CHAPTER 13

Adding Text to Documents

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Adding Text to Documents

You can add text to documents using two different types of tools:

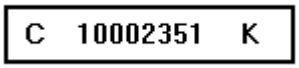
- Annotations, such as text boxes and balloons.
- Labels, such as text labels and Smart Labels.

Annotations

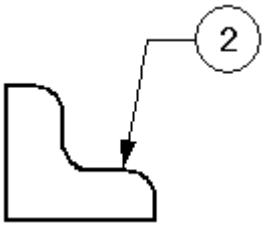
An essential part of the drawing process is adding text, notes, and annotations. Annotations are text and graphics that give information about a drawing. You can add this information quickly and easily with the text and annotation commands in the software.

To place annotations in a document, you can click one of the following buttons:

- Text Box button on the Draw toolbar

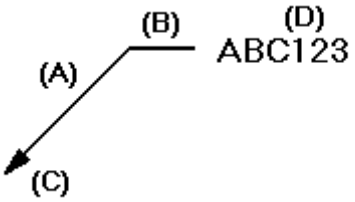


- Balloon button on the Dimension toolbar



Annotations with Leaders

When you create a balloon, you can place it with a leader by setting options on the ribbon bar. The leader can point to another element or be placed in free space. Annotations with leaders have the following components:



(A) Leader line

(B) Break line

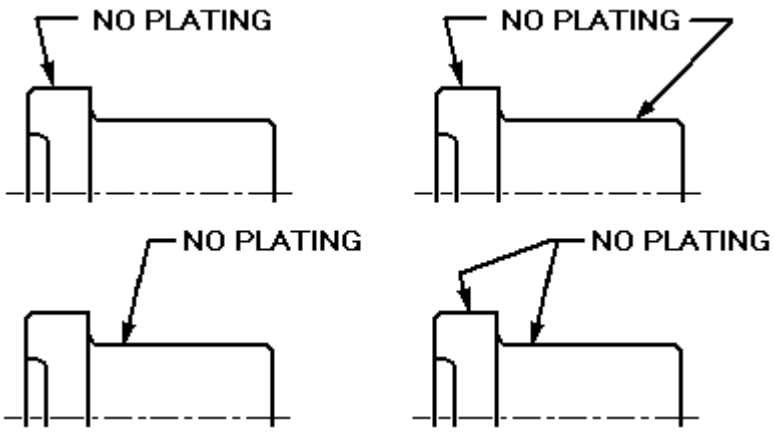
(C) Terminator

(D) Annotation

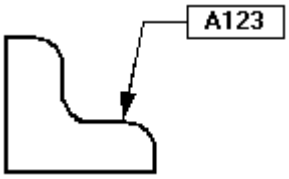
You can manipulate the annotation by selecting the leader and moving parts of it. You can control the display of a leader break line and terminator and insert or delete vertices on a leader.

Adding Leaders

You can add a leader to an annotation with the Leader command. An annotation can have more than one leader. The terminator end of the annotation can point to an element or be placed in free space. The annotation end of a new leader must connect to an annotation or the leader on an annotation.



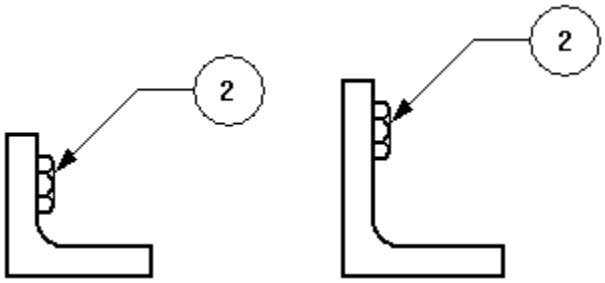
You can create a callout by placing a text box and adding a leader to it with the Leader command.



Annotations and Associations

Annotations can be associative or non-associative. An associative annotation moves when its associated element moves. Text boxes differ from the other annotations in that they are always non-associative.

If you attach the terminator of a leader to an element, the annotation moves with the element.



If you point the terminator of a leader to free space, the annotation is not associative to any element in the drawing. To make an annotation associative, you can select the terminator of the leader and drag it to an element.

Formatting Annotations

You can format an annotation several ways. If you want several annotations to look the same, you can apply a style by selecting it on the ribbon bar. You can apply dimension styles to balloons, but not text boxes. You can apply text styles to text boxes.

Tip If you want to format an existing balloon, you must click the balloon leader to select the balloon. Then, you can change the formats of the balloon.

If you want annotations to look unique, you can select an annotation and edit its properties with the ribbon bar or the Properties command on the Edit menu.

Engineering 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.

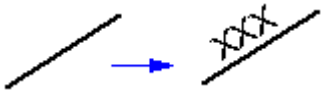
The software provides TrueType® fonts; with TrueType fonts, what you see on the screen is what appears on the printed page. The screen display of the document closely matches the printed document.

When you open a .DWG or .DGN document, the software provides a 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 mapping. You can change the mapping with the Options command on the Tools menu. On the Foreign Data tab, you can click the appropriate Options button.

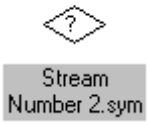
Labels

Technical drawings often have many labels that display important information about the objects or elements in the document. There are two types of labels: text labels and Smart Labels.

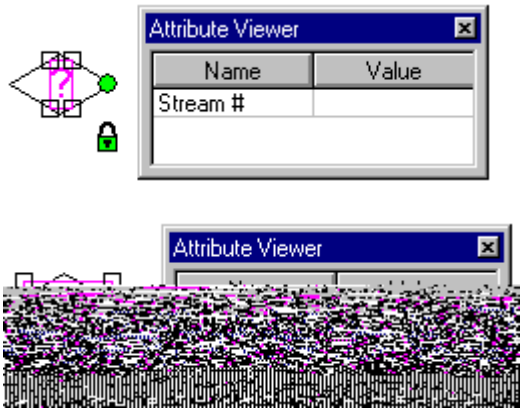
Text labels are associated with an element or object. If you move the element or object, the text label moves with it. You can create text labels in a document by double-clicking an element or object. A text label appears near the element so that you can type text. The default position of the label is bottom 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. When you enter the text that you want, the text label orients itself with the element.



Another kind of label, a Smart Label, can be associated to an object or element's text attributes. A Smart Label is often distinguished by a question mark in the display.



A Smart Label displays the attributes of the element or object as text in the document. You can drag Smart Labels from the Symbol Explorer into the current document. Smart Labels have an .sym extension. As you change an element's or object's attributes in the Attribute Viewer or the Properties dialog box, the Smart Label's text displays those changes.



If an element or object has no attributes, and you drag a Smart Label to it, the element or object picks up the attributes of the Smart Label. For example, you might want to assign attributes to a valve, such as pressure and material type. If the

you could update the drawing by editing the SmartSketch drawing. Each report would automatically display the results of the drawing updates.

If the document's size is not an important factor, then you might want to use embedding. Embedded objects make the file size larger. Embedding is useful when the person viewing the document with the information does not have access to the software that created the information. The person can still view the information if it is embedded in the document. For example, if you wanted to send out several drawings for review, you could embed each SmartSketch drawing into a Word document and mail the Word document to each reviewer.

Embedded information is also a good choice if the information does not need updating frequently. If you want to embed an object or edit an embedded object, you must have access to the software that created it.

Working with Raster Images

Images created by lines of pixels are called raster images, and the code that represents the pixels in the raster image is called raster data. The format of the raster data in the document determines how the pixels are arranged when the image is displayed. You can edit raster data

Differences Between Raster Data and Vector Data

Raster data is unintelligent. A line segment in a raster image cannot be manipulated as a line segment. Rather, it must be manipulated as the collection of pixels that form the line segment.

A vector data document is a collection of coherent, geometric elements. Vector data is intelligent. A line segment in a vector file can be manipulated as a line segment.

A vector data object is an indivisible entity. A raster image, by contrast, is composed of pixels, arranged to give the appearance of lines, shapes, and characters. When zoomed out, a raster image appears as contiguous lines and shapes. However, if you zoom in, it becomes apparent that a raster image consists of individual foreground pixels, represented as small squares on the screen.

Displaying Raster Images

ImageScape™ LT allows you to display and manipulate raster images along with the vector data on the drawing sheet. When you save the current document, which contains vector graphics, you can save a link to a raster image along with it.

