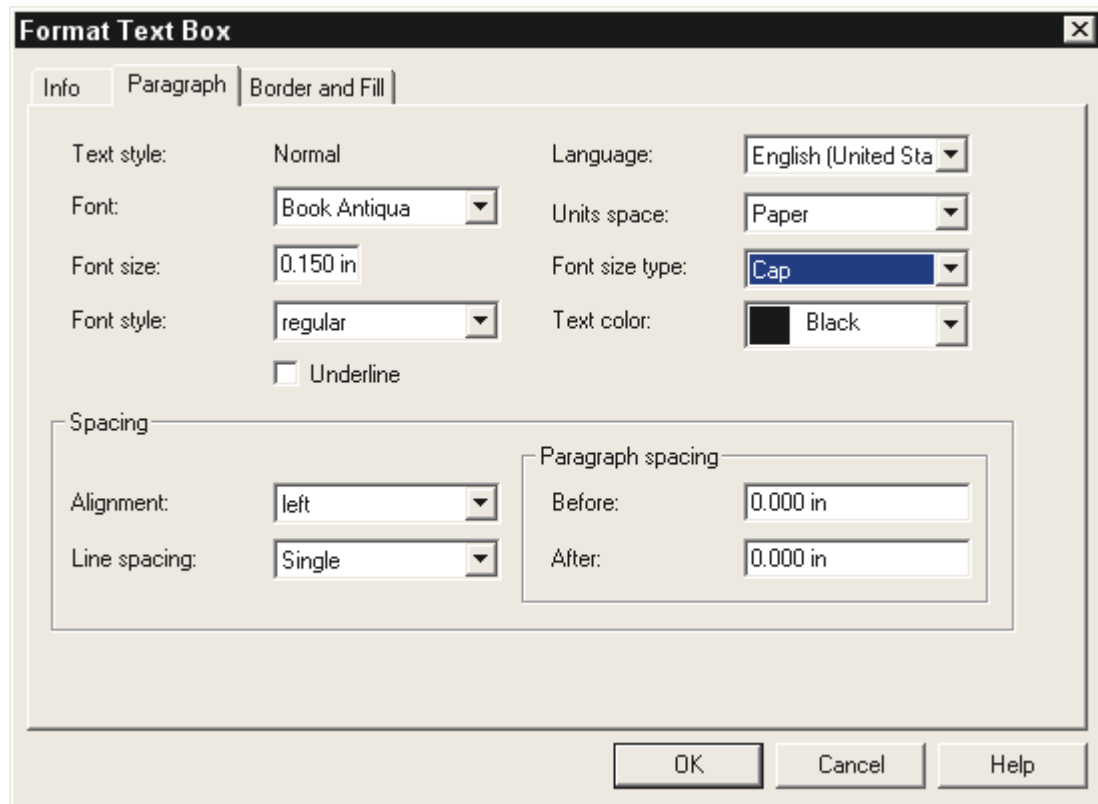
Vertical

Places the block of formatted text at **Top**, **Center**, **Bottom**, or **Justify** area of the text box.

Margins

Sets the distance between the edge of the text box and the edge of the text area. Margins are in the same units as the text box. **Top**, **Bottom**, **Left** and **Right** are the margin options that can be set.

Paragraph Tab



Text Style

Displays the named style used for a text box, this is a static value.

Font

Lists and applies the available fonts.

Font Size

Applies a text size.

Font Style

Lists the available font styles.

Underline

Underlines the text characters.

Language

Lists and applies a language for text in a text box.

Units space

Emphasizes the text choice made on the **Info** tab for this setting. It is Read-Only in the **Text Box Properties** dialog box.

Font size type

Specify the method for measuring font size: **Ascender**, **Cap**, or **Tile**.

Text Color

Sets the color of the font.

Spacing

Determines the amount of space between lines and paragraphs in a text box.

Alignment

Specifies alignment for each paragraph within the block of formatted text. Values for horizontal alignment include **Left**, **Center**, and **Right**.

Line spacing

Lists and applies the amount of vertical space between lines of text.

Single sets the line spacing for each line to display the largest font in the line.

1.5 sets the line space for the line to one-and-a-half that of single lines.

Double sets the line spacing for the line to twice that of single lines.

Paragraph Spacing

Specifies the amount of space before and after paragraphs. No spacing exists before the first paragraph and or after the last paragraph.

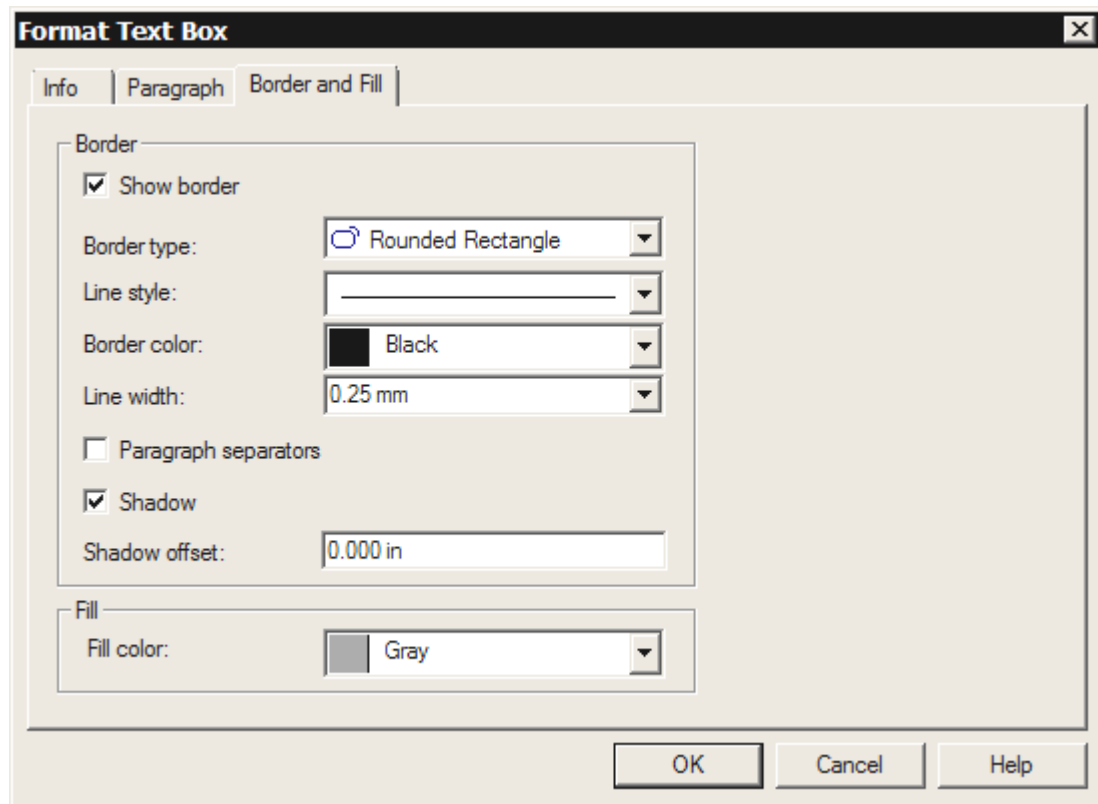
Before

Adds space before a paragraph. This value is set in the current-working units, not in points.

After

Adds space after a paragraph. This value is set in the current-working units, not in points. You can set the units using **Units** on the **Format** menu.

Border and Fill Tab



Border

Specifies the settings for the appearance of a border around a text box.

Show border

Determines whether to display a border outline around the text box.

Border type

Sets the shape of the border around a text box. You can choose from a selection of common geometric shapes.

Line style

Sets the line style for the text box.

Border color

Sets the color of the border.

Line width

Specifies the width of the border in paper or world units of the text box.

Paragraph separators

Places a horizontal line between each paragraph in a text box. This option is only available if **Show Border** is selected. A paragraph is defined by a carriage return within the text body. Using the carriage return as the sole means to create a new line of text will result in a text object with many linear separators.

Shadow

Places a shadow on the borders of text in a text box.

Shadow offset

Specifies a distance, in paper or world units, to offset the shadow of the text box.

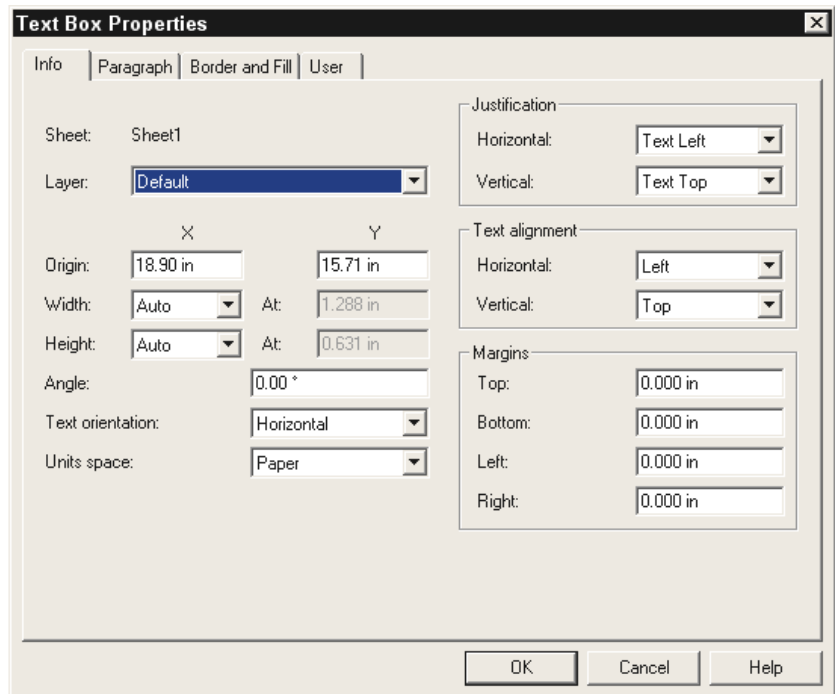
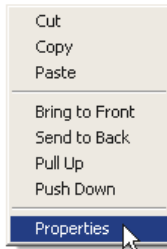
Fill color

Sets the display of the color for a solid fill within the border.

Additional Text Box Properties

Choose a text box using the **Select Tool** and select the **Properties** command, which is activated by using the right-click short-cut menu. This will open the **Text Box Properties** dialog box, which will allow the user to view or modify text box information, behavior and appearance.

Unit 902



Opening the **Text Box Properties** dialog box, you have access to an additional six settings that are not available when setting up a text format or style. See below:

Sheet

Identifies which drawing sheet the text is on.

Layer

Shows the layer that the text was placed on.

Origin

Specifies the coordinates, or location, of the text box origin along the **x** and **y** axes.

Width and Height

Sets the width and height of the text box to **At Least** (*at least* the value you define), **Auto** (the text drives the width or height), or **Exactly** (*exactly* at the value you specify).

Angle

Sets the angle of the text box.

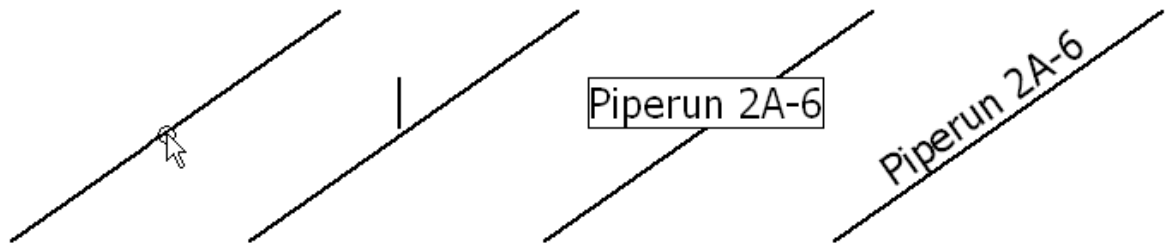
Text Labels

Text labels are associated with an element or object. If the element or object is moved, the label moves with it. If it is deleted, the label will also be erased.

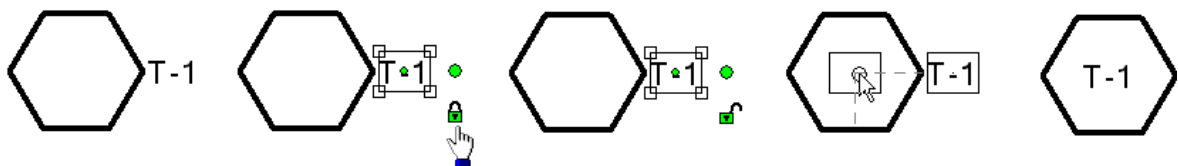
The default position of the label is top center on all elements, except for closed elements, such as a circle or rectangle. The default position on closed elements is the center of the element. A label will orient itself along linear graphic objects.

To create a text label

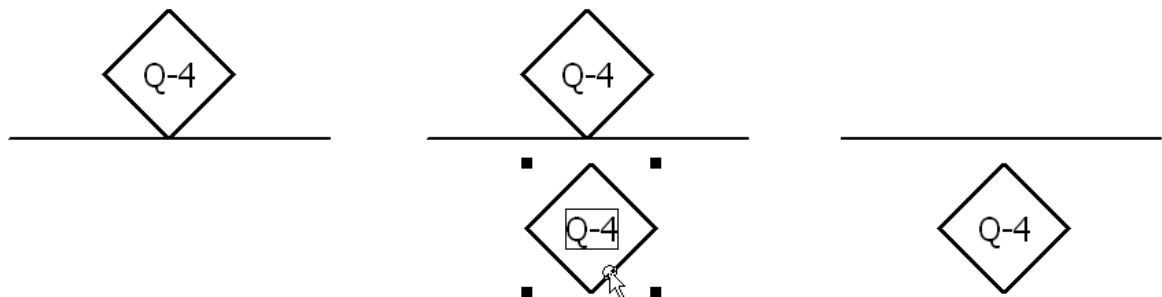
Double-click an element or object. A small, blinking pointer will appear near the object. Set desired text settings such as font and font size in the **Text Box** ribbon bar. Type the text in the label. Right-click the mouse outside the label to complete it.



To reposition only the text label, Click on the lock handle to unlock it. Then, drag the label to the new location. When the label is in its new location, right-click the mouse to clear-out the drag operation.



The text label is still associated to the element or object. If the graphic object is moved to a new location, the text label moves with it maintaining its new relative position.

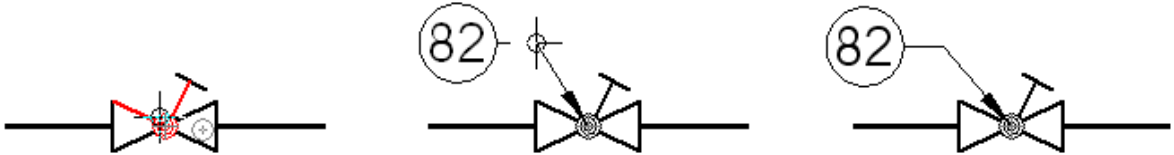


Balloon

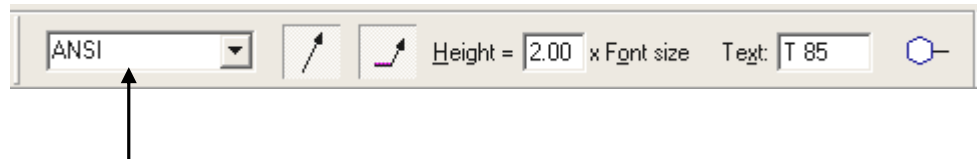


This tool, found on the **Dimension** toolbar, places a balloon containing text. You can use balloons to refer to an object or a point in free space.