This structure can be useful in many ways. For example, if a shopping center is planned for a piece of property, and proposals are sought for the design of the shopping center, an aerial photograph may be taken of the property. The photograph can then be converted into a raster image. Proposals can be submitted as vector documents that each contain a link to the original raster image. In this way, different proposals for the shopping center can be easily evaluated, with each proposal starting from the same raster image of the property.

Inserting an Image

To insert a raster image, you must first install ImageScape LT with the Add-Ins command on the Tools menu. If you do not see the add-in listed in the Add-In Manager, you must run Setup again and select the Custom installation. You can then select an option for the add-in.

After installing ImageScape LT, you can then insert the raster image with the Image command on the Insert menu. You can only link the image; you cannot embed it. To edit various properties of the image border, you can select the image and click the Properties command on the shortcut menu.

Use the Position command to move, scale, rotate, and skew a source image to match a target image or vector frame. All alignment modifications are made by placing up to three source points and three target points. You can define source points by clicking and dragging or by a single click on the source image.

Expanding the Raster Capabilities

You can expand the capabilities of the software with another Intergraph product called ImageScape™. ImageScape allows efficient and productive use of scanned drawings, aerial photographs, and rendered images. With ImageScape, you can do the following:

- Clean up and modify scanned drawings, maps, and images.
- Revise old scanned drawings with powerful drawing tools in the software.
- Display raster or vector data transparently over an image backdrop.
- Place images in a precise position.
- Warp images or scanned drawings to correct for geometric distortions in the original image.
- Produce and print hybrid raster or vector drawings for more effective presentations and proposals.

Procedures to Explore in On-line Help

- Embed an Object
- Link an Object
- Insert a Raster Image

CHAPTER 16

The Internet and Intranet

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Using the Internet and Intranet

SmartSketch is an excellent tool for publishing on the Internet. While you work on a document in the software, you can access the Internet and Intranet, which allows you to share design information within and across organizational boundaries.

Viewing the Web

You can view the World Wide Web from within SmartSketch with a special tool called the Symbol Explorer. The Symbol Explorer is available only if you have installed Internet Explorer 3.0 or later. After you click the Symbol Explorer button on the Main toolbar, the Symbol Explorer appears on the right of document window. You can type the URL or directory path of an HTML page in the Address **Box** and press Enter to view the HTML page.

Publishing on the Web

SmartSketch is an ActiveX® document. This feature means that you can open a SmartSketch document with its own toolbars and menus in the Internet Explorer window, without having to activate SmartSketch. SmartSketch toolbars and menus combine with those of the Internet Explorer, allowing you to work on the document from the Web viewer. This practice can be convenient if you want to work on a SmartSketch document and then go back to the HTML page that you were viewing. Web administrators can use existing SmartSketch documents on the intranet without having to convert them to another format for viewing on the Web.

Tip You can also view ActiveX documents in Netscape™ Navigator if you install the appropriate plug-in software. For more information, you should visit www.netscape.com on the World Wide Web.

If you want to place a graphic document in an HTML page for viewing on the Web, you can save the document to .cgm format.

Accessing the Internet Without Leaving SmartSketch

If you want to access the Internet, you must first be connected to the Internet with a modem. Then, you can access the Internet without leaving SmartSketch by clicking the SmartSketch on the Web command on the Help menu.

Tip The SmartSketch on the Web command displays in your system default web browser. If you do not have a web browser, you can install Internet Explorer,

which is delivered with SmartSketch. SmartSketch on the Web works with any web browser, but you do get extra settings and hot links by using Internet Explorer.

From the SmartSketch on the Web, you can access a myriad of tools to help you use SmartSketch more efficiently, keep up-to-date on product happenings, view a Web gallery of SmartSketch drawings, and contact Intergraph.

You can also access information from your computer or from the SmartSketch CD. So, you do not have to connect to the Internet to use the SmartSketch on the Web. Some of the types of information you can access locally are tutorials, gallery documents, and a symbol library.

Publishing Customer Drawings on the SmartSketch Web Gallery

The SmartSketch on the Web HTML page also has links to a Web Gallery that displays a collection of customer drawings produced with SmartSketch. The Web Gallery shows drawings from each industry: mechanical, civil, and architectural engineering, utilities, and so forth.

You can submit your own drawings to the Web Gallery by following the instructions on the Web Gallery HTML page. Just follow the links from the SmartSketch on the Web page.

Hyperlinks

You may want to link an element or object in a document to other documents, such as a web page. The Hyperlink button on the Main toolbar is used to create, edit and follow hyperlink addresses attached to objects in a document.

Tip If you use the Save As command to save the document to a .CGM file, the hyperlinks are retained during the translation process, creating a .CGM drawing file that is ready for web publishing. You can also use the Save As Web Page(s) command to do this.

ActiveCGM®

If you want to place a SmartSketch document on an HTML page for viewing on the Web, you can save the document to .CGM format. Computer Graphic Metafile (CGM), a standard file format for vector and raster data, is an ideal format for publishing 2D vector drawings on the Web. This format requires little storage space, displays quickly in a web browser, and has the ability to zoom and display more detail. CGM documents can also contain embedded hotspots, hyperlinks, and database links, allowing you to access information outside the document instantly.

You can save a document as a .CGM graphic embedded in a web page with the Save As Web Page command on the File menu. You can also save to .CGM with the Save As command.

Definition of CGM

CGM supports the definition of graphic objects bound to non-graphic attributes. The CGM file format is defined by ANSI/ISO 8632-1992 (American National Standards Institute/International Standards Organization) and MIL-STD-2301 (Military Standard). CGM is the international standard that defines a hybrid (raster, vector, and text) graphics format. Definition of graphical objects is supported at any arbitrary level of abstraction through a tagging mechanism similar to SGML and HTML.

Definition of ActiveCGM

ActiveCGM® technology converts 2D design files, such as construction drawings, into CGM files to link non-graphical data into graphical elements. For example, you can navigate an intelligent drawing using a Web browser. ActiveCGM provides rapid, accurate, enterprise-wide access and display of graphic information. It allows hyperlinking of vector and raster graphics defined in the Version 4.0 Computer Graphics Metafile format. ActiveCGM products based on ActiveCGM technology support zooming, panning, magnification, navigation of CGM graphics.

Procedures to Explore in On-line Help

- Access the SmartSketch World Wide Web Page
- Open a SmartSketch Document Inside the Internet Explorer

CHAPTER 17

Customizing the Software

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Customization

Customization allows you to modify and enhance the software to meet your specific needs. You can use your favorite standard Windows programming tools and languages that are OLE-aware, such as Microsoft Visual Basic. There are several ways to customize the software:

- You can add built-in commands or macros to the toolbars and menus in the software.
- You can go a step further and create your own macros in Visual Basic or other OLE-aware programming applications. Macros can automate a series of tasks and speed up your work.

Customizing with Built-In Commands

You can add built-in commands or macros to the toolbars and menus in the software. For example, the Text Box button is on the Dimension toolbar. If you use the Text Box button more frequently than any other button on the toolbar, you could add the Text Box button to the Main toolbar. If you want to offset elements, you can add an Offset button to any toolbar that you want.

You can also add button commands to a menu or a menu command to a toolbar. For example, you can place the Sheet Setup command on the Main toolbar as a button.

To add a button to a toolbar or a command to a menu, you click the Customize command on the Tools menu. From there, you can click the Menu or Toolbars tab and click the options that you want.

If you want to create a new toolbar that does not exist in the software, you can start by clicking the Toolbars command on the View menu. You then click the New button and type the name of the new toolbar on the New Toolbar dialog box. After you click OK, the Customize dialog box automatically appears so that you can add buttons to the new toolbar.

Tip When you add commands or buttons to the menus or toolbars in the software, the changes that you made appear only if a document is open.

Customizing with Macros

Not only can you add built-in commands or buttons to the software, you can also create your own macros that can be added to the menus and toolbars in the same manner. The Customize command on the Tools menu allows you to add macros to

software's standard menus and toolbars. You can also run the macros directly by clicking the Macro command on the Tools menu and selecting the macro.

You can create the macros in Visual Basic or other OLE-aware programming applications. Some of the OLE-aware programming applications are VBA (inside Excel), Visual C++ Delphi, Visual Basic, and so forth.

A type library is delivered with the software containing all the objects, properties, and methods available with the product. On the Help menu, click Programming with SmartSketch to access the programming Help. In the Table of Contents, you can also get a complete reference to the objects, methods, and properties. You can view the type library with Visual Basic's type library browser or the browser for an OLE-aware programming application. Practical examples of macros are also delivered with the software.

Custom Command Macros Delivered with the Software

The software provides custom command macros whose source can be viewed in the \Smartsketch\CUSTOM directory. These macros have been compiled into the \Smartsketch\CUSTOM\BIN directory. There you will find stand-alone .EXE files and a subdirectory called OLESVR that contains OLE-server .EXE files and .DLL files. The files in the OLESVR directory can executed directly from within the software. A README.TXT document located in the source directory for each macro explains how to run them.

Tip If you did not install the macros and macros source during installation of the software, you must run the Setup program again. In the Setup program, you must select the Custom setup.

For example, you can run the EXTEND.EXE custom command macro inside the software. This sample was written in Visual Basic and compiled as an OLE-server executable program. A README.TXT document for each custom command macro provides more details on compiling as an OLE server. You can find the README.TXT for each custom command macro in a specific directory: \PROGRAM FILES\Smartsketch\CUSTOM\<SAMPLE MACRO NAME>\README.TXT. For example, the README.TXT document for EXTEND.EXE is found in \PROGRAM FILE\Smartsketch\CUSTOM\EXTEND\README.TXT.

For a complete list of macros, click Help and then Programming with SmartSketch.

Example: Customizing a Toolbar

With the Extend to Next button on the Draw toolbar, you can extend one line to the next line. This command is easy and intuitive to use. However, sometimes you

might want to extend a line a specific distance that does not depend on the location of another element.

1. In the document, place two lines that are not connected, but are perpendicular.
2. On the Tools menu, click Customize.
3. On the Customize dialog box, click the Toolbars tab.
4. In the Categories box, click Macros.
5. Click the Browse button.
6. On the Select Macro Directory dialog box, click EXTEND.EXE in the \PROGRAM FILES\SMARTSKETCH\CUSTOM\BIN\OLESVR directory.

   Note If you cannot find the Custom directory, you must run the Setup program again and select the Custom option.

7. Click OK.
8. On the Customize dialog box, in the Macros box, drag the EXTEND.EXE filename out of the dialog box.
9. On the Custom Button dialog box, in the Buttons box, click the yellow happy face button.
10. Click the Assign button.
11. The new button appears on a separate floating toolbar. Drag this toolbar over to the top of the window and place it beside the Main toolbar.
12. On the new toolbar, click the happy face button.
13. On the Extend by Distance dialog box, in the Distance box, type 10 and press ENTER. The dialog box automatically converts whatever you type into the units of measure that are set for the current document. You can set the units with the Properties command on the File menu.
14. In the document, drag the mouse over the line that you want to extend. A preview of what the line will look like shows up in a red color.
15. Click the line to set the distance.

16. On the Extend by Distance dialog box, click Close.

You can write your own applications that are very similar to the EXTEND.EXE file and run them from within the current document. Click the Programming with SmartSketch command on the Help menu to get more information about creating your own macros.

Adding Additional Tools to the Software

Add-ins are commands or functions that add special capabilities the software. To install an add-in, choose the Add-Ins command from the Tools menu. After you install an add-in, its commands or functions become an integrated part of the software until you remove the add-in.

Add-Ins Included with the Software

The following add-ins are included with the software and are located in the LIBRARY directory or one of its subdirectories. If you cannot locate the add-in on your hard disk or network drive, you can select the Macro Library option or other add-in option in the Setup program to install these add-ins.

Add-in	*Description*
Auto Save	Automatically saves open documents at the interval you specify.
The Wizard of ORG	Creates organization charts quickly and easily.

Procedures to Explore in On-line Help

- Run a Macro
- Add a Button to a Toolbar
- Customize SmartSketch with the Options Command

CHAPTER 18

Managing and Printing Documents

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Saving Documents *254*
Saving a Document Automatically *256*
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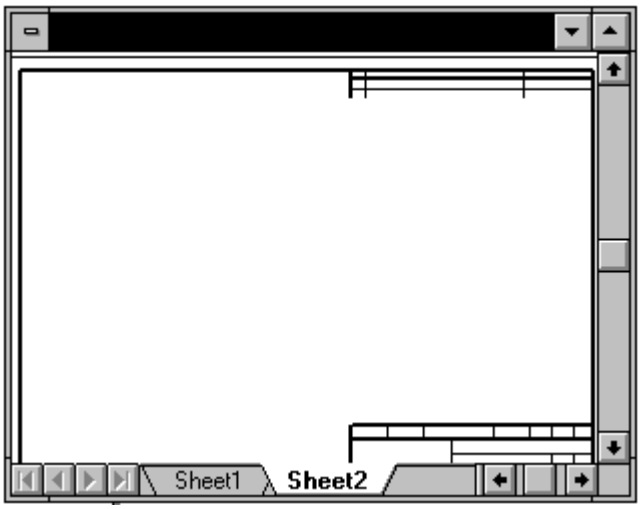
Managing Multi-sheet Documents

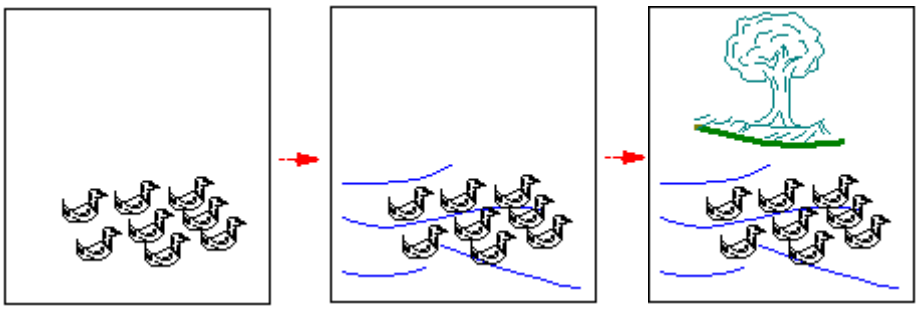
Drawing sheets are similar to pages in a notebook. You can place different sketches or drawings on different drawing sheets in the document. For example, you can draw one idea for a design on one drawing sheet and another idea on another drawing sheet. Both drawings are saved in the same document. There is no limit to the number of sheets you can have in a document.

When you create a new document, a default sheet appears with a border and a title box. You can modify the default settings with the Sheet Setup command on the File menu. For example, you might want to give a specific name to a sheet by setting options with this command. If you want to insert new sheets, click the New Sheet command on the Insert menu.



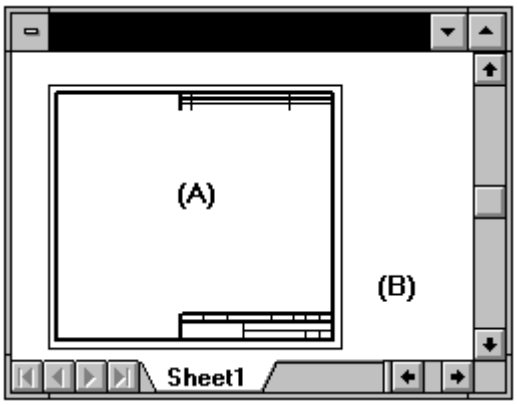
Sheets essentially act like separate drawings. Individual sheets can have different scales and sheet borders. If you want to be able to locate or use relationships between elements, you should place the elements on the same sheet.

You can display a number of drawing sheets on top of each other and simultaneously view the drawings on all of the drawing sheets, much as you would view a stack of transparencies.



Areas of a Drawing Sheet

A drawing sheet has two areas that affect the way you work. The sheet outline (A) shows the orientation of the sheet and the default print region of the sheet. You can change the size and orientation of the sheet outline with the Sheet Setup command on the File menu.



You can draw new elements on the sheet outline or outside the borders of the outline. All new graphics that you create are placed on the active drawing sheet.

Setting Up a Drawing Sheet

You can set up a drawing sheet with the Sheet Setup command on the File menu. The sheet where you will create most of your graphics is called a working sheet. In a document, you can create as many working sheets as you need to plan your project. A working sheet can have a single background sheet attached to it.

Choosing a Sheet Size

You can modify a drawing sheet's characteristics, such as the size and scale of the sheet, with the Sheet Setup command on the File menu. On the Sheet Setup dialog box, you can click the Size and Scale tab and select a standard sheet size or set a custom size. The dialog box also allows you to set the defaults for all new working sheets created in the document. You can do this by setting the options you want and then clicking the Save as Default button.