To begin, click where you want to place the terminator end of the leader. The default terminator used when an element is selected is an arrow. Place where you want to place the balloon end of the leader to complete the command.



The **Balloon** ribbon bar is used to adjust the shape of the balloon and the text within the balloon.



Dimension Style

Lists and applies the available styles.



Leader

Sets or hides the display of a leader line from the balloon.



Break Line

Places a horizontal break line into the leader.

Height

Specifies the height of the balloon. The actual height of the balloon is the value you enter multiplied by the dimension text height.

Text

Specifies the text note you want shown inside the balloon.

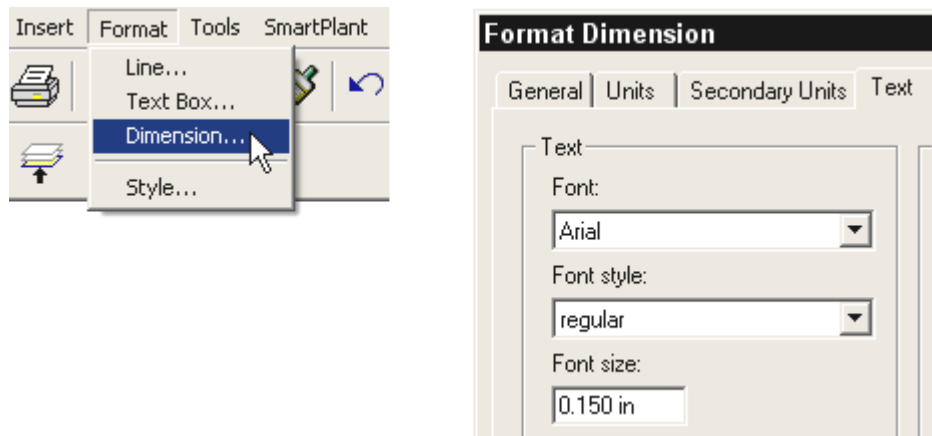


Shape

Specifies which balloon shape you want from the list of available shapes.

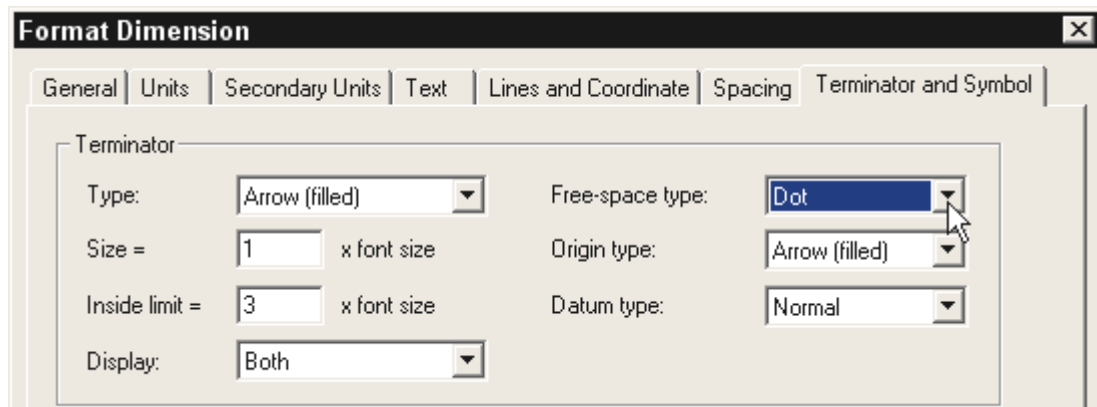
Balloon Text Settings

The balloon font, font style, and font size are controlled from the **Text** tab of the **Format Dimension** dialog box.



Balloon Terminator Settings

The balloon terminator settings are controlled with the **Terminator and Symbol** tab of the **Format Dimension** dialog box.



NOTE: The default free space terminator is a dot.

Other Balloon Settings

The color of a balloon is controlled by the **Driven dimension** color setting on the **General** tab in the **Format Dimension** dialog box.

Finally, the break line of a balloon is controlled from the **Break line** setting on the **Lines and Coordinates** tab of the **Format Dimension** dialog box.

Leader



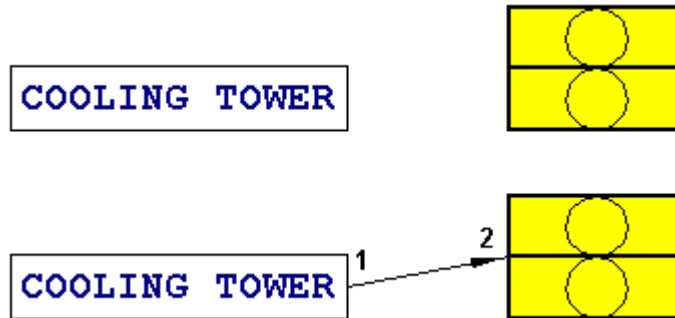
The **Leader** command adds a leader to an annotation or to another leader. You can place either end of the leader first: the annotation end or the terminator end. You can attach the annotation end of a leader to one of the following:

- Text box
- Balloon
- Another leader on a dimension or annotation

You can place the terminator end of a leader in one of the following ways:

- Attach it to an element
- Place it in free space

To add a leader that runs from a text box to an object in the drawing, start the leader by clicking somewhere along the text box. Complete the process by clicking to attach the terminator end of the leader to the object.



The **Leader** ribbon bar is used to set the dimension style and toggle the leader options on and off.



Dimension Style

Lists and applies the available styles.



Break Line

Displays a horizontal break line at the notation end of a leader.



Automatic Shape Connection Behavior

Specifies automatic, shape-changing connection point behavior for the notation (end) reference. If the option is toggled on, the notation end of the leader uses information from the notation object in combination with the current leader position to determine the attachment point and break line direction. If the option is not toggled on, the notation end of the leader attaches to the point on the notation object that was selected when the leader was created, and the break line direction will not automatically break away from the notation object.



Break Line Along

If the option is selected, the break line displays at an angle along the notation object according to information provided by the object. For example, if a text box is at an angle, the break line displays parallel to the text box at the same angle. If the option is not selected, the break line remains in a horizontal position regardless of the angle of the notation object.



Notation Object Attachment Lock

Locks the notation end of the leader to the notation object so that any drag/modify operations on the leader will not detach it from the object.

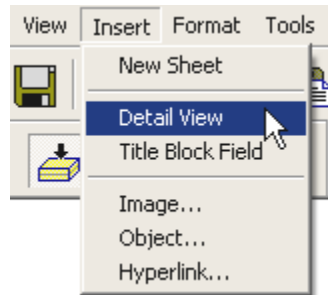


Graphic Object Attachment Lock

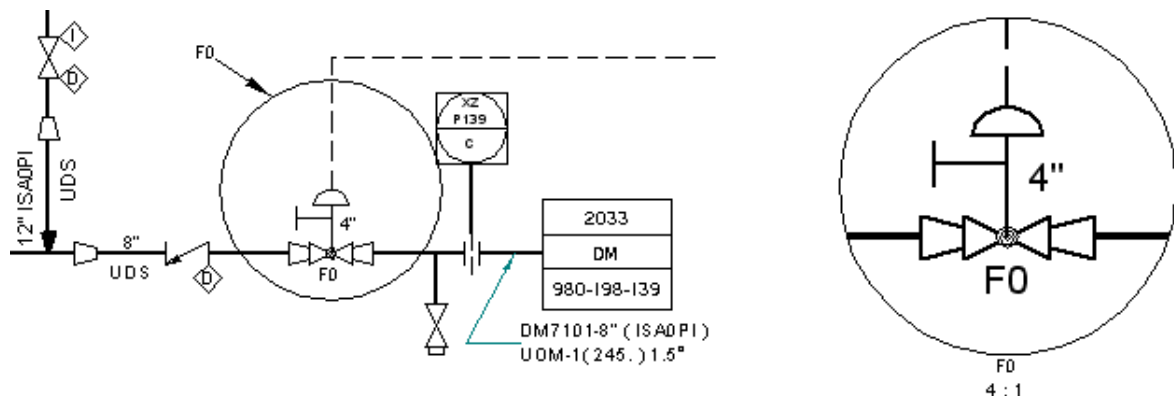
Locks the graphic end of the leader to the graphic object so that any drag/modify operations on the leader will not detach it from the object.

Detail View

The **Detail View** is designed to allow you to enlarge a portion of the data displayed in the SmartSketch document. It can place either a circular or rectangular frame into your drawing containing the desired information.



The **Detail View** command is available through the **Insert** menu. Or, you can add an icon to a toolbar by using the **Tools >Customize** command. Select the **Insert** category to access the icon and drag it to the preferred toolbar.

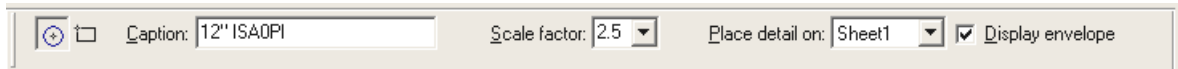


The detail view may be moved to a new location at any time, and the labels may be changed anytime.

The elements displayed in a detail view cannot be modified. However, if the original is altered, the detail view will be updated when the screen is repainted.

The labels and envelope may be deleted without effecting the detail view, but if the detail view itself is deleted all components in the drawing associated with it will be removed from display as well.

A **Detail View** is a **SmartFrame**, (a border for an object that you insert or paste into SmartSketch). The **Properties** option is accessed from the right-click short-cut menu that appears when you select a **Detail View** with the **Select Tool**. The **SmartFrame Properties** dialog box will allow you to modify the border settings, add attribute settings as well as change the layer, the coordinate position within the drawing, and set the display scale of the detail view.



The **Insert Detail View** ribbon bar places a detail view based on graphics inside a user-defined envelope.



Circular detail

Places a detail with a circular envelope and detail view shape.



Rectangular detail

Places a detail with a rectangular envelope and detail view shape.

Caption

Specifies the label for the envelope and detail.

Scale Factor

Specifies the factor to which the detail view is scaled. You can either select a value from the drop-down list, or type a value of your own.

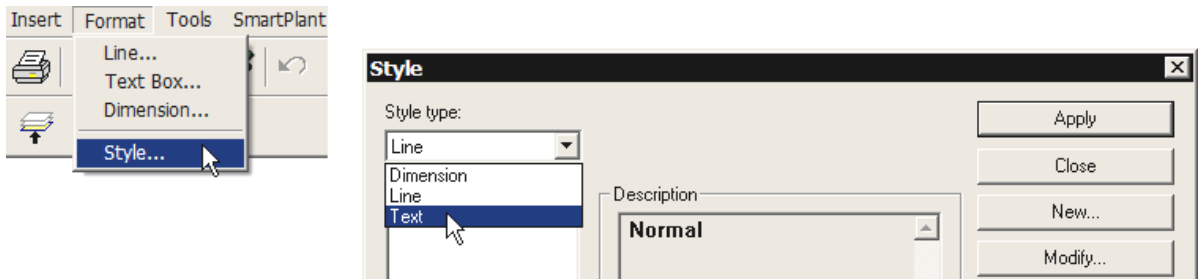
Place detail on

Specifies the sheet in the document in which the detail is to be placed.

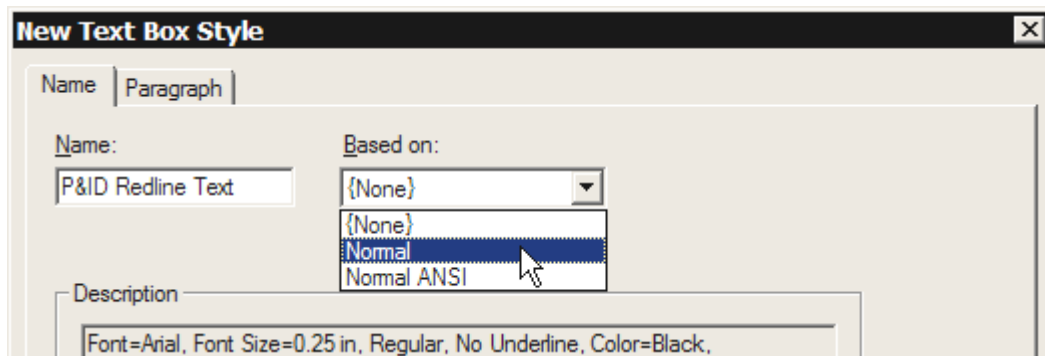
Display envelope

Toggles on or off the display of the detail envelope.

Custom Text Styles



For more recurrent format settings, remember that you can create unique text box styles with **Style** on the **Format** pull down menu. However, you can only set data available on the **Paragraph** tab with the **New Text Box Style** dialog box.



Name Tab

Names a style when you create or modify a style. This tab appears only when you click **New** or **Modify** on the **Styles** dialog box.

Name:

Style names can contain up to 253 characters (including aliases and separators) and can include any combination of characters and spaces, except the backslash character (\), semicolon (;), and braces ({}). Style names are case sensitive.

Based On:

Displays the name of the style that the current or new style is based on.

Description:

Displays a description of the formatting options.

SECTION 12

Basic Symbol Creation

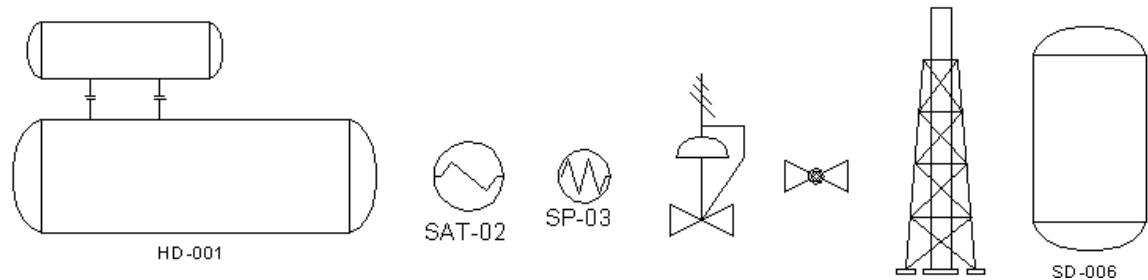
A symbol is an element or group of elements combined into one document. Symbols can be easily recalled and placed in a document, and then manipulated as a single element. Symbols reduce repetitive drawing and encourage standardization.

Topics

- Create symbols
- Place symbols
- Scale, rotate, and mirror symbols with symbol handles
- Change symbol behavior while placing symbols
- Edit symbols

Symbols Overview

A frequently used drawing object can be stored in one document and placed in other documents at a scale, position, and orientation that you define. Graphic data used in this way is called a symbol. SmartSketch includes dozens of application or designated industry solution templates with over 7,500 industry specific symbols. For most standard templates, the **Symbol Explorer** automatically opens and displays a default directory for the symbols that would be appropriate to use with that template.



Symbols increase drawing productivity because they allow you to access existing graphic data quickly and easily. With a symbol, you can place graphic information repeatedly without re-creating it. Symbols save you time by eliminating the need to re-create information, as well as help you maintain accurate graphic data throughout a project. Symbols are contained in documents with a .sym extension.

NOTE: The **Symbol Explorer** is discussed at length in Lesson 1 *Starting SmartSketch*.