Drawing Sheet Scale

Drawing sheets allow you to use a specified scale as you draw. When you specify the scale with the Sheet Setup command on the File menu, everything on the drawing sheet is scaled except for dimensions and annotations. For example, if the scale is one inch to ten inches and you place a line 30 inches long, the line is three

inches long in proportion to the drawing sheet and three inches long when printed. However, a dimension measures the length of the line as 30 inches.

The dimension and annotation sizes in your working sheets are independent of the drawing view scale. For example, when you define the height and size of dimension text, these values specify how you want the dimensions to appear when you print the drawing.

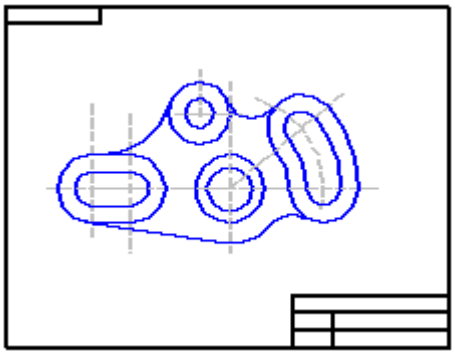
The graphics on the attached background sheet are displayed at a 1:1 scale. They are not affected by the scale set with the Sheet Setup command on the File menu.

Manipulating Drawing Sheets

At the bottom of the drawing sheet, you can see drawing sheet tabs that allow you to manipulate and display the drawing sheets in your document. When you open a document or create a new one, the tabs are displayed at the bottom of the work space. There is a tab for each drawing sheet in your document.

Attaching a Background Sheet

After you set up the drawing sheet, you might want to attach a background sheet. A background sheet is used as a backdrop to the working sheet. For example, you can add borders to a drawing, draw a title block, insert a raster image, or draw geometry. You can make one background sheet and attach the same background sheet to several different drawing sheets.



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

Background sheets allow you to customize your borders and title blocks so that they match your company's standards and requirements. You can use the background sheet delivered in a template or design your own background sheet. You can create tailor-made background sheets by adding your company's logo,

creating any other graphics that you need, and using various styles and options. A typical customization scheme would be to have a different background sheet for each standard-sized drawing (A, B, C, D; or A0, A1, A2, A3, A4).

Sheets and Document Templates

You can re-use your sheets by creating and saving them in a document template. When you create a new document based on a template, all of the drawing sheets that are part of the template are copied into your new document.

Printing Documents

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 prints exactly as it appears on the drawing sheet.

Preparing to Print

The printer you use affects the way the current document prints and displays text on the screen. Before you print a document, you must install and select the printer that you want to use. For more information about installation, see your printer documentation.

Before you print, you will want to choose a printer and the settings for it. To do this, click the Print command on the File menu and then click the printer that you want to use. You can set properties for the printer by clicking the Properties button. To set the print range and scale, click the Settings button.

Printing a Document

As you work on a document, you might need to send a copy of it to a specified printer, plotter, or file. You can click the Print command on the File menu to do the following:

- Print an entire document or specific sheets from a document.
- Print a draft copy of a document.
- Set printing options, such as the range of sheets or number of copies to print.

The software supports WSYWIG plotting, using standard Windows NT plotting capabilities. The software also supports pen plotters, subject to the limitations of the device driver. Elements appear the same on the screen and in the printed document.

However, the fonts that you choose can affect the match between what you see on the screen and what appears on the printed page. Three kinds of fonts affect your work: scalable fonts, printer fonts, and screen fonts. Use scalable fonts, such as TrueType® fonts, to confirm that what you see on the screen is what appears on the printed page. If you use printer fonts, you must have a corresponding screen font and font size to display each font on the screen. If each screen font you use has a matching printer font, the screen display of the document will closely match the printed document.

Printing Part of a Document

You can print selected sheets in a document or a selected area. This practice is handy if you are working on a complex drawing and you want to print only certain parts to proof them.

To print selected sheets, you should first select sheets in the document to make them active. To do this, you can click the drawing sheet tabs of the sheets that you want while pressing Ctrl. Then, you select the Print command on the File menu and click the Selected Sheets option on the dialog box.

When you select several sheets and then click the Selected Sheets option on the Print dialog box, all the selected sheets are printed as one composite drawing, like layers on top of one another. The software uses the scale, paper size, and orientation of the active sheet for printing the final document.

To print a selected area, you should first select the drawing sheets that you want to print. Then, you select the Print command on the File menu and click the Print Area option on the dialog box. After you click the OK button, a set of crosshairs appears. You drag over the area that you want to print. The Print Area dialog box then automatically appears and allows you to set options for printing the selected area.

Printing Time

Required time for printing varies by document. Expect longer print times when you have drawings that contain a large number of the following items:

- Curved elements
- Many fills that have complex colors, patterns, or textures
- Large bitmaps
- Links to various documents

The type of printer that you have also affects the printing time. PostScript® printers print faster than LaserJet printers because they can calculate faster. Some older versions of PostScript printers are slower when compared to the newer ones. The type of controller boards and the amount of RAM in the printer also affect your printing times.

Saving Documents

When you first save a new document, the software provides a default name and directory location. You can give the document a meaningful name and specify where you want to store the document on the disk.

When you open an existing document, the software copies the document from the disk and displays it on the screen. As you work, the changes you make to the document are displayed on your screen. To preserve these changes, you must save the document. When you save the document with the Save command, the changes are copied to the document on the disk.

Tip You can quickly save changes to an open document by clicking the Save button on the Main toolbar.

After you have saved a document, it remains displayed on the screen. You can use the Close command to close the document and clear it from your screen, or you can use the Exit command to close the document and exit the software.

Saving a Copy

If you need to make a copy of your work, you can click the Save As command on the File menu. You should type a different name for the document in the Save As dialog box.

Saving a Document as a Template

Sometimes you might want to make a copy of the same document as a starting point for creating other documents. In these cases, you can create a template to use as a starting point for many different drawings. A template is a collection of settings that you use over and over in different drawings. The settings might include drawing sheet settings, scales, actual elements that you draw, and a background sheet.

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 you installed the software. A different file extension in the name of the document is not necessary. To base new documents on the template, choose the New command on the File menu and select the template from the list. You can also open the template and save the document under another name.

Tip You can change the directory where templates are saved by selecting the Options command on the Tools menu and setting the directory that you want on the File Locations tab of the Options dialog box.

Saving Documents to CGM Format

The Computer Graphics Metafile (.CGM) provides an efficient way to make information available on the Internet. You can use the Save As command to save documents in .CGM format. Once in .CGM format, you can place the graphics on a web page (.HTML or .HTM) and view them with browsers such as the Internet Explorer or Netscape Navigator.

CGM is an efficient format for publishing 2D vector drawings on the Web because it takes up little storage space, renders quickly, and has the ability to zoom and achieve more detail. These types of drawings can contain embedded hyperlinks, allowing quick access to information outside the drawing.

Saving Documents to HTML Format

In addition to the CGM format, you can use the Save As Web Page command to save documents in .HTML format. This command saves the document in an HTML document; the graphics in the document are embedded in the HTML as .CGM graphics. You can then view the document with browsers such as the Internet Explorer or Netscape Navigator.

Closing a Document

If you want to close a document without saving your work, click the Close command on the File menu and then, when prompted to save your work, click the No button. If you want to save your work, you can click the Yes button instead. If you decide not to close the document at all, you can click the Cancel button.

Sending a Document to Others

After you finish drawing a sketch or adding notes to it, you can send the document to other users by clicking the Send command on the File menu. An e-mail message appears, allowing you to attach the drawing to the message. You can also embed or link the drawing to a Word document and send the document in e-mail to other users.

Saving a Document Automatically

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 often 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.

Caution If the AutoSave option does not appear on the Tools menu, you must install it by clicking Add-Ins on the Tools menu.

Procedures to Explore in On-line Help

- Create a Document
- Create a Template
- Set Up a Drawing Sheet
- Add Styles to the Current Document
- Set the Unit of Measure
- Create a New Drawing Sheet
- Print a Document

CHAPTER 19

MicroStation Documents in SmartSketch

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Opening and Saving MicroStation Files

Opening and Saving MicroStation Files

SmartSketch supports opening and saving MicroStation files in two ways: you can either select a MicroStation file and translate it into equivalent SmartSketch elements, or you can create a reference file from a MicroStation file.