Creating Symbols




A symbol is a SmartSketch document with a **.sym** extension. You can create a symbol by selecting the desired geometry and clicking the **Create Symbol** button on the **Draw** toolbar.

Default Behaviors


The default behaviors for symbols that you create are:

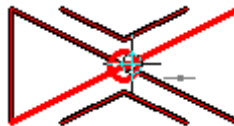
- 90° rotation angles. You can press the **left** and **right** arrow keys on the keyboard to rotate symbol 90° while placing.
- One drag point at the origin point defined when symbol is created.
- Scale, mirror, and rotate handles display when symbol has been selected.
- Will **not** automatically align or lock to other objects.

Steps to create a symbol

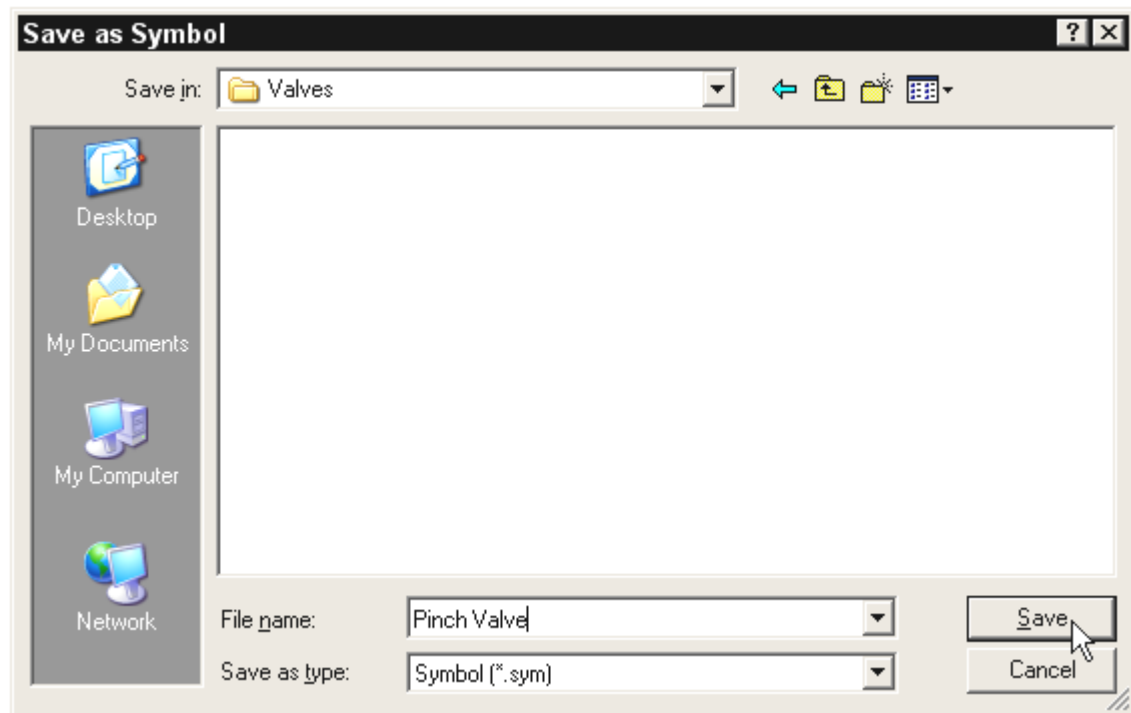
1. Using the **Select Tool** , pick the elements that the symbol is to be made from.



2. On the **Draw** toolbar, click the **Create Symbol**  button.
3. Click a point on the drawing sheet to define the origin of the symbol.



4. In the **Save As** dialog box, select the directory where you want to save the symbol.
5. Type the name that you want for the symbol. The software saves the document with a **.sym** extension. Click the **OK** button.

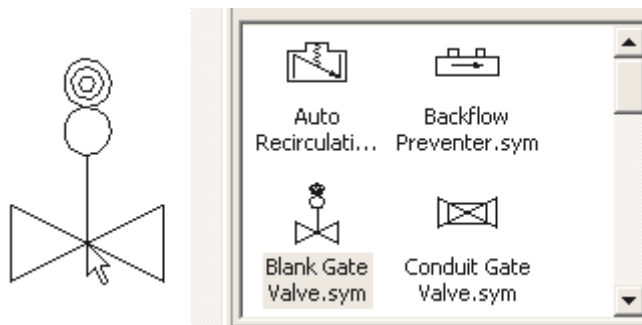


Placing Symbols

To place a symbol, you can use **click** placement or **drag** the symbol from the **Symbol Explorer** or **Windows Explorer** into the current document. When you select a symbol in the **Symbol Explorer**, the pointer attaches to the origin of the symbol by default. The origin was defined when the symbol was originally created.

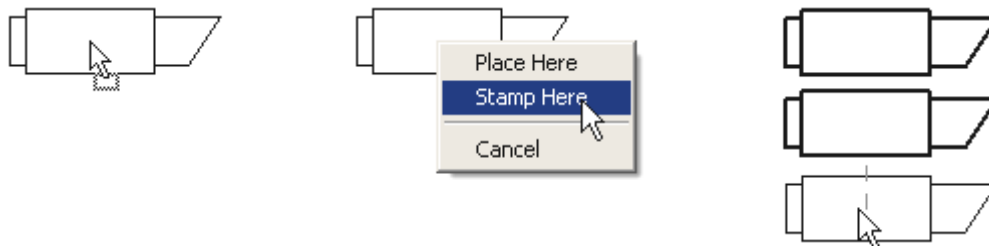
Single-Click Placement

For the click placement method, you can only use the **left** mouse button. Select the desired symbol with the mouse by clicking it with the left mouse button a single time in the **Symbol Explorer**. Move the cursor to the desired position in the drawing and click to place the symbol. The symbol will remain active until the right mouse button is selected or another symbol is chosen from the **Symbol Explorer**.



Stamp method

Select a symbol with the **right** mouse button, and while holding it down, drag the symbol into the document. Release the right mouse button in the desired location of the first symbol. On the displayed shortcut menu, click **Stamp Here**. Click another point in the document (with the left button) to place a copy of the symbol. You can click as many points as you want to place multiple copies of the same symbol. Click the right mouse button when you are finished placing the symbol.

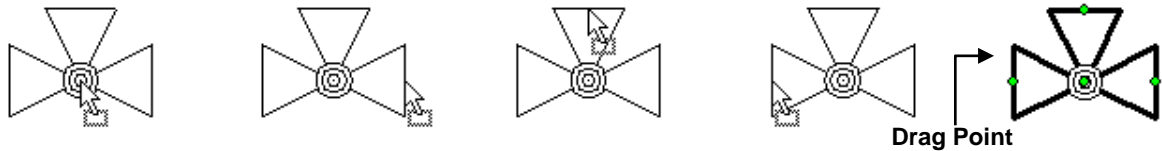


NOTE: If you decide that you do not want to place several copies of a symbol, you can click **Place Here** on the shortcut menu. This places only one instance of the symbol.

Drag Placement Behavior

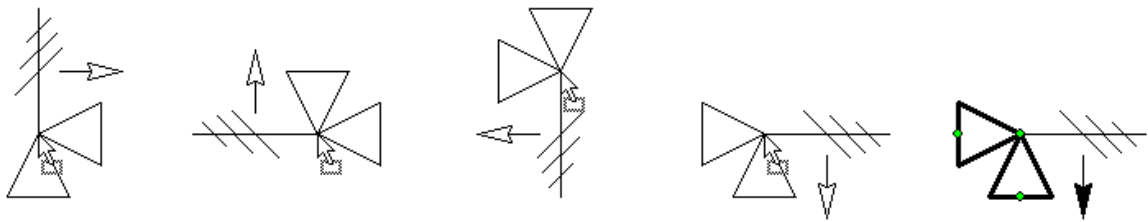
Drag Points

A drag point is the point to which the mouse cursor attaches to a symbol for dragging and dropping. A symbol can have multiple drag points, which can be toggled using the **up** and **down** arrow keys as the symbol is being dragged into the drawing. The small green circles represent drag points. You can use these points to attach the symbol at a precise point to another symbol or element. The drag points can also be used to move the symbol with precision.



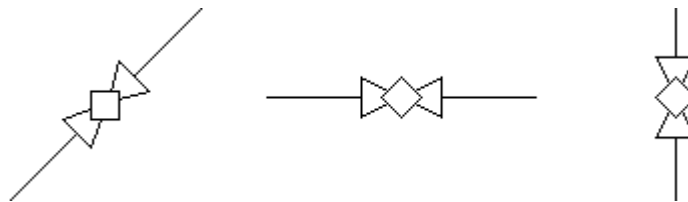
Rotation

When you drag a symbol or align it to an element, you can press the **left** and **right** arrow keys on the keyboard. This rotates the symbol in 90° increments by default. Some symbols are designed to rotate at different increments. A symbol defined with a rotation of 0° will not rotate using the arrow keys.



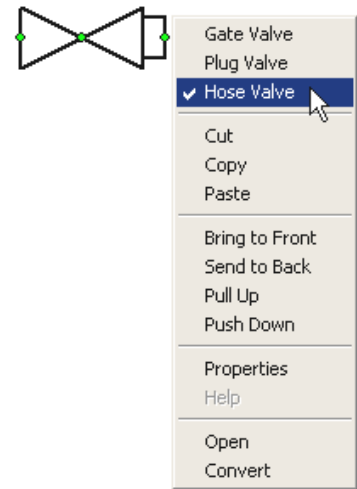
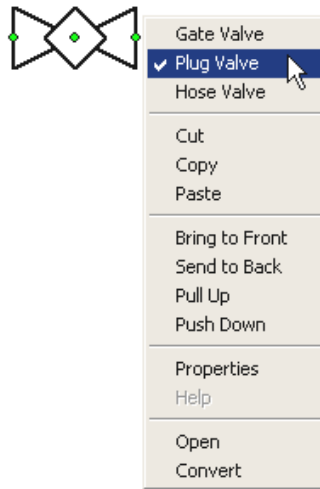
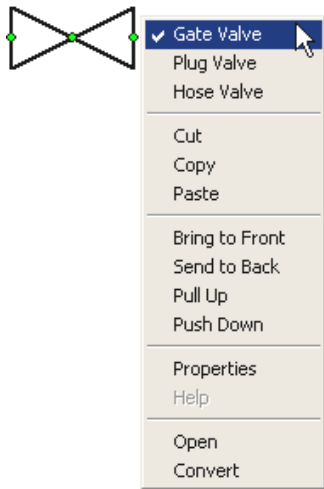
Alignment

Some symbols are created with a special alignment behavior. This behavior allows the symbol to automatically rotate and align itself parallel to another graphical object during drag and drop placement or when the symbol is repositioned after placement.



Multiple Representation

Symbols can be created with multiple graphical representations. After the symbol is placed, a right mouse click will display a menu containing a list of names of each representation available for the symbol. This example illustrates the use of a symbol with multiple representations.

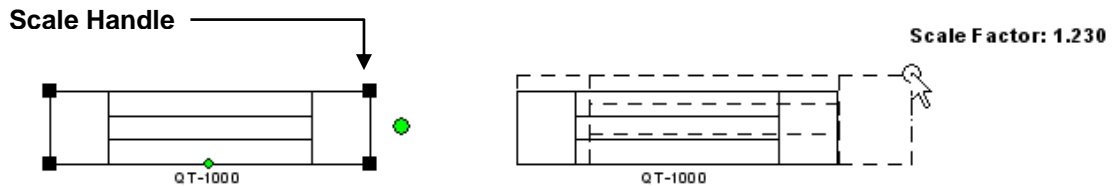


Symbol Handles

When you select a symbol, different types of handles can appear on the symbol.

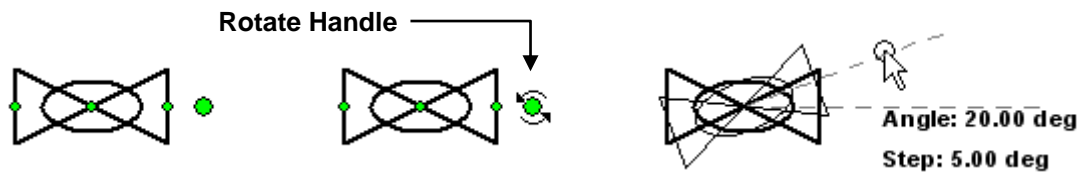
Scale handles

Black solid boxes on the outside corners of the range box represent scale handles. You can drag a scale handle to resize the entire symbol uniformly in the X and Y directions.



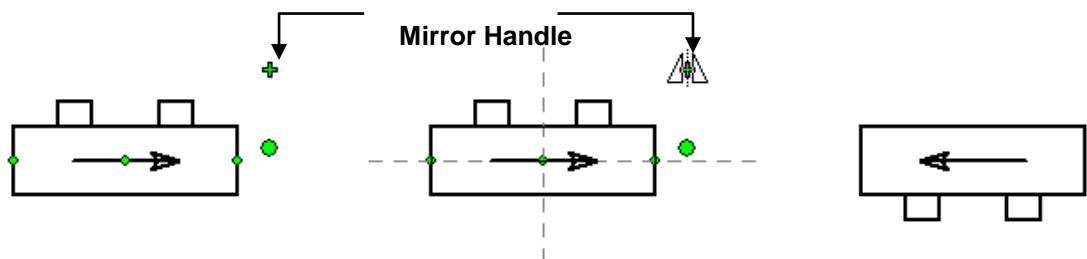
Rotate handle

A large green circle represents a rotate handle. You can drag this handle to rotate the entire symbol. The pointer displays specific increments while you rotate the symbol.



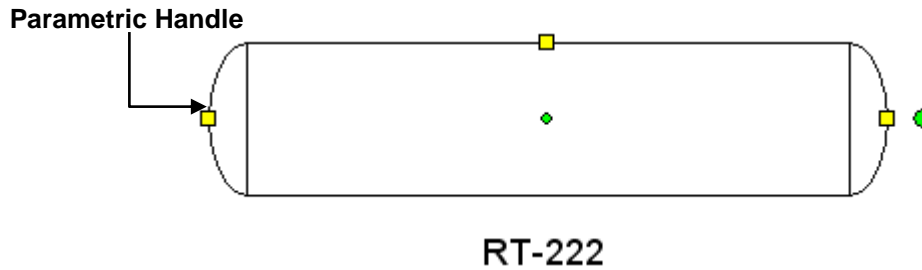
Mirror handle

A green plus sign represents a mirror handle. You can drag the mirror handle across either the vertical or horizontal mirror line to mirror the symbol.



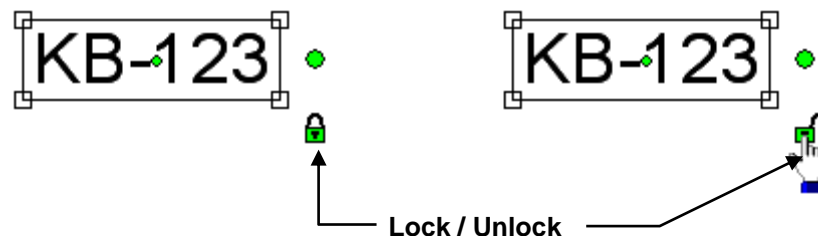
Parametric handles

Yellow boxes in the middle of the range box represent standard parametric handles. You can drag a parametric handle to resize part of the symbol. Parametric handles allow you to scale the symbol in a manner defined when the symbol was originally created.



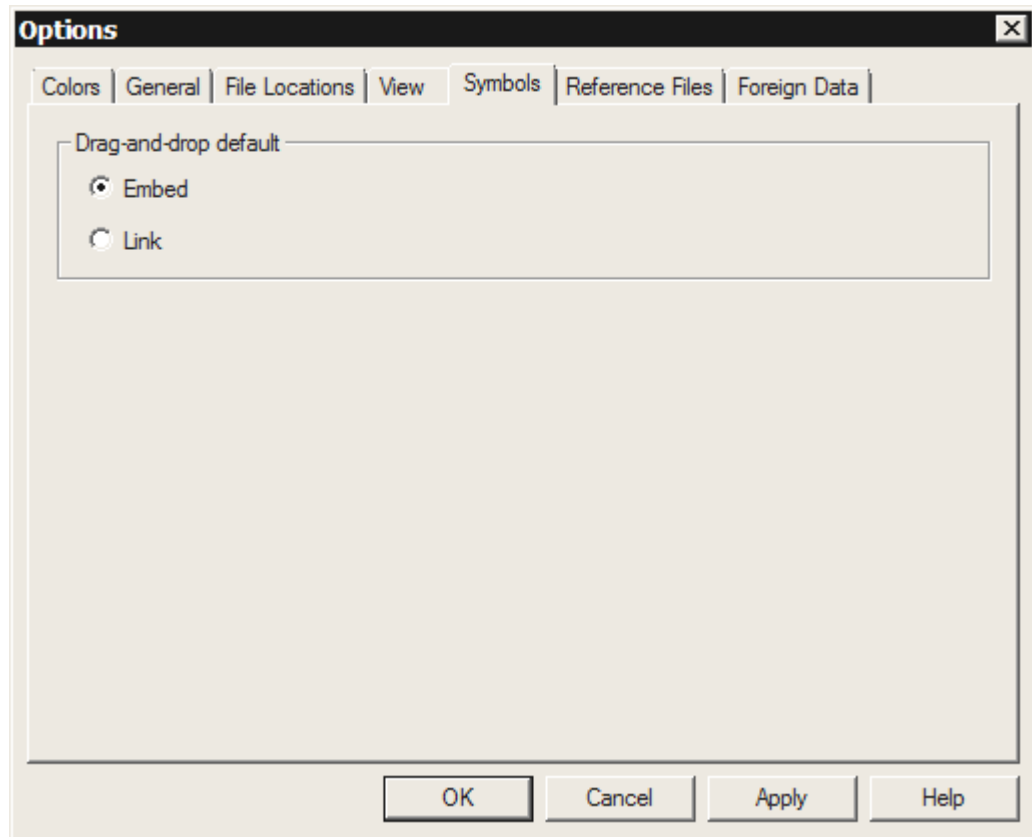
Lock handle

A small green lock represents a lock handle. You can click the lock handle to unlock the symbol. Unlocking allows you to relocate a symbol away from the point on the element to which it first was associated.



Symbol Options

Before placing a symbol, you can set an option to link or embed the symbol when you drag and drop it onto the drawing sheet. You can do this by clicking the **Options** command on the **Tools** menu and then setting the option you want on the **Symbols** tab.



The **Symbols** tab of the **Options** dialog box determines the default actions when you drag a symbol into the document. You can override the default setting, - **Embed**, when you drag a symbol by pressing both the <Ctrl> + <Shift> keys to link the symbol.

Embed

Sets the default action so that the symbol embeds when you drag it on the drawing sheet. Embedding the symbol means that the software places a copy of the symbol in the document. If you edit one instance of the embedded symbol in a document, all copies of that symbol within the current document reflect those changes.

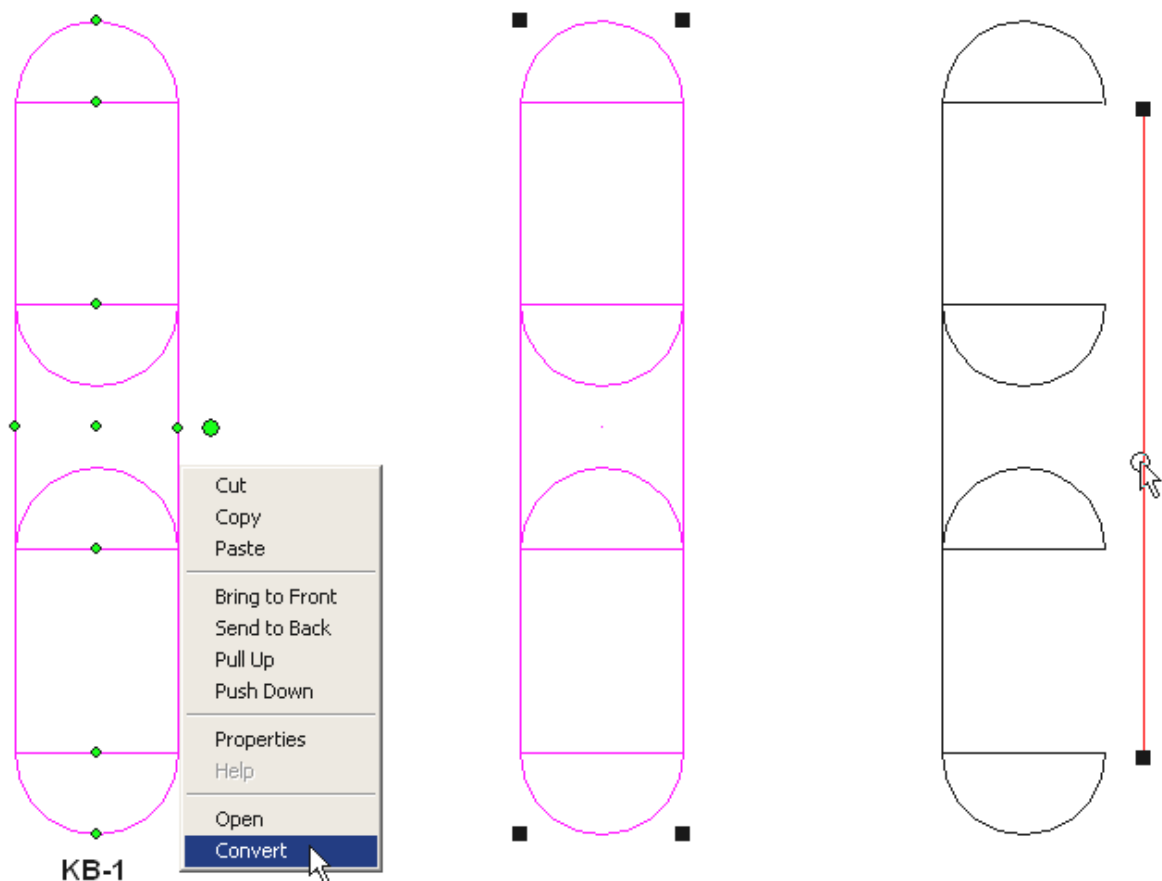
Link

Sets the default action so that the symbol is linked when you drag it into the document. Linking the symbol allows you to edit the .sym file. The symbol that you placed on the drawing sheet will then update automatically. If you edit the symbol inside the active document, those changes are saved in the .sym document on your computer.

Editing Symbols

If you want to quickly edit a symbol, you can open double-click any symbol in the **Symbol Explorer**. The way you edit a symbol depends on whether the symbol is linked or embedded.

To convert a symbol to individual elements, click the right-mouse button on the symbol and choose **Convert** from the shortcut menu. This will convert the symbol into a group of graphic objects.



On the **Change** toolbar, click the **Ungroup**  button to separate the elements so that you can edit them individually.

The symbol has now been converted to individual elements and can be edited like any other drawing element.

SECTION 13

Attributes

Attributes can help you work more efficiently by allowing you to enter and edit values for a symbol, object, or element. Attributes can include such items as the manufacturer, price, size, voltage, and other data. Attributes include user-defined properties and parameters. You can display attributes in the **Attribute Viewer**, normally located at the bottom of the Symbol Explorer. But Attribute Viewer can also be undocked and sized and positioned to best meet your requirements.

Topics

- Display attributes
- Edit symbols in the Attribute Viewer
- Adding User Properties
- Create a report using the delivered report symbols

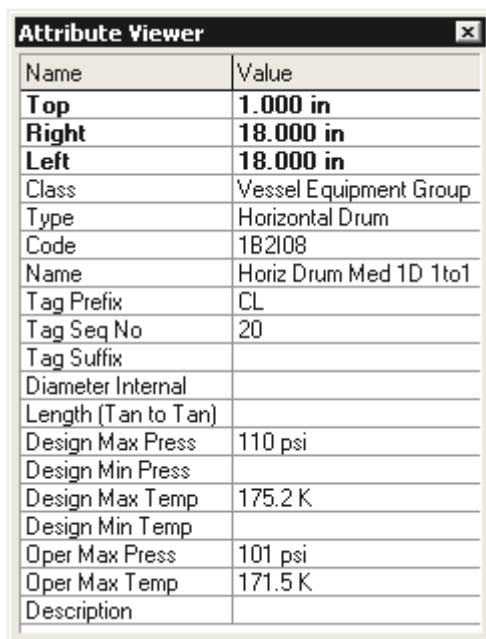
Display Attributes

Attributes

An attribute is a user-defined property or parameter assigned to an object or document. Attributes can help you work more efficiently by allowing you to enter and edit values for a symbol, or object. Attributes can include a great variety of information that is useful to the designer or anyone reviewing the document.

User-defined properties are attributes assigned to an object that define a characteristic about that object. User-defined properties are usually in the form of a text notation, such as cost, manufacturer, and location. You can change the user-defined properties without affecting the appearance of the object. User-defined properties can be specified on the **User** tab of the **Properties** dialog box.

A parameter is an attribute assigned to a graphic object, usually a numerical value representing a dimension. It is a mechanism for changing a property. A parameter refers to variables in a symbol document that drive dimensional values. Parameters appear as bold characters in the **Attribute Viewer**.



Attribute Viewer	
Name	Value
Top	1.000 in
Right	18.000 in
Left	18.000 in
Class	Vessel Equipment Group
Type	Horizontal Drum
Code	182108
Name	Horiz Drum Med 1D 1to1
Tag Prefix	CL
Tag Seq No	20
Tag Suffix	
Diameter Internal	
Length (Tan to Tan)	
Design Max Press	110 psi
Design Min Press	
Design Max Temp	175.2 K
Design Min Temp	
Oper Max Press	101 psi
Oper Max Temp	171.5 K
Description	

Attribute Viewer

The **Attribute Viewer** displays the user-defined properties and parameters of a selected symbol or object. The viewer displays two columns, one for the name of the attribute and one for the value. You can edit the values of the properties or parameters. User-defined properties appear in plain text. Parameters will appear in **bold** text.

Important:

- You cannot edit the names of attributes; you can only edit their values. The name can be changed with the **User** tab of the **Properties** dialog box.
- User-defined properties also appear on the **User** tab of the **Properties** dialog box. Parameters appear only in the **Attribute Viewer**.
- You can edit the parameter values of a parametric symbol to automatically change the dimensions of the symbol.
- If multiple symbols containing the same attributes are selected, their common attributes will be displayed in the **Attribute Viewer** for editing. Any changes made to one symbol's attributes are applied to all selected symbols.
- You may edit displayed attributes before and during symbol placement.
- Use the **Up** and **Down** arrow keys (or the **Enter** key) to scroll through **Attribute Viewer** values.

Displaying the Attribute Viewer

If the **Symbol Explorer** window is open, you can display or hide the **Attribute Viewer**. You can drag the splitter bar between the **Symbol Explorer** and the **Attribute Viewer** to resize or hide the viewer, while it is docked with the **Symbol Explorer**. However, the **Attribute Viewer** does not have to be docked. It can “float” in the drawing space of SmartSketch.

To display or hide the viewer, place the pointer over a toolbar at the top of the document window and right-click. On the shortcut menu, select or clear the **Attribute Viewer** option.



Attribute Viewer	
Name	Value
Class	Mechanical Equipment Group
Type	Pump
Code	1D4A24
Name	Gear Pump
Tag Prefix	GP
Tag Seq No	15
Tag Suffix	A
Design Max Flow Rate	475 gpm
Design Max Sp Grav	.90
Design Max Vapor Press	4000 lbf/in ²
Design Max Viscosity	25,000 SSU
Description	Super Gear Pump

To resize the **Attribute Viewer**, you can double-click the title bar of the viewer to undock it. Then drag the sides of the viewer to resize.

Working with Attributes

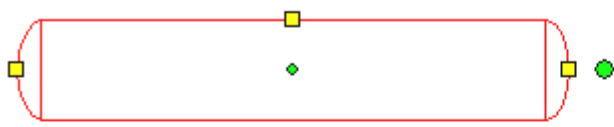
If the selected item has assigned attributes, the **Attribute Viewer** displays its properties and parameters. Select a symbol, or other object and click a cell in the **Attribute Viewer**.

Attribute Viewer	
Name	Value
Right	0.800 in
Top	2.000 in
Bottom	4.500 in
Class	Vessel Equipment Group
Type	Vertical Drum
Code	1B2G04
Name	Vert Drum Narrow 1D 2to1
Tag Prefix	KB
Tag Seq No	1
Tag Suffix	
Diameter Internal	48 in
Length (Tan To Tan)	107 in
Design Max Press	55 psi
Design Min Press	35 psi
Design Max Temp	
Design Min Temp	
Oper Max Press	45 psi
Oper Max Temp	
Description	

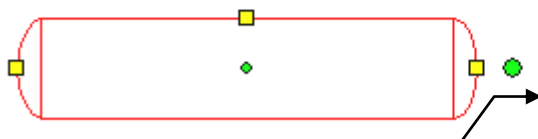
Attribute Viewer	
Name	Value
Right	0.800 in
Top	2.000 in
Bottom	4.500 in
Class	Vessel Equipment Group
Type	Vertical Drum
Code	1B2G04
Name	Vert Drum Narrow 1D 2to1
Tag Prefix	KB
Tag Seq No	1
Tag Suffix	
Diameter Internal	48 in
Length (Tan To Tan)	107 in
Design Max Press	55 psi
Design Min Press	35 psi
Design Max Temp	
Design Min Temp	
Oper Max Press	45 psi
Oper Max Temp	
Description	Downstroke Baler

Type new information or change existing information. Either pressing the **Enter** button on the keyboard or moving the cursor out of the **Attribute Viewer** will complete the change.

Follow the same steps in order to change parametric values. After you select a value field that has parametric data in it, enter a new value and notice the change in the graphics.