If you use the method for selecting a MicroStation file and translating it into equivalent SmartSketch elements, you can click Open on the File menu. Once a MicroStation file has been opened using this method, the resulting file is in a SmartSketch format (IGR).

Tip If the MicroStation file extension is not a .DGN extension, you may need to use a utility to define other extensions as MicroStation files.

If you use the method for creating a reference file from a MicroStation file, you can either click Object on the Insert menu, or you can drag and drop from the Symbol Explorer or Windows Explorer. This enables you to display and, if MicroStation is available on your PC, edit the MicroStation file while working in SmartSketch. You can also select the view position of a 3-D MicroStation file upon placement. Once a MicroStation file has been referenced using this method, you can then locate and establish relationships between the new information and elements on the drawing sheet.

Note The Options command on the Tools menu displays the Options dialog box. By selecting the Foreign Data tab and setting the format to MicroStation, you can configure the settings for importing MicroStation data that applies to both of these methods.

Caution Roundtrip of MicroStation data (importing MicroStation data to SmartSketch data and exporting back to MicroStation data) is currently supported visually. Data fidelity for information such as tags, database linkages, and user data are not supported during export to the MicroStation file format.

Other Methods for Importing MicroStation Data

You can use the following methods to move MicroStation (.DGN) documents into SmartSketch:

- Cutting and pasting.
- Dragging and dropping.

- Inserting the information as an object.

When you open a MicroStation document that has references to other documents, those referenced documents show up as well. You can locate referenced documents in the current document.

Tip You can also use the Open command on the File menu to translate and open documents. You can use seed files to preserve the document's original fonts and line types when you open a MicroStation document.

Cutting and Pasting

You can select the information in the MicroStation document and then cut or copy the information and paste it into SmartSketch with the Paste Special command.

Dragging and Dropping

You can select the information in the MicroStation document and then drag and drop it into SmartSketch. In the Windows Explorer, select the MicroStation document and then drag and drop it into SmartSketch. The document will be embedded in the drawing by default. Right before you drop the document, you can embed the document in SmartSketch by pressing the CTRL key. If you press CTRL + SHIFT, the document is linked. When you drop the document, handles appear that allow you to scale the object. The foreign document is placed into SmartSketch and behaves much like a symbol, allowing you to click to place it or move it around.

Tip You can set placement options by clicking the Options command on the Tools menu. Then, you can set the options you want on the Reference File tab of the Options dialog box.

Inserting as an Object

Inside SmartSketch, you can insert the entire foreign document with the Object command on the Insert menu. On the Insert Object dialog box, you can set the option to link or embed the document. 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 geometry of the inserted document is recognized; 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.

Tip The Object command inserts any OLE 2.0 enabled object, such as a Word or .AVI document, or CAD document, such as .DGN. The inserted object can then be edited by double-clicking it.

Saving SmartSketch to MicroStation

You can save a SmartSketch document as a MicroStation document. Before you do this, you can adjust the translation options with the Options command on the Tools menu. On the Foreign Data tab, you can click the Export Options button.

When you save a SmartSketch document as a .DGN document, the active sheet is saved. On the Options dialog box, you can map SmartSketch layers to MicroStation levels (1–63). All the remaining layers are placed on the last layer or level of the foreign document. The layer names are preserved, whether importing or exporting MicroStation documents.

Caution If you save a SmartSketch document with raster reference files as MicroStation documents, only the files of the same type (.DGN) can be exported back into SmartSketch.

Accuracy of MicroStation Data During Import

When you translate MicroStation data into SmartSketch, you are concerned with either Visual fidelity, Data fidelity, or both.

Visual Fidelity

Visual fidelity is a phrase used to describe the visual accuracy of a picture after translation. SmartSketch preserves the visual accuracy of a MicroStation file by providing support for the following areas:

Colors

MicroStation maintains a fixed number of colors in a color table for each document. Because SmartSketch uses true color, the color table of the seed file is used as the basis of the translation. The default color table provided with SmartSketch defines the first 16 colors as standard and fills in the rest of the colors as they are needed. If the color table is full, then each color is matched to the closest MicroStation color. MicroStation colors are automatically translated to the closest red, green, blue (RGB) values in SmartSketch.

Widths

SmartSketch provides a default set of line widths that corresponds to standard metric pencil leads: .13, .18, .25, .35, .5, .7, 1.0, 1.4, and 2.0. When you open a MicroStation document, a default mapping determines which SmartSketch line width is assigned to go with which MicroStation weight. You can change any of the default mappings to define your own mapping with the Line Width tab of the MicroStation Import Options dialog box (accessed through the Foreign Data tab after you select the Options command on the Tools menu).

Linestyles

The core MicroStation linestyles 0-7 default to matching SmartSketch linestyles, unless they are mapped on the Linestyle tab of the MicroStation Import Options dialog box. MicroStation User Defined Line Styles (UDLS) are supported directly. If you use the Style Resources dialog box, you can attach a MicroStation linestyle resource (*.RSC) file as a resource for linestyles to use in a document.

Patterns

MicroStation associative patterns are automatically recreated as fill styles in the resulting SmartSketch files. Non-associative MicroStation patterns are imported as groups.

Font Styles

SmartSketch supports the system TrueType fonts delivered with Windows NT and supplies additional ANSI and ISO TrueType™ engineering fonts. SmartSketch also supports MicroStation fonts. When you open a .DGN document, SmartSketch provides a default mapping to determine which TrueType font is assigned to which MicroStation font. You can change to this mapping to define your own mapping. You can change the mapping with the Options command on the Tools menu. On the File Locations tab, you can set the paths to the font resources by double-clicking on Fonts 1, 2, and 3.

References

MicroStation references are imported as SmartSketch reference files, which use the automatic defaults or same options for configuring how color, weights, line styles, patterns, and fonts are handled for visual accuracy.

Tips

- ImageScape LT must be downloaded to support import of raster reference files.
- None of the data that is part of the reference file can be deleted or altered using SmartSketch commands. However, if MicroStation software is installed on your computer, then you can click Open on the Shortcut menu over the reference file. This action opens the MicroStation reference file in the MicroStation application.
- If you establish relationships between elements in the SmartSketch file and a reference file, the relationships will be removed if you change the position of elements in the source file and then update the reference file in SmartSketch.
- If you change the inserted information (copied from MicroStation), the original information does not change.

Configuring the Fit to Sheet Feature

SmartSketch automatically calculates the sheet scale when you import MicroStation documents so that the range of graphics automatically fits within the lower left-hand corner of the drawing sheet. If you do not want the sheet scale automatically calculated, then you must manually modify the INI file.

To modify the INI file, you click Explore on the Start menu and go to the directory where SmartSketch resides. You can use any text editor to modify the file. Double-click itmstn. In the [Options] section of the file, set Ignore sheet scale = 0.

Tip You can create a template for calculating sheet scale at the scale you want and insert a MicroStation document to base the template on. On the Sheet Setup dialog box, you set the Drawing Scale and Paper Units that you want.

Using MicroStation Fonts and Styles in a SmartSketch Document

If you want to make line styles from MicroStation resource files available in a SmartSketch document, you can reference the styles with style resource documents. You can reference a MicroStation .RSC file from the active template or document by clicking the Resources button on the Styles dialog box.

When you attach a .RSC file from the current document or template, the styles in the resource document are added to the Style dropdown list on the ribbon bar for the active document.

Tip MicroStation resource files for line styles are treated as a document setting. If you want to have all documents that are created by importing use the same styles, you must create an .IGR template with those styles attached. You then specify the .IGR template as the template to use during import of MicroStation documents.

Data Fidelity

Data fidelity is a phrase that describes how accurately the data is preserved when compared to its original form in the application that created it. SmartSketch preserves the data accuracy of a MicroStation file by providing equivalent element types with mapped entities to SmartSketch. SmartSketch also provides support for non-displayed data such as database linkages, units, and coordinates.

SmartSketch preserves data accuracy of a MicroStation file by providing support for the following areas:

Units

MicroStation files are determined by the master and sub-unit labels in the file.

- If meaningful master units and sub-units are found, the resulting SmartSketch document has the closest matching units set on the Units tab of the Properties dialog box.
- If no meaningful master units and sub-units are found, the File Units setting in the Import area of the Options dialog box is used to determine the units for translated or referenced MicroStation files.