Attribute Viewer	
Name	Value
Top	0.800 in
Right	4.500 in
Left	4.500 in
Class	Vessel Equipment Group
Type	Horizontal Drum
Code	1B2I04

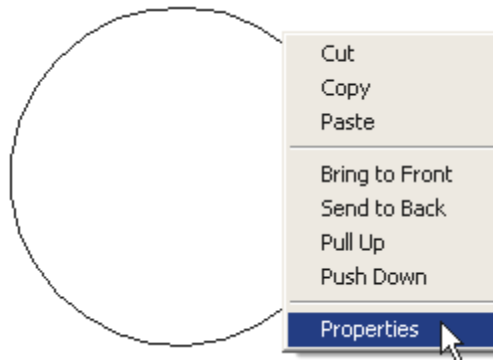


Attribute Viewer	
Name	Value
Top	0.800 in
Right	4.500 in
Left	3.0
Class	Vessel Equipment Group
Type	Horizontal Drum
Code	1B2I04

Changed Value

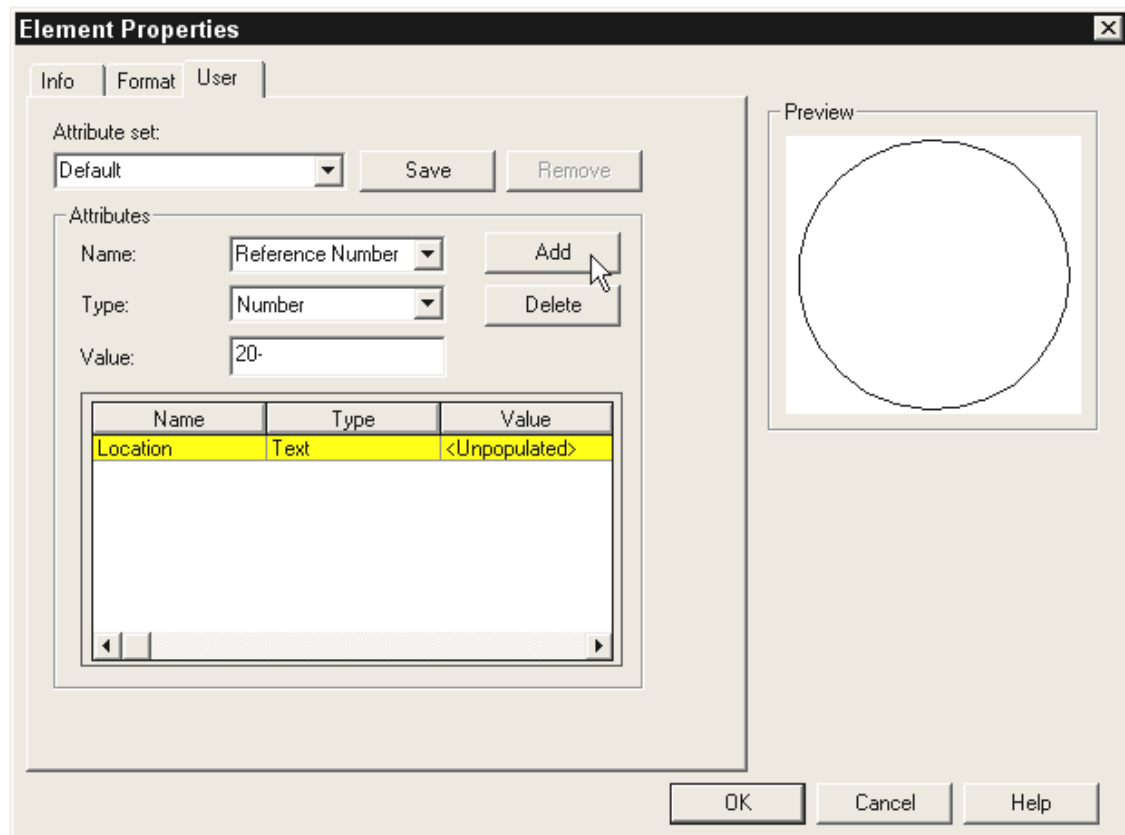
Adding User Properties

Adding user-defined properties is how you create your own attributes for graphic objects. Start by select the object you want to add user-defined properties to and right-click on the graphic object. Select **Properties** from the shortcut menu.



This will open the **Element Properties** dialog box, click on the **User** tab. In the **Attributes** section key in a name, select a type, and enter a value; to finish the attribute, press the **Add** button.

The attribute(s) must appear in the table near the bottom before they can be applied to an element. Continue adding attributes as needed, then select the **OK** button.

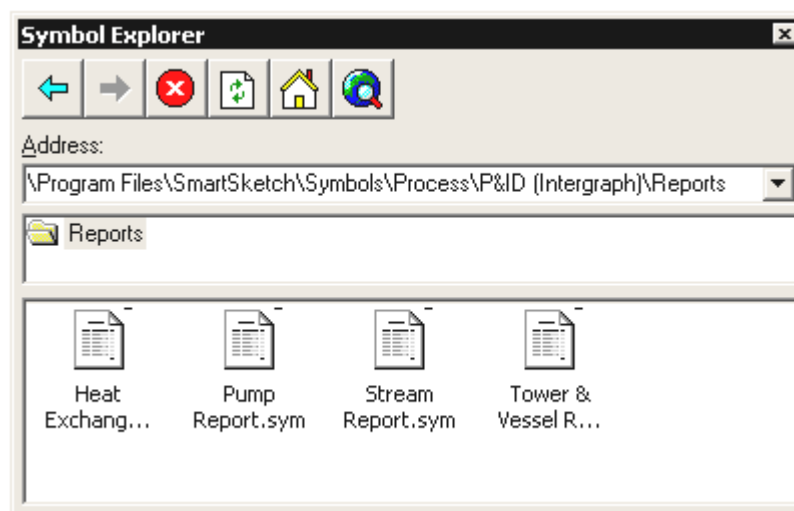


Delivered Reports

SmartSketch delivers report symbols that can be dragged into a document to generate a report as an Excel spreadsheet. The reports work with symbols that have specific attributes assigned to them. You must have Microsoft® Excel™ 97 or later installed on your computer.

SmartSketch includes several sample reports. When you create a document from a template that has reports associated with specific symbol directories, you will have access to those reports.

The following Process templates contain **Reports** folders: P&ID(Intergraph); P&ID (Intergraph)Split on Drop; PFD (ANSI); PFD (Intergraph); and PFD (ISO). The Reports associated with the Process templates are: heat exchanger report, pump report, stream report, tower and vessel report. (See below.)

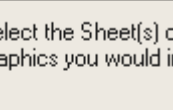


NOTE: There are also delivered reports associated with Architecture templates, Network Diagramming templates, Office Layout templates, and AEC\\HVAC templates.

Below is an example of a populated Tower and Vessel Report for a P&ID.

TOWERS & VESSELS						
TAG		B-1	B-2	B-3A	B-3B	K-1
NAME	Barrel Drum	Distillate Vessel	Vessel	Vessel	Vessel	Bubble Cap tray Column
INSIDE DIA		900	1000	1800	1800	1200
LENGTH		1400	1800	3200	3200	9000
MAX PRESSURE		0.3	0.3	0.3	0.3	0.2
MIN PRESSURE						0.1
MAX TEMP		150	150	150	150	250
MIN TEMP						
DESIGN PRESSURE						
DESIGN TEMP						
REMARKS						

The screenshot displays the HYSYS software interface. On the left is a process flow diagram (PFD) showing a distillation column (B-1) with various pumps (P-1A, P-1B) and piping. A red rectangular box is overlaid on the PFD, and a black arrow points from the 'Tower & Vessel R...' report in the file explorer on the right to this box. The file explorer on the right shows a tree structure with 'Labels', 'Piping', and 'Reports' folders. Under 'Reports', there are three report icons: 'Heat Exchang...', 'Stream Report.sym', and 'Tower & Vessel R...'. The 'Tower & Vessel R...' report is highlighted with a black border.



Select Sheet(s)

Select the Sheet(s) containing those graphics you would included in your report.

☐ Sheet1

☒ Sheet2

OK

A zoomed in copy of the table is at the bottom of page

**A zoomed in copy of this report
is at the bottom of page 208.**

Update an existing report

A report is a snapshot of the current information in a document or on a sheet at the moment it is dragged in as a symbol. If the information changes after that point in time, the report may become inaccurate due to those subsequent changes.

To update an existing report in the document you will need to delete the current report and drag in a new copy of the report symbol from the **Symbol Explorer**.

TOWERS & VESSELS						
TAG		B-1	B-2	B-3A	B-3B	K-1
NAME	Barrel Drum	Distillate Vessel	Vessel	Vessel	Vessel	Bubble Cap tray Column
INSIDE DIA		900	1000	1800	1800	1200
LENGTH		1400	1800	3200	3200	9000
MAX PRESSURE		0.3	0.3	0.3	0.3	0.2
MIN PRESSURE						0.1
MAX TEMP		150	150	150	150	250
MIN TEMP						
DESIGN PRESSURE						
DESIGN TEMP						
REMARKS						

Before Update

The report reflects the latest additions and changes.

TOWERS & VESSELS				
TAG	B-1	B-2	B-3	K-1
NAME	Distillate Vessel	Distillate Vessel	Distillate Vessel	Bubble Cap tray Column
INSIDE DIA	900	900	900	1200
LENGTH	1400	1400	1400	9000
MAX PRESSURE	0.3	0.3	0.3	70 psi
MIN PRESSURE				60 psi
MAX TEMP	150	150	150	220 F
MIN TEMP				38 F
DESIGN PRESSURE				68 psi
DESIGN TEMP				200 F
REMARKS				

After Update

SECTION 14

Working with Foreign Data

SmartSketch has the ability to work with different data formats, allowing you to integrate information from several different sources into a SmartSketch document. SmartSketch also has the ability to be exported out to different file formats.

Topics

- Reference files
- Translating Data
- Hyperlinks

Working with Foreign Data

This lesson is designed to be a basic introduction to working with foreign data. An advanced course, *CAD Translator Optimization*, is available which is designed to take a more in depth look at this broad and detailed subject.

There are several ways to work with foreign data — documents that were created using other software applications. The following can be done:

- Display and reference foreign data documents in the desired position and scale within SmartSketch by **Insert >Object** or drag and drop of the foreign document into SmartSketch.
- Translate foreign data into SmartSketch by simply opening the foreign data document with SmartSketch.
- Translate SmartSketch documents into other CAD data formats by selecting the desired format in the **Save As** dialog box.

Microsoft Office Documents

SmartSketch is fully compatible with Microsoft® Office™ software. You can transfer text, numbers, sound bites, or intelligent graphics between documents that were created with SmartSketch and other Microsoft Office applications. You can move whole documents around or just parts of a document that you selected. The information that you move is called an object.

For example, you can use Microsoft® Word™ to create a materials report and then display the report in your SmartSketch document. You can also link a Microsoft® Excel™ spreadsheet to the **Variable Table** in a SmartSketch document. You can then use the value in the spreadsheet to control the dimensions in the SmartSketch drawing. You can even create a document that contains a drawing created by SmartSketch, notes from Microsoft® Word™ and a spreadsheet created by Microsoft® Excel™.

You can move information around with one of many methods:

- **Cutting and pasting** deletes data from one location and places it in another location.
- **Copying and pasting** duplicates the information and places it into a new location. The original information does not change.

Two common methods used to import CAD data

SmartSketch supports opening and saving CAD files in two ways: you can either select the CAD file and translate it into equivalent SmartSketch graphic objects or you can create a reference file from the CAD file. A reference file is any file that is not native to SmartSketch that is linked or embedded and used for reference information.

Object Linking and Embedding (OLE)

SmartSketch supports Object Linking and Embedding (OLE) to provide a flexible and efficient means of inserting and working with data that was created in another application, items known as objects. Part of the power OLE provides is the ability to update objects automatically if they've been modified in their source document, or edit objects in place.

Linking

Linking takes data that is stored in one location (the source document) and places a reference to it in another location (the destination document). When linking data, the document containing the data is not copied into the SmartSketch document but is retrieved directly from its location on disc whenever the SmartSketch file is opened.

When you link an object, SmartSketch stores information about where the object is located. The object is not stored in the SmartSketch file. You can edit a linked object only by opening the source document and making your changes there. When you change the original data, the linked data automatically updates.

Embedding

Embedding takes data from a source document and stores a copy of the information in the destination document. The copy becomes an independent version of the original information. When you make changes to an embedded object, only the copy of the object that is stored in the destination document updates. Although editing the data in the original source document has no effect on the copied version, you can edit the embedded object in-place from within the SmartSketch drawing.

When to Link

Choosing whether to link or embed data depends, in large part, on how you intend to use your SmartSketch drawing. Link is useful when you want to share the same information in many different places and have that information update automatically when the original/source data is edited. Another consideration is file size. Because linking only stores file location information, this method does not significantly increase the size of the SmartSketch drawing.

When to Embed

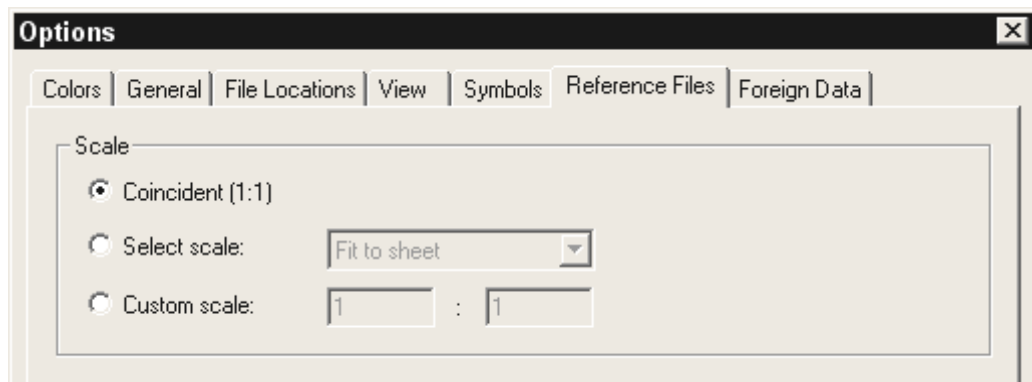
An object should be embedded in a SmartSketch document if the data is to become part of the drawing. Because an embedded object becomes part of the SmartSketch drawing, it will increase the file size. A document containing only embedded data is portable. The single document can be sent to someone who does not have access to the source documents. Embedding information is also a good choice if the information does not require frequent updating.

Reference Data

You can use the following methods to move MicroStation® (.dgn) and/or AutoCAD® (.dwg or .dxf) documents into SmartSketch:

- Cut, Copy and Paste
- Drag and Drop
- Insert> Object

On the **Reference Files** tab of the **Tools> Options** dialog, if **Coincident (1:1)** is selected, the placed object (reference file), is brought in at full scale and coincident to its original coordinate system.



Cut/Copy and Paste

You can move or copy data between a SmartSketch document and other CAD documents using the **Cut**, **Copy**, **Paste**, and **Paste Special** commands on the **Edit** menu. The information is placed in the destination document as an embedded object. The data is placed in the lower left hand corner of the drawing at 0,0.

NOTE: The **Edit → Paste Special** command inserts the **Clipboard** contents into the document using one of the following formats: CAD Drawing, Picture (Metafile), or Bitmap.

Once the data has been pasted into the SmartSketch drawing, you can then edit the embedded object. To edit the object, you must have access to the software that created it and the software must support OLE. If the data is displayed as a static picture or bitmap, it cannot be edited.

Drag and Drop

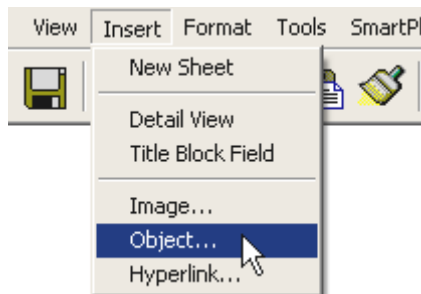
Another method that can be used to access data from other CAD applications is **Drag and Drop**. Using this method, you can drag and drop a document from either the **Windows Explorer** or the **Symbol Explorer** into a SmartSketch drawing.

Using the left-mouse button, drag the document from the **Symbol Explorer** window into the active document. The object is placed with the mouse drop point at the center of the object's range box. Once the object is placed, click outside the **SmartFrame** container to deselect.

The document will be embedded in the drawing by default. For the document to be linked, you must press the **Ctrl + Shift** keys while dragging and dropping. You cannot create a link to a selection in a CAD document. You can only create a link to the entire document.

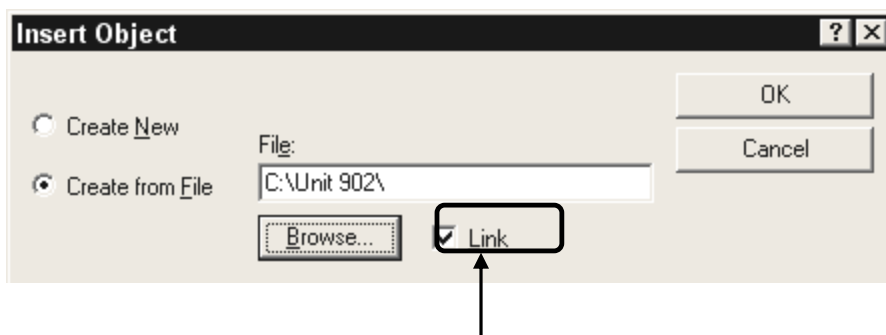
Inserting as an Object

The **Insert → Object** command allows users to insert objects into a SmartSketch document through linking or embedding. This command inserts OLE 2.0-enabled objects, such as Word documents, Excel spreadsheets, or other CAD documents.



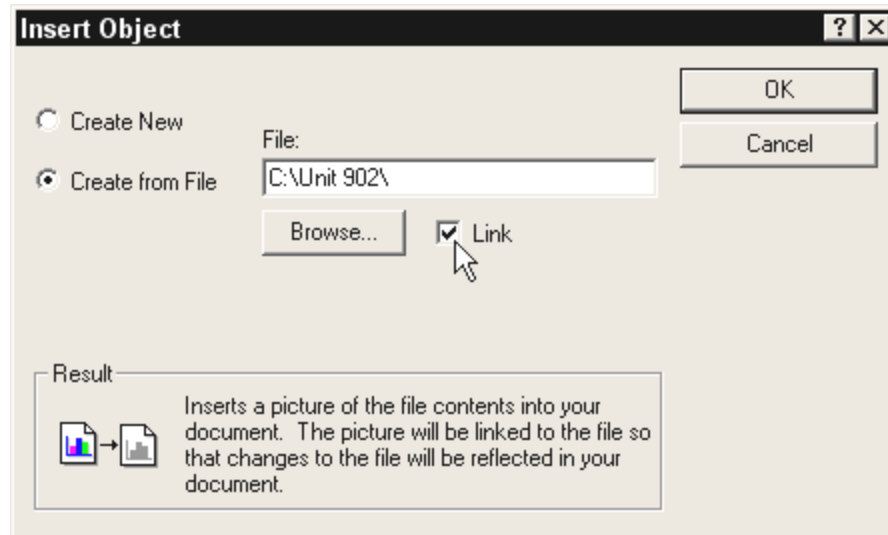
After you insert the object, SmartSketch places a **SmartFrame** border around the object, allowing you to crop it without altering the scale of the object. The software recognizes the geometry of the inserted document. The **SmartFrame** usually appears as a rectangular outline in the document, but it can also have other boundary type shapes. **SmartFrames** provide ways for these objects to be manipulated: including moving, scaling, rotating, mirroring, and cropping. You can use the relationship indicators to draw or create relationships between the geometry of the inserted object and other elements in the SmartSketch document. You can edit the inserted object by double-clicking it.

If the **Link** check box is checked, the file will be linked. If it is not checked, the file is embedded. Also important, the **Create New** option inserts the object as embedded.



Insert a New Object

Select the **Object** command from the **Insert** menu. On the **Insert Object** dialog, click the **Create from File** option. In the **File** field, type the name of the file you want to insert, or click the **Browse** button to change to the directory that contains the file you want to insert.



NOTE: You can specify a default directory for the **File** field or the **Browse** dialog by specifying either a MicroStation or AutoCAD References directory on the **File Locations** tab of the **Tools → Options** dialog.

Once the file has been selected, click the **OK** button. A box the size of the object appears beside the cursor. If you click on the drawing sheet to place the object, the object is placed with the mouse drop point at the center of the object's range box. If you hit the **Esc** key while the box appears beside the cursor, the object is placed in the lower left hand corner of the drawing at **0,0**. If you want to cancel the insert, click the right-mouse button.

Editing Links

The **Edit → Links** command displays information about the links in the active document including the file name, file location, and whether the link is automatically or manually updated. The **Links** dialog box also will allow you to: break a link, reconnect or change a link if the source file is moved to a new location.

Performing Actions on Foreign Data

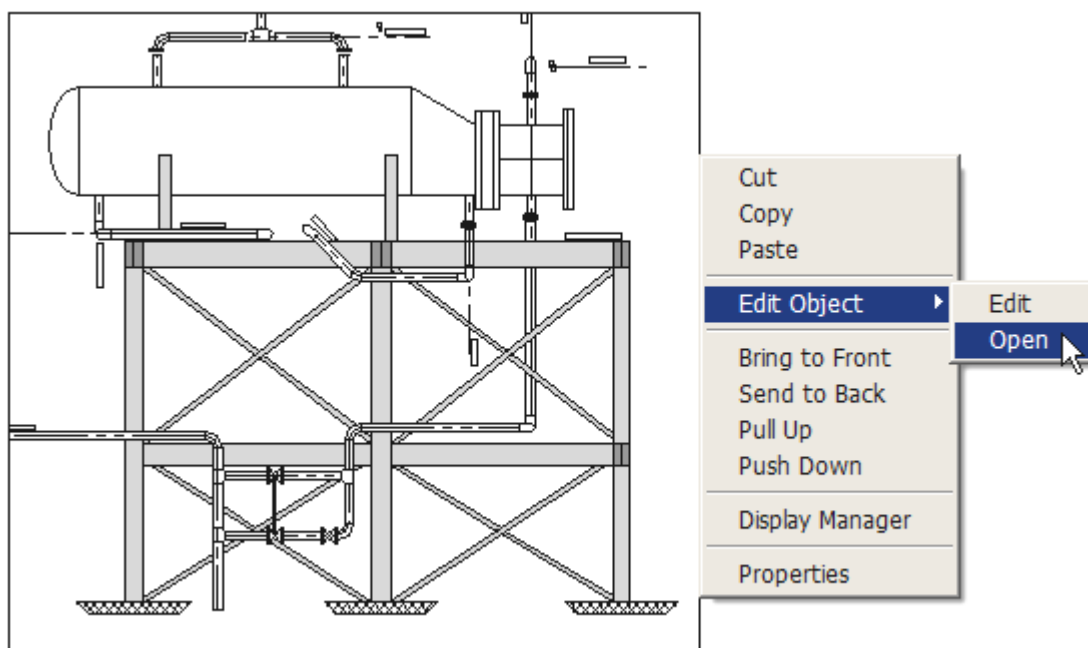
Once an object has been created or placed into a SmartSketch document from an external CAD document, the following actions can be performed:

- Scale, move, rotate, copy, or delete the object.
- Create geometric relations from SmartSketch graphic elements to elements in the referenced document.

Editing an Object

To edit a linked or embedded object, you can double-click the object to open the software that created it. You must have access to the software that created the object and the software must support OLE.

For embedded objects, you can also select the **Edit Object** command located on the shortcut menu which is activated by selecting the embedded object and click the right mouse button.. This will activate the software used to generate the original file.



Importing Foreign Data

SmartSketch provides the capability of importing data from other CAD applications, such as AutoCAD® and MicroStation®. SmartSketch can be used as a container to merely view and redline the data, as discussed in the previous section, or it may be used to translate the data into equivalent SmartSketch objects. The **SmartSketch CAD Translators** support translation of DWG, DXF and DGN file formats into SmartSketch. This capability - giving you access to legacy data - makes SmartSketch a powerful tool on your desktop.

Translate a Foreign CAD file

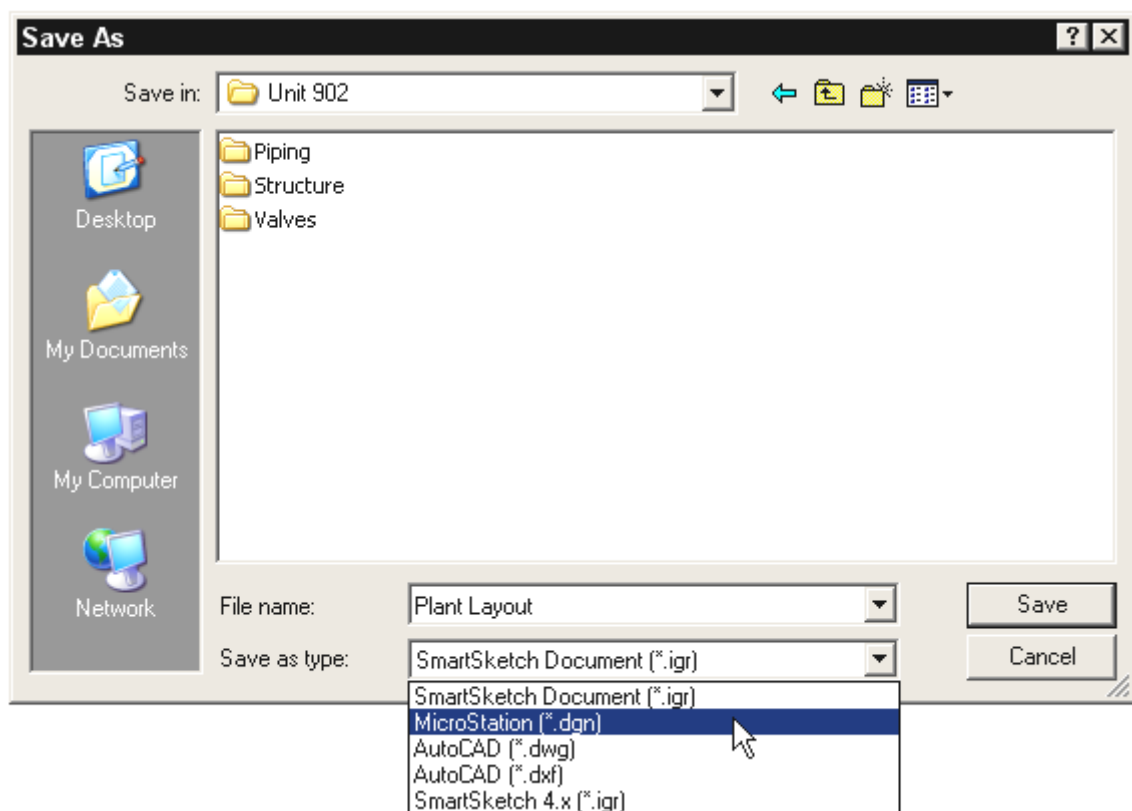
The SmartSketch **File> Open** command is used to translate foreign CAD files into a SmartSketch document. After opening the file, the design elements are translated into equivalent SmartSketch objects.

From the **File** menu, click the **Open** item to display the **Open** dialog. In the **Open** dialog, change the **Files of type** filter to the extension of the file format that you wish to translate. Locate the file that you wish to translate in the dialog box and complete the task by selecting the **Open** command.

Exporting SmartSketch Data to Foreign CAD Formats

In addition to being able to import foreign CAD data into SmartSketch format, SmartSketch provides the capability of exporting data into DWG, DXF and DGN file formats for use in other CAD applications, such as AutoCAD® and MicroStation®.

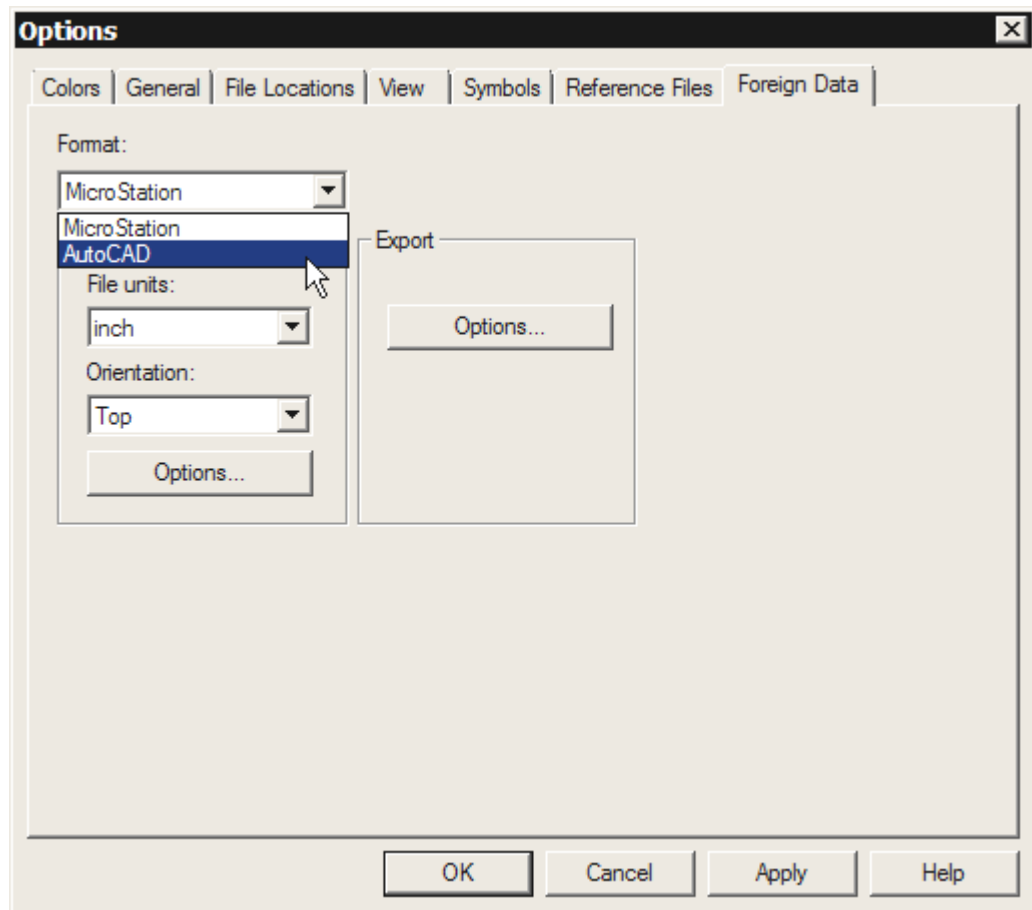
This is accomplished when using the **Save As...** command on the **File** menu.



Foreign Data Settings

Settings that control the import and export of foreign CAD data are accessible from the **Foreign Data** tab of the **Options** dialog. To display this dialog, click on the **Tools** menu and select the **Options** item, then click on the **Foreign Data** tab.

The **Foreign Data** tab is only available on the **Options** dialog box if you install the **CAD Translators** module.