Coordinates

MicroStation data is automatically positioned to matching coordinate positions when you translate the MicroStation files (click Open on the File menu). Referenced MicroStation file coordinate positions are determined by the settings on the Reference File tab on the Options dialog box.

Tip You can use the Coincident option to import the data in its original coordinate position at a 1 to 1 scale.

Levels

MicroStation level numbers, names, or both are preserved as SmartSketch layer names, but the color and linetype settings (level symbology) are not. The color and linestyle settings apply directly to the resulting elements.

Within a SmartSketch document, there are an unlimited number of layers per sheet. When translating foreign files into SmartSketch, the levels in MicroStation keep the same names. The MicroStation Export Options dialog box provides a layer mapping table, due to the limit of 63 levels in MicroStation. You can access this dialog box with the Foreign Data tab after you select the Options command on the Tools menu.

Cells

When you translate a .DGN document, the default is to translate cells into symbols. You can change this default with the Options command on the Tools menu so that cells or blocks are embedded groups. This action translates cells and blocks into embedded symbols, or groups, in SmartSketch.

Tip All MicroStation Type 2 cells are imported into SmartSketch as a group. The group contains a dynamic attribute set that stores the name of the cell, its origin, and any other data characterizing the cell. There is, therefore, enough information stored in the group to reconstruct an element back into a cell to export in the future.

When you import a MicroStation document that has cells, a temporary folder is created in the Temp folder on your computer. The temporary folder contains the symbols that are embedded in the MicroStation document. By default, these symbols are deleted after the document is imported into the drawing sheet. You can change a setting in \\Smartsketch\PROGRAM \ITMSTN.INI to save the symbols instead. In Notepad, open the .INI file and change the following line in the Options section: Delete Symbol Definitions = 0.

3D Data

All 3D data is flattened into a 2D view when you translate files using Open on the File menu. The 2D view orientation is determined by the Orientation setting in the Import area of the Options dialog box.

The following information is an entity-to-element map when you translate files using Open **on the** File **menu.**

MicroStation	*SmartSketch*
Line	Line
Linestring	Linestring
Bspline Curve	Bspline Curve
Shape	Linestring
Ellipse	Ellipse
Arc	Arc
Complex String	Complex String
Complex Shape	Complex String
Bspline Surface	Bspline Curve
Curve	Bspline Curve
Cone	Flattened wire frame
Surface	Group
Dimension	Dimension

Text	Text
Text Node	Text
Cell	Group
Shared Cell	Symbol (Shared Embed)
Multiline	Group
Point String	Points
Reference	Reference
Raster	Reference
Database Linkages	Dynamic Attribute set on the element
Tags	Dynamic Attribute set on the element
User Data	Dynamic Attribute set on the element

Accuracy of MicroStation Data During Export

When you translate SmartSketch files into MicroStation data, you are concerned with either Visual fidelity, Data fidelity, or both.

Visual Fidelity

Visual fidelity is a phrase used to describe the visual accuracy of a picture after translation. SmartSketch preserves the visual accuracy of a file during export to MicroStation by providing support for the following different areas:

Colors

When SmartSketch colors are saved to a .dgn file, a color table containing matching colors is created and attached to the resulting .dgn by default. If a seed file with a color table attached is specified on the MicroStation Export Options dialog box (accessed through the Foreign Data tab after you select the Options command on the Tools menu), SmartSketch colors are automatically mapped to the closest RGB equivalent colors in that table.

Widths

SmartSketch default widths are pre-mapped to their most equivalent MicroStation widths on the Line Width tab of the MicroStation Export Options dialog box (accessed through the Foreign Data tab after you select the Options command on the Tools menu).

Linestyles

SmartSketch line styles are mapped on the Linestyle tab of the MicroStation Export Options dialog box. If styles from a referenced MicroStation .RSC file have been used in the SmartSketch file being exported, the line style names are transferred to the resulting .dgn file. If a style exists in the MicroStation application by the same name, that style will be used. If there is no style in the MicroStation application by the same name, MicroStation assigns a default style.

Fill Styles

Basic fill styles such as Normal are automatically recreated as MicroStation associative hatch patterns in the resulting .dgn files. Fill styles that use symbols

such as Earth (in AEC templates) are exported as individual elements in the resulting .dgn file.

Font Styles

SmartSketch default font styles (TTF) are pre-mapped to their most equivalent MicroStation font styles on the Font tab of the MicroStation Export Options dialog box. If the font is not listed in the mapping table on the font tab, it defaults to font 3 (Engineering) in the resulting MicroStation file. If font styles from a referenced MicroStation .RSC file have been used in the SmartSketch file being exported, the font style names are passed into the resulting .dgn file, provided it has not been mapped in the font mapping table. Mappings override automatic behaviors. If a font style exists in the MicroStation application by the same name, that font style will be used. If there is no font style in the MicroStation application by the same name, MicroStation assigns the default font 3 (Engineering) style.

References

MicroStation .dgn files that are referenced in SmartSketch will be saved as MicroStation reference files in the resulting MicroStation file. Raster reference files in SmartSketch will be saved as MicroStation raster files in the resulting MicroStation file. Other reference files (.igr, .dgn, .or dxf files) are not currently supported during the export to MicroStation. Inserted objects such as .XLS, .DOC, or .BMP are not currently supported during the export to the MicroStation file format.

Multiple Sheets

Only the active sheet and background sheet in a SmartSketch file is exported to a MicroStation file. Exporting multiple sheets from a document is not currently supported.

Tip The active background sheet graphics are merged into the resulting .dgn file.

Data Fidelity

Data fidelity is a phrase used to describe how accurately the data is preserved when compared to its original form in the application that created it. SmartSketch preserves the data accuracy of a MicroStation file during export by mapping data to its most equivalent form in MicroStation. SmartSketch does not support exporting data such as attribute sets that can be associated with elements in the file.

SmartSketch preserves data accuracy of a MicroStation file by providing support for the following areas:

Units

If a seed file is specified on the MicroStation Export Options dialog box, then the units of that seed file are used in the resulting file. Otherwise, the following chart indicates what happens to SmartSketch units as they are exported to MicroStation units.

igr Primary Units	*MSTN MU*	*MSTN Resolution*
inches	IN	1000:254
feet	FT	12:8000
meters	M	1000:10
millimeters	MM	1000:100
centimeters	CM	1000:100
manometers	NM	Seed file values unchanged
yards	YD	Seed file values unchanged
miles	MI	Seed file values unchanged
kilometers	KM	Seed file values unchanged
tenths	TE	Seed file values unchanged
hundreds	HN	Seed file values unchanged
thousands	TH	Seed file values unchanged
poles	PL	Seed file values unchanged
rods	RD	Seed file values unchanged
links	LK	Seed file values unchanged
chains	CH	Seed file values unchanged
points	PT	Seed file values unchanged
furlongs	FR	Seed file values unchanged

Coordinates

SmartSketch data is automatically positioned to matching coordinate positions when you translate to the MicroStation files.

Layers to Levels

SmartSketch layer names are preserved as MicroStation level names and numbers, but the color and linestyle settings (level symbology) are not. The color and linestyle layer settings are applied directly to the resulting elements.

Tip Any translated levels past Level 63 are combined together on Level 63.

The following is an element to element map when you translate files using Save As on the File menu.

SmartSketch	*MicroStation*
Line	Line
Ellipse	Ellipse
Arc	Ellipse
Point	Line
Linestring	Linestring
Connector	Linestring
Bspline Curve	Bspline Curve
Rectangle	Shape
Complex String	Complex Chain
Text	Text
Dimension	Dimension
Symbol	Shared Cell
Leader	Dimension
Group	Group
SmartLabel	Elements
Parametric Symbols	Elements

Multi Representation Symbol	Elements
.dgn Reference	.dgn Reference
.igr, dwg, or .dxf Reference	Ignored (no equivalent)
OLE Objects (.doc, .xls, etc.)	Ignored (no equivalent)
Attribute sets	Ignored
Raster Reference	Raster Reference

MicroStation Cells in SmartSketch

To use MicroStation cells as symbols in SmartSketch, the cells must first be converted into individual SmartSketch symbols (.SYM). On the File menu, the Open command allows you to open a MicroStation cell library. A directory will be created, with the same name as the cell library. The individual cells within the library will be saved as individual SmartSketch symbols in the new directory.

Tip All MicroStation Type 2 cells are imported into SmartSketch as a group. The group contains a dynamic attribute set that stores the name of the cell, its origin, and any other data characterizing the cell. There is, therefore, enough information stored in the group to reconstruct an element back into a cell to be exported in the future.