Hyperlinks



Hyperlinks allow for logical links from objects, symbols, text, connectors, and elements in a drawing to files or web pages. This gives you the ability to add context sensitive or informational content from anywhere in an organization to a drawing.

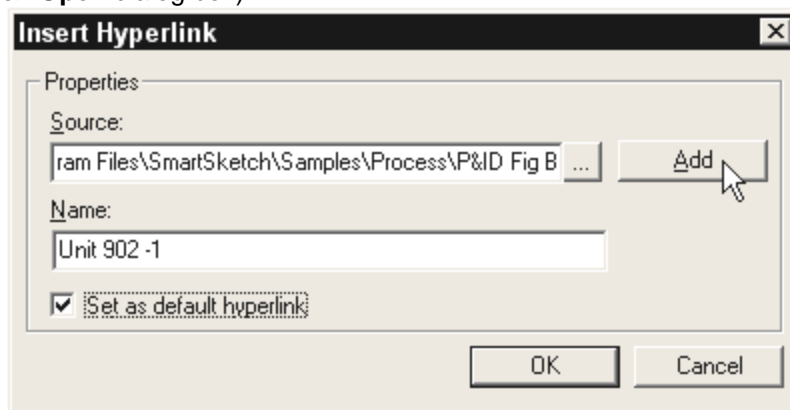
The Hyperlink command on the **Main** toolbar is used to create, edit and follow hyperlink addresses attached to objects in a document. The command is also located on the Insert menu. When you activate the **Hyperlink** command, a globe will appear on the cursor.



Pause the pointer over the object or element with a hyperlink. The pointer changes to a hand icon if a hyperlink exist. Click the left mouse button to follow the link.

Create a Hyperlink

With **Hyperlink** active, select the object that you want to associate with a hyperlink. This will open the **Add Hyperlink** dialog box. To define the file, URL or webpage that the hyperlink will open when selected, click the **Browse** button. This will open the **Link to File** dialog box (which looks similar to the **File> Open** dialog box).



If you anticipate more than one hyperlink will be applied to an object, you can turn on the **Set as default hyperlink** option. Once the file location has been identified and added to the **Source** field of the **Insert Hyperlink** dialog, enter a name that will appear as a **“tooltip”** when the cursor touches the hyperlink.

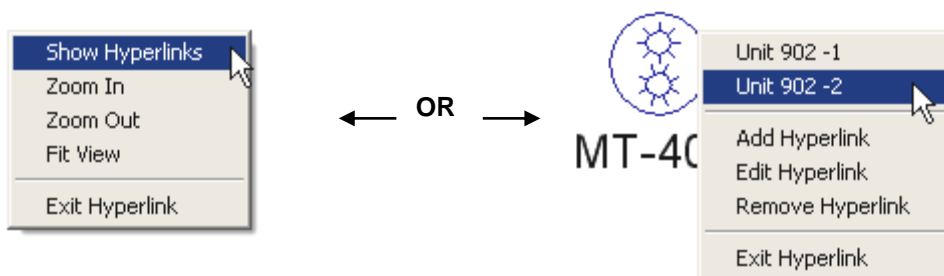


When you have completed the dialog box, click the **Add** button to complete the link. If you link to a SmartSketch document that contains multiple sheets, you are prompted to select which sheet to link to.



Show Links

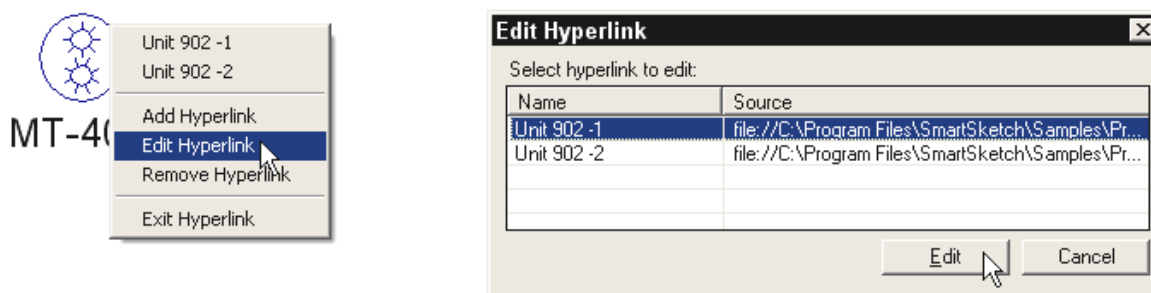
One way to display objects in a document that have a hyperlink is to click the right-mouse button in an empty area of the drawing window. This will cause all objects that have a hyperlink to be highlighted. You can also use the **Show Links** command listed on the **Hyperlink** shortcut menu.



Simply right-click on any drawing object to display a list of associated hyperlinks for that object.

Edit Link

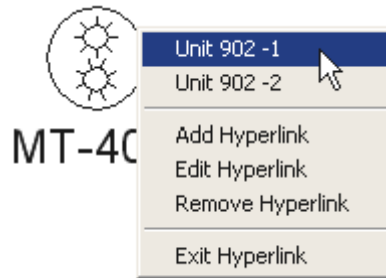
Allows you to change the web addresses or paths for hyperlinks that are applied to objects, symbols, text, connectors, and elements, such as lines, circles, ellipses, and groups.



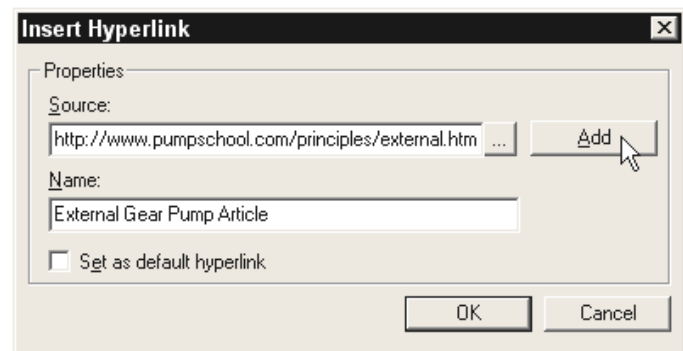
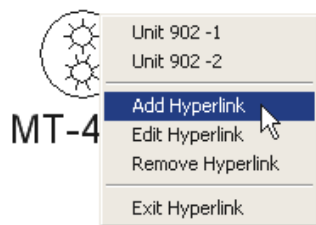
Pause the pointer over the object or element with a hyperlink. Click the right-mouse button on the object that contains the link to be edited. Select **Edit Hyperlink** from the shortcut menu and on the **Edit Hyperlink** dialog box, pick the hyperlink to edit, and then click **Edit**. In the **Edit Hyperlink** dialog box, type your changes to the hyperlink, when you are finished, click **OK**.

Adding Multiple Hyperlinks

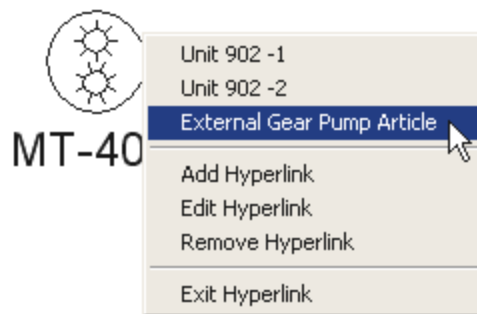
You can add as many hyperlinks as necessary: URL addresses, or file paths to an element or object. If you add more than one hyperlink, you must choose one of the links as the default. You can specify the link to follow from the list of links that display at the top of the shortcut menu that displays when the right-mouse button is clicked on a hyperlinked object.



Click the right-mouse button on the object that you wish to add a hyperlink to. Select **Add Hyperlink** from the shortcut menu. In the **Add Hyperlink** dialog box, specify an additional link.

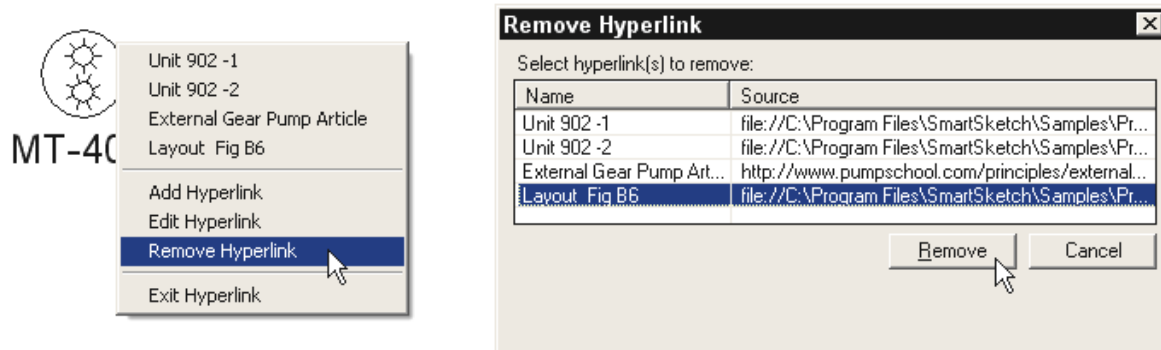


Click the **Add** button to accept the new link and then press **OK** to close the dialog box. If you move the cursor over the object, you should now see the new hyperlink listed on the right-click shortcut menu.

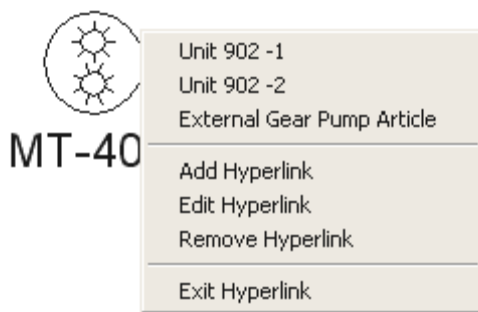


Remove a Hyperlink

It is also possible to remove a hyperlink from the document. Activate the Hyperlink command and position the cursor on the object that contains the link to be deleted. Click the right-mouse button to display the shortcut menu, select **Remove Hyperlink**.

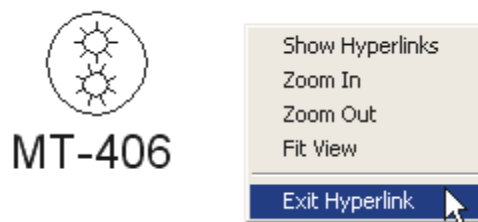


On the **Remove Hyperlink** dialog box, select the hyperlink that you want to remove and complete process by pressing the **Remove** button.



Exit Hyperlink

To exit the **Hyperlink** command, either select the **Exit Hyperlink** option from the shortcut menu or simply click on any other tool in SmartSketch.




Appendix Quick Labs

Quick Lab — Drawing File Type Comparison

1. Navigate to the training class folder.
2. Open the file named *Isometric.igr*.
3. Experiment with selecting objects on the sheet. Notice which commands load (ribbon bar) and what types of changes you can make to the objects.
4. Open the file named *Residential House Plan.igr*.
5. Again, experiment with object selection and the types of changes you can make to the data in the file.

Question: What is the main difference between *Isometric.igr* and *Residential House Plan.igr*?

6. Now open the file named Drawings-type file.sha.
7. Experiment with selecting objects that you see on the sheet. Notice that often there is only one object selected, the SmartFrame.
8. Pause over an object in the SmartFrame and use PickQuick  to select it. Try to change the color of the object with the ribbon bar. Is the color change applied?
9. Use one of the following methods to open the embedded file.
 - Right-mouse click on the SmartFrame and select Edit Object→Edit/Open.
 - Select the SmartFrame; select Edit->Shape2DServer Document Object→Edit/Open.
 - Double-click on the SmartFrame.
10. Select Fit and notice how object selection behaves.
11. Make some edits/revisions to the file.
12. Select File→Update/Close and save the changes.
13. Close all open drawings.

Overview: The intent of this lab is to demonstrate what types of objects are editable in a file and which objects are not.

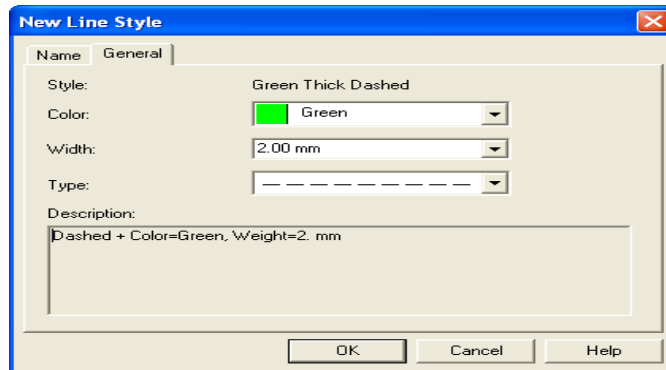
Object(s) displayed via a SmartFrame are not directly editable; the referenced (embedded) document must be opened.

Quick Lab — Creating Linear Styles

In this lab you will create several new linear styles using the Style dialog box and the Line Style Editor.

Create a Green Thick Dashed Linear Style

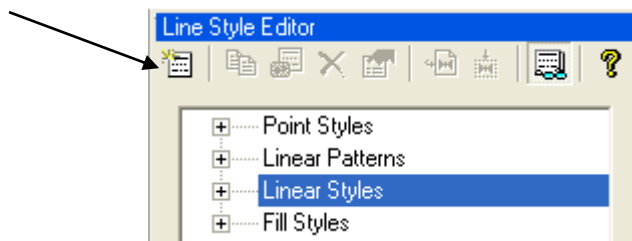
1. Start SmartSketch if it is not already running. Notice that a new file is automatically created from the Normal template.
2. Select **Format→Style**.
3. In the Style dialog box, click the **Resources** button to determine the style resource files that are attached to your document. Cancel out of the dialog boxes.
4. Select **Format→Style**.
5. Change the **Style type** to *Line* and click the **New** button.
6. In the New Line Style dialog on the **Name** tab, enter *Green Thick Dashed* for the **Name** and select *Dashed* in the **Based on** dropdown list.
7. Select the **General** tab and change the **Color** to *Green* and the **Width** to *2.00 mm*. Click the **OK** button.



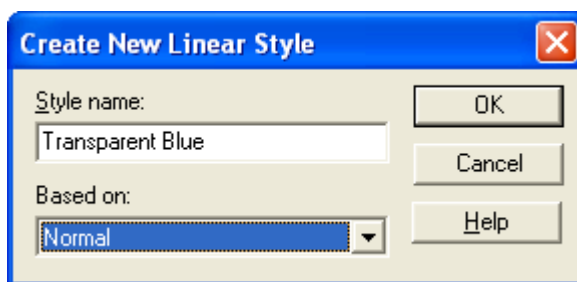
8. Select **Save As** from the **File** menu and save the file to the *C:\Drawing Files* folder as *ClassStyles.igr*.