You can drag these SmartSketch symbols from the Symbol Explorer or Windows Explorer into other SmartSketch documents.

Options for the cell library units (SU and PU) can be set on the MicroStation Import Options dialog box. You can access this dialog box by clicking the Options command on the Tools menu. Then, on the Foreign Data tab, you can enter the appropriate working units in the SU: and MU: fields.

SmartSketch symbols are saved as MicroStation cells during translation from SmartSketch to MicroStation formats.

MicroStation Construction Class Information

When you open a MicroStation document, the construction class information is preserved with layer names. When importing the document, the software creates special layers for construction elements. The layers are named so that you can recognize the layers, including the layer for the construction element. For example, level 6 would become Constr. 6 and level Border would also become Constr. Border. You can also save these layers back out to the MicroStation document format.

Procedures to Explore in On-line Help

- Establish Relationships on a MicroStation Reference File
- Open a MicroStation Cell Library
- Add MicroStation Linestyles to a Document
- Open a MicroStation Document
- Place MicroStation Information in the Document

CHAPTER 20

AutoCAD Documents in SmartSketch

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Opening and Saving AutoCAD Files

Opening and Saving AutoCAD Files

SmartSketch supports opening and saving AutoCAD files in two ways: you can either select an AutoCAD file and translate it into equivalent SmartSketch elements, or you can create a reference file from an AutoCAD file.

If you use the method for selecting an AutoCAD file and translating it into equivalent SmartSketch elements, you can click Open on the File menu. Once an AutoCAD file has been opened using this method, the resulting file is in a SmartSketch format (IGR).

Tip If an AutoCAD file contains paper space data, the resulting SmartSketch file will contain two sheets: one sheet represents the paper space data, and is named paper space; the other sheet represents model space data, and is named model space. The paper space views are recreated as reference views of the model sheet on the paper space sheet.

If you use the method for creating a reference file from an AutoCAD file, you can either click Object on the Insert menu, or you can drag and drop from the Symbol Explorer or Windows Explorer. This enables you to display and, if AutoCAD is available on your PC, edit the AutoCAD file while working in SmartSketch. You can also select the view position of a 3-D AutoCAD file upon placement. Once an AutoCAD file has been referenced using this method, you can then locate and establish relationships between the new information and elements on the drawing sheet.

Note The Options command on the Tools menu displays the Options dialog box. By selecting the Foreign Data tab and setting the format to AutoCAD, you can configure the settings for importing AutoCAD data that applies to both of these methods.

Caution Roundtrip of AutoCAD data (importing AutoCAD data to SmartSketch data and exporting back to AutoCAD data) is currently supported visually. Data fidelity for information such as attributes, database linkages, and Xdata are not supported during export to the AutoCAD file format.

Other Methods for Importing AutoCAD Data

You can use the following methods to move AutoCAD (.DWG or .DXF) files into SmartSketch:

- Cutting and pasting.
- Dragging and dropping.
- Inserting the information as an object.

When you open an AutoCAD document that has references to other documents, those referenced documents show up as well. You can locate referenced documents in the current document.

Tip You can also use the Open command on the File menu to translate and open documents.

Cutting and Pasting

You can select the information in the AutoCAD document and then cut or copy the information and paste it into SmartSketch with the Paste Special command.

Dragging and Dropping

You can select the information in the AutoCAD document and then drag and drop it into SmartSketch. In the Windows Explorer, select the AutoCAD document and then drag and drop it into SmartSketch. The document will be embedded in the drawing by default. Right before you drop the document, you can embed the document in SmartSketch by pressing the CTRL key. If you press CTRL + SHIFT, the document is linked. When you drop the document, handles appear that allow you to scale the object. The foreign document is placed into SmartSketch and behaves much like a symbol, allowing you to click to place it or move it around.

Tip You can set placement options by clicking the Options command on the Tools menu. Then, you can set the options you want on the Reference File tab of the Options dialog box.

Inserting as an Object

Inside SmartSketch, you can insert the entire foreign document with the Object command on the Insert menu. On the Insert Object dialog box, you can set the option to link or embed the document. 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 geometry of the inserted document is recognized; 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.

Tip The Object command inserts any OLE 2.0 enabled object, such as a Word or .AVI document, or CAD document, such as .DWG. The inserted object can then be edited by double-clicking it.

Saving SmartSketch to AutoCAD

You can save a SmartSketch file as an AutoCAD file. Before you do this, you can adjust the translation options with the Options command on the Tools menu. On the Foreign Data tab, you can click the appropriate Export Options button.

When you save a SmartSketch document as a .DWG or .DXF document, the active sheet is saved.

You do not have to map layers for AutoCAD documents; the layer names are imported or exported as named in the native document. If a SmartSketch layer has not been mapped, it is automatically placed on the next available layer or level in the foreign document until the last level or layer has been encountered.

Caution If you save a SmartSketch document with raster reference files as AutoCAD documents, only the files of the same type (.DWG) can be exported back into SmartSketch.

Accuracy of AutoCAD Data During Import

When you translate AutoCAD data into SmartSketch, you are concerned with either visual fidelity or data fidelity or both.

Visual Fidelity

Visual fidelity is a phrase used to describe the visual accuracy of a picture after it has been translated. SmartSketch preserves the visual accuracy of an AutoCAD file by providing support for the following seven different areas:

Colors

AutoCAD colors are automatically mapped to the closest matching red, green, blue (RGB) color in the SmartSketch color table.

Tip Colors that were used in AutoCAD to determine plotting pen widths can also be mapped to a SmartSketch width by using the AutoCAD Color option on the Line Width tab of the AutoCAD Import Options dialog box.

Widths

AutoCAD polylines are the only type of entity that support a width. Polyline widths are automatically mapped to an exact matching in SmartSketch model width. All other entities such as lines, arcs, and circles in AutoCAD do not support width. For the entities that do not support width, a default width is applied (all elements in SmartSketch have a width). You can make changes to any of the default mappings to define your own mapping with the Line Width tab of the AutoCAD Import Options dialog box (accessed through the Foreign Data tab after you select the Options command on the Tools menu). SmartSketch provides a default set of line widths that correspond to standard metric pencil leads: .13, .18, .25, .35, .5, .7, 1.0, 1.4, and 2.0.

Linestyles

AutoCAD default line styles are pre-mapped to the most equivalent SmartSketch line styles on the Linestyle tab of the AutoCAD Import Options dialog box. Line styles other than the default line styles or any complex line styles containing shapes are mapped to continuous unless they are specifically mapped on the Linestyle tab.

Hatches

AutoCAD default hatches are automatically mapped to matching fill styles in the resulting SmartSketch files. Hatches other than the defaults delivered with AutoCAD are stroked into their primitive elements to preserve their visual accuracy

Font Styles

AutoCAD default Font styles are pre-mapped to their most equivalent SmartSketch Font styles, True Type Fonts, on the Font tab of the AutoCAD Import Options dialog box. Other fonts default to the Arial true type font delivered with Windows. If the font is not listed in the mapping on the tab, it defaults to Arial.

Xrefs

AutoCAD Xrefs are imported as SmartSketch reference files, which use the automatic defaults or same options for configuring how color, width, line styles, hatches, and fonts are handled for visual accuracy.

Tips

- ImageScape LT must be downloaded to support raster reference files.
- None of the data that is part of the reference file can be deleted or altered using SmartSketch commands. However, if AutoCAD software is located on your machine, then you can click Open on the Shortcut menu over the reference file. This will open the AutoCAD reference file in the AutoCAD application.
- If you establish relationships between elements in the SmartSketch file and a reference file, the relationships will be removed if you change the position of elements in the source file and then update the reference file in SmartSketch.
- If you change the inserted information (copied from AutoCAD), the original information does not change.

Paper Space

If paper space was active when an AutoCAD file was saved, the file containing paper space data causes two sheets to be created when you click Open on the File menu. One sheet represents the paper space data and is named Paper Space. The other sheet represents model space data and is named Model Space. The paper

space views are recreated as reference views of the model sheet on the paper space sheet.

Tip AutoCAD files that are reference files instead of files translated using Open on the File menu, do not support the display of paper space data; instead, the model space information is displayed.