Create a Transparent Blue Linear Style

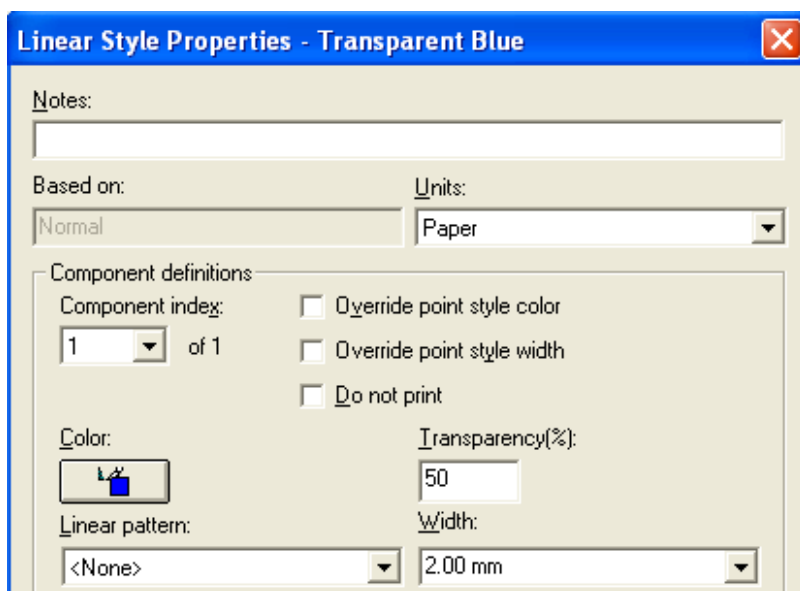
1. Select **Tools**→**Line Style Editor**.
2. In the Line Style Editor dialog bar, select *Linear Styles* and then click the **Create New Style** button.



3. In the Create New Linear Style dialog, enter *Transparent Blue* for the **Style name**, select *Normal* in the **Based on** dropdown list, and click the **OK** button.



4. In the Line Style Properties dialog, change the **Color** to *Blue*, the **Width** to *2.00 mm*, the **Transparency** to *50*, and then click the **OK** button.



5. Save *ClassStyles.igr*.

Create a Water Line (Optional Lab)

Place a text character

1. Select the **Text Box** command on the **Draw** toolbar.
2. In the ribbon bar, change the **Font Size** to *.125 in* if necessary.
3. Click once in the drawing window and key-in **W** for water. Right-click when done.



Create a point style from the text character

4. Select the **Zoom Area** command on the **Main** toolbar. Click once to the upper left of the **W** and then click again to the lower right of the **W** to zoom into the letter.
5. Select the **W** using the **Select Tool**.
6. In the Line Style Editor dialog bar, select Point Styles and then click the Create New Style button.
7. In the Create New Point Style dialog, enter *Water Symbol* for the **Style name** and click the **OK** button.
8. In the **Notes** field, key in *Symbol for Water Line* and click the **OK** button.
9. Make sure the **W** is still selected and then click the **Define Point Style Graphics** button from the Line Style Editor dialog bar.
10. When the target icon is displayed on the cursor, click in the drawing window near the center of the **W** to define the point style origin.



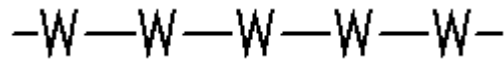
Create a linear pattern that includes the new point style

11. Select the *Water Symbol* point style in the Line Style Editor dialog bar. The **W** character is displayed in the preview pane.
12. In the Line Style Editor dialog bar, click the plus sign next to the *Linear Patterns* entry to expand it and then select the *Dash* pattern.
13. Click the *Create New Style* button.
14. In the Create New Linear Pattern dialog, enter *Water Pattern* for the Style name and click the **OK** button.

15. In the Linear Pattern Properties dialog, change the following properties and then click the **OK** button.



- Stroke index: 1
- Dash length: 0.13 in
- Stroke index: 2
- Gap length: 0.13 in
- Point style -
- Name: *Water Symbol*
- Position along stroke(%): 50

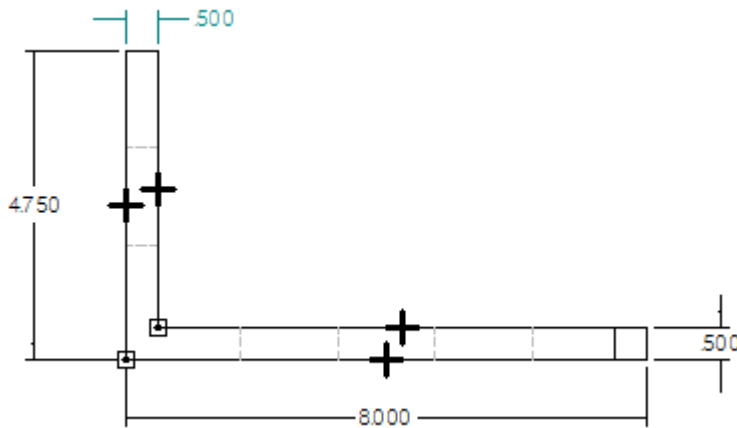
Create a linear style using the Water Pattern



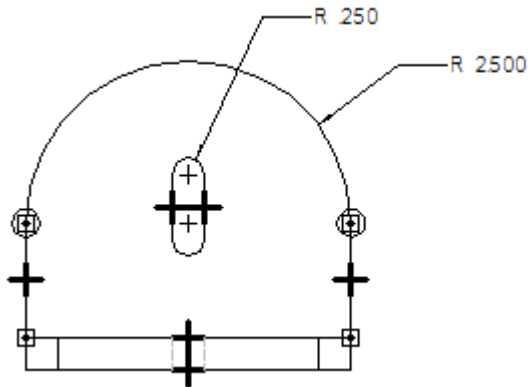
16. In the Line Style Editor dialog bar, expand the *Linear Styles* entry and select the *Normal* style.
17. Click the **Create New Style** button.
18. In the Create New Linear Style dialog, enter *Water Style* for the **Style name** and click the **OK** button.
19. In the Linear Styles Properties dialog, click on the **Linear pattern** dropdown and select *Water Pattern* from the list. Click the **OK** button.
20. Select **Tools→Options**. Select the **View** tab and toggle on *Display as printed*. Click the **OK** button.
21. Select the **Line** command from the **Draw** toolbar. On the ribbon bar, click on the **Style** dropdown list and select *Water Style*. Place a line using *Water Style*. Also place lines using the other linear styles created in this lab.
22. Save *ClassStyles.igr*.


Quick Lab — Dimensions

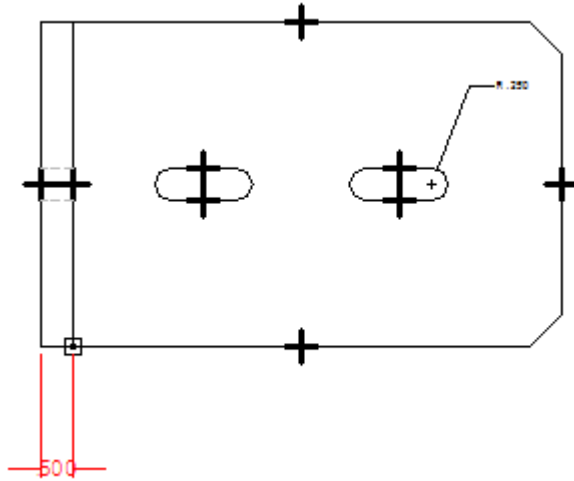
1. Open the file DimensionLab.igr.
2. Display the Dimension toolbar by clicking the **Dimension** command  on the **Main** toolbar.
3. Place the dimensions shown below using the **SmartDimension** command . Dimension the left hand face, the bottom face, the right hand edge, and the top edge of *View 1* in the drawing. Notice that the **SmartDimension** command is used to dimension individual objects.



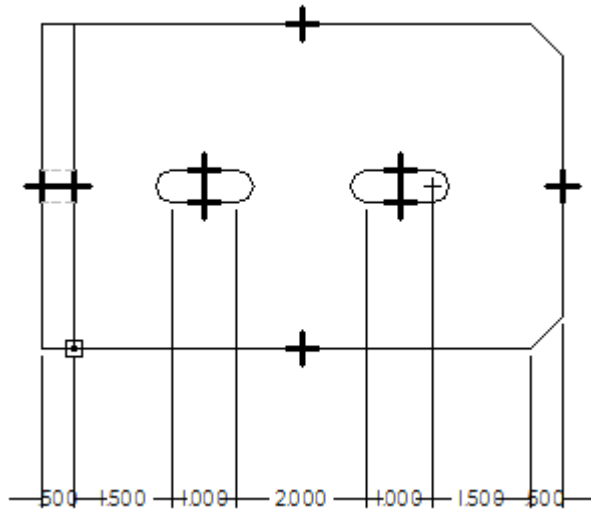
4. While still in the **SmartDimension** command, place a radius dimension on the vertical portion of the bracket in *View 2* of the drawing and on the top of the slot as shown below.



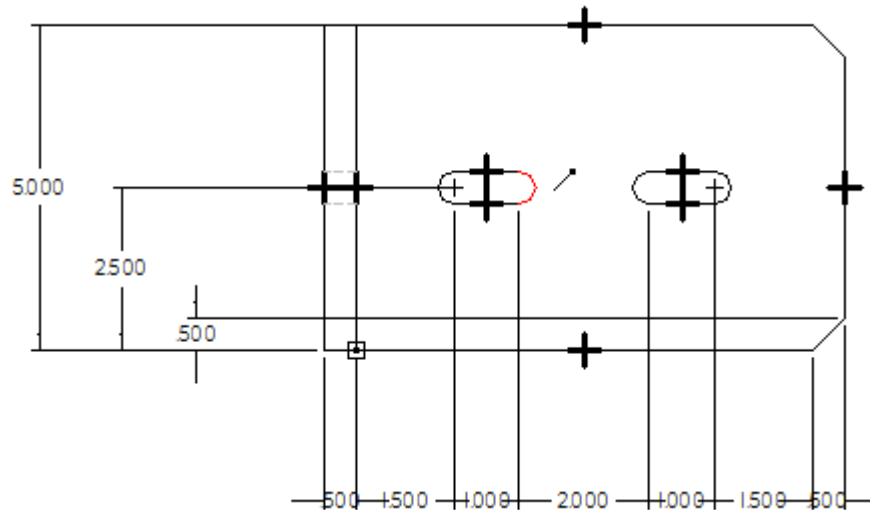
5. Zoom into *View 3* of the bracket and select the **Distance Between** command . Click the left hand face and the inside face of the vertical section of the bracket, as shown below. Click a third point to place the dimension between the two faces.




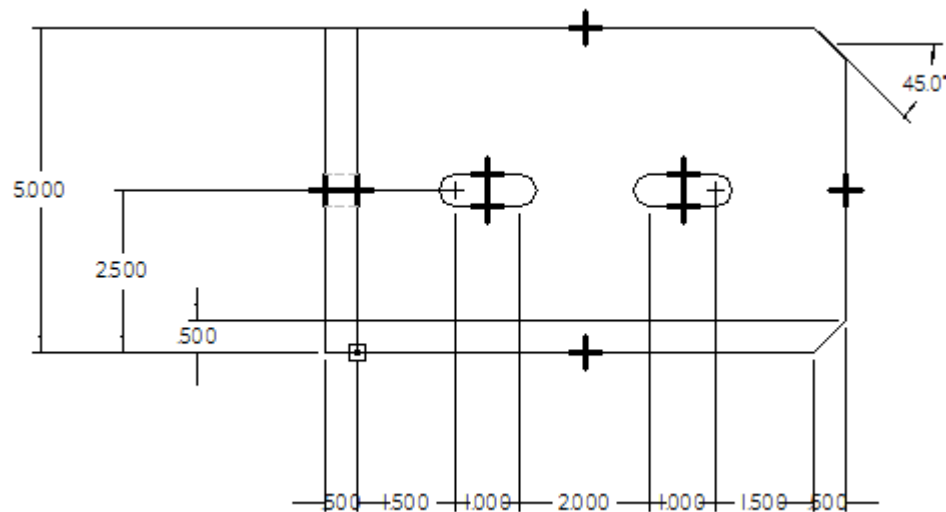
6. Continue in the **Distance Between** command and place dimensions by clicking each outside radius of the slots cut in the bottom portion of the bracket and the end points of the chamfer, as shown below. Notice that the dimensions display as either chained dimensions or stacked dimensions. Place the dimensions as chained dimensions. Right-click to exit the chained dimension string.



7. Select the **Distance Between** command again and click the bottom line on the lower left hand corner of *View 3*. After the first line is selected, click the top of the chamfer, the radius of the slot, and the top edge; place stacked dimensions instead of chained dimensions (see diagram below).



8. Finally, select the **Angle Between** command  and dimension the angle of the chamfered edge on the upper right hand corner of *View 3*.



9. Save the drawing.

Quick Lab — Environment

1. Select the **Circle by 3 Points** command from the **Circle** fly-out on the **Draw** toolbar.
2. Review the properties for circle placement in the **Dynamic Ribbon Bar**.
3. Follow the prompts displayed in the Status Bar and place a circle.
4. Select the **Esc** key or the **Select Tool** to terminate the command.
5. Move the cursor around the drawing sheet and read the **Cursor Coordinate Display**.
6. Display the **Dimension** and **Change** toolbars and dock them under the **Main** toolbar.
7. Undock the **Dimension** toolbar and leave it floating in the application window.
8. Select the **Fit** command from the **Main** toolbar to display all objects on the sheet.
9. Select the **New Window** command from the **Window** pull-down menu to create a new window.
10. Select the **Zoom Area** command and zoom in to the circle.
11. Select the **Tile Horizontally** command from the **Window** pull-down menu.
12. Close one of the windows using the **Close** icon in the upper right corner of the window.
13. Maximize the remaining window using the **Maximize** icon.
14. Exit SmartSketch without saving the document.

Quick Lab — Selecting and Finding Objects

1. Select **File→Open**.
2. Browse to the training class folder and select the sample file *P&ID #2.igr*.

Locate Gate Valve symbols and change color to blue

3. Select **Edit→Find**.
4. In the Find dialog box, select the **General** tab. Enter *Gate Valve.sym* in the **Symbol name** field and click the **Find Now** button.
5. Click the **Close** button to dismiss the dialog box.
6. In the ribbon bar, click the **Expand** button. Change the color of all selected symbols to *blue*.

Locate text with Arial font and change font to Times New Roman.

7. Select **Edit→Find..**
8. In the Find dialog box, select the **Type** tab.
9. Toggle off **All** and then toggle on **Label** and **Text box**.
10. Select the **Text** tab. Under **Font name**, toggle off **All** and toggle on **Arial**. Click the **Find Now** button.
11. Click the **Close** button to dismiss the dialog box.
12. In the ribbon bar, click the **Expand** button. Change the font of all selected text to *Times New Roman*.
13. Close the file without saving changes.

Quick Lab — Object Concepts

1. Place a line using the **Line** command on the **Draw** toolbar.
2. Dimension the line using the **SmartDimension** command on the **Dimension** toolbar.
3. Select the dimension and review its properties using the **Properties** command from the **Edit** pull-down menu.
4. Right-click on the dimension and select **Drop Dimension to Graphics**. The dimension is dropped to a group.
5. Select the group and then select the **Ungroup** command on the **Change** toolbar. The group is dropped to individual graphic objects.
6. Place a rectangle using the **Rectangle** command on the **Draw** toolbar.
7. Select the rectangle and review its properties using the **Properties** command from the **Edit** pull-down menu.
8. Right-click on the rectangle and select the **Convert** command. The rectangle is converted to four lines.
9. Place *Ball Valve.sym*, located in the class *Symbols* directory, in the drawing. **Zoom in** on the symbol if necessary.
10. Right-click on the symbol and select the **Convert** command. The symbol is converted to a group.
11. Select the group and then select the **Ungroup** command on the **Change** toolbar. The group is dropped to individual graphic objects.