Configuring the Fit to Sheet Feature

SmartSketch automatically calculates the sheet scale when you import AutoCAD documents so that the range of graphics automatically fits within the lower left-hand corner of the drawing sheet. If you do not want the sheet scale automatically calculated, then you must manually modify the INI file.

To modify the INI file, you click Explore on the Start menu and go to the directory where SmartSketch resides. You can use any text editor to modify the file. Double-click itacad. In the [Options] section of the file, set Ignore sheet scale = 0.

Tip You can create a template for calculating sheet scale at the scale you want and insert an AutoCAD document to base the template on. On the Sheet Setup dialog box, you set the Drawing Scale and Paper Units that you want

Using AutoCAD Fonts and Styles in a SmartSketch Document

Fonts

SmartSketch supports the system TrueType fonts delivered with Windows and supplies additional ANSI and ISO TrueType™ engineering fonts. When you open a .dwg document, SmartSketch provides a default mapping to determine which TrueType font will be assigned to which AutoCAD font. You can make changes to this mapping to define your own mapping. You can change the mapping with the Options command on the Tools menu. On the File Locations tab, you can set the paths to the font resources by double clicking on Fonts 1, 2, and 3.

Data fidelity

Data fidelity is a phrase used to describe how accurately the data is preserved when compared to its original form in the application that created it. SmartSketch

preserves the data accuracy of an AutoCAD file by providing equivalent element types that entities are mapped to. SmartSketch also provides support for non-displayed data such as attributes on the resulting elements.

SmartSketch preserves data accuracy of an AutoCAD file by providing support for the following areas:

Units

AutoCAD files are considered unit-less files. The File Units setting for AutoCAD import data on the Options dialog box is used to determine the units for translated AutoCAD data or referenced AutoCAD data.

Coordinates

AutoCAD data is automatically positioned to the matching coordinate positions when you translate the AutoCAD files (click Open on the File menu). Referenced AutoCAD file coordinate positions are determined by the settings on the Reference File tab on the Options dialog box.

Tip You can use the Coincident option to import the data in its original coordinate position at a 1 to 1 scale.

Layers

Within a SmartSketch document, there are an unlimited number of layers per sheet. AutoCAD layer names are preserved as SmartSketch layer names, but the color and linetype settings are not. The color and linestyle settings are applied directly to the resulting element.

Caution In AutoCAD, colors, line styles, and line widths are applied on a layer-by-layer basis. When you select a different layer, the colors, line styles, and line widths change. In SmartSketch, colors, line styles, and line widths are applied directly to each individual element or object. When you select a different layer, the colors, line styles, and line widths are not affected. When you export back to AutoCAD, colors, line styles, and line widths are applied on an entity basis.

AutoCAD entity colors & linetypes are determined one of three ways on an element:

- Byblock–a property that specifies that an entity inherit the color or linetype of a block containing it. SmartSketch does not support the entity concept so the Byblock properties are applied directly to the resulting element.

- Bylayer–a property that specifies that an entity inherit the color or linetype of its associated layer. SmartSketch does not support the entity concept so the Bylayer properties are applied directly to the resulting element.
- Byentity–occurs when colors and linetypes are directly applied to the entity and override any Byblock or Bylayer properties. SmartSketch supports this concept for all of its elements and preserves Byentity properties for colors and linestyles.

Blocks

When translating a .dwg document, the default is to translate cells or blocks into symbols. You can change this default with the Options command on the Tools menu so that cells or blocks are embedded groups. This translates cells and blocks into embedded symbols, or groups, in SmartSketch.

When you import an AutoCAD document that blocks, a temporary folder is created in the Temp folder on your hard drive. The temporary folder contains the symbols that are embedded in the AutoCAD document. By default, these symbols are deleted after the document is imported into the drawing sheet. You can change a setting in \\SMARTSKETCH\PROGRAM \ITACAD.INI to save the symbols instead. In Notepad, open the .INI file and change the following line in the Options section: Delete Symbol Definitions = 0.

3D Data

All 3D data is flattened into a 2D view when you translate files using Open on the File menu. The 2D view orientation is determined by the Orientation setting in the Import area of the Options dialog box.

The following is an entity to element map when you translate files using Open on the File menu.

AutoCAD	*SmartSketch*
Line	Line
Circle	Ellipse
Polyline (lines only)	Linestring
Polyline (lines & arcs)	Complex String
Arc	Arc
Trace	Line

Point	Point
Face3d	Line
Solid	Linestring
Block	Symbol
Multiline	Linestring
Spline	Bspline Curve
Dimension	Dimension
Text	Text
Paperspace Viewport	Reference
Xline	Line
Ray	Line
Database Linkages	Dynamic Attribute set on element
Attributes	Dynamic Attribute set on element
Ellipse	Ellipse
Region	Ignored
Solid3d	Ignored
Mtext	Text
Leader	Leader
Tolerance	Group
Xdata	Dynamic Attribute set on the element
Hatch	Fill
Image	Reference

Accuracy of AutoCAD Data During Export

When you translate SmartSketch files into AutoCAD data, you are concerned with either Visual fidelity or Data fidelity or both.

Visual Fidelity

Visual fidelity is a phrase used to describe the visual accuracy of a picture after it has been translated. SmartSketch preserves the visual accuracy of a file during export to AutoCAD file by providing support for the following seven different areas:

Colors

SmartSketch colors map to the closest matching red, green, blue (RGB) color in the default AutoCAD color table.

Tip SmartSketch widths can be mapped to an AutoCAD color by using the Line Width tab of the AutoCAD Export Options dialog box. AutoCAD colors are used to determine plotting pen widths.

Widths

Every element in SmartSketch has widths applied that are exported as polyline objects with equivalent AutoCAD widths if the widths are equal to or greater than the polyline width threshold. The Polyline width threshold field is located on the Line Width tab of the AutoCAD Export Options dialog box. Whenever possible, elements that have widths less than the threshold are exported as AutoCAD entities that do not support width.

Linestyles

SmartSketch default line styles are pre-mapped to the most equivalent AutoCAD line styles on the Linestyle tab of the AutoCAD Export Options dialog box. Line styles other than the default line styles or any complex line styles containing shapes are mapped to continuous unless they are specifically mapped on the Linestyle tab.

Fill Styles

SmartSketch default fill styles are exported as matching hatch styles in the resulting AutoCAD files.

Font Styles

SmartSketch default font styles are pre-mapped to their most equivalent AutoCAD font styles (True Type Fonts (TTF)) on the Font tab of the AutoCAD Export Options dialog box. Other fonts default to the txt.shx font deliver with Windows. If the font is not listed in the mapping table on the font tab, it defaults to txt.shx in the resulting AutoCAD file.

Dimension Styles

SmartSketch dimension styles are preserved as AutoCAD dimension styles in the resulting dwg files.

Reference Files and OLE Inserts

AutoCAD dwg files that are referenced in SmartSketch are saved as AutoCAD Xrefs in the resulting AutoCAD file. Raster reference files in SmartSketch are saved as AutoCAD raster files in the resulting AutoCAD file. Other reference files that are not currently supported during export to AutoCAD include .igr, .dgn, and .dxf, files. OLE inserted objects such as .XLS, .DOC, or .BMP are not currently supported during export to the AutoCAD file format

Multiple Sheets

Only the active sheet in a SmartSketch file is exported to an AutoCAD file. Exporting multiple sheets from a document is not currently supported.

Data fidelity

Data fidelity is a phrase used to describe how accurately the data is preserved when compared to its original form in the application that created it. SmartSketch preserves the data accuracy of an AutoCAD file during export by mapping data to its most equivalent form in AutoCAD. SmartSketch does not support exporting data such as attribute sets that can be associated with elements in the file.

SmartSketch preserves element and document data support for the following areas:

Units

The subunits of the file being saved as a dwg file determines the model size of the graphics in the resulting AutoCAD file

Coordinates

SmartSketch data is automatically positioned to matching coordinate positions when you translate to AutoCAD.

Layers

SmartSketch layer names are preserved as AutoCAD layer names, but the color and linetype settings are not. The color and linestyle settings are applied directly to Byentity, the resulting element In AutoCAD.

AutoCAD entity colors & linetypes are determined one of three ways on an element:

- Byblock—a property that specifies that an entity inherit the color or linetype of a block containing it. SmartSketch does not support the entity concept so the Byblock properties are applied directly to the resulting element.
- Bylayer—a property that specifies that an entity inherit the color or linetype of its associated layer. SmartSketch does not support the entity concept so the Bylayer properties are applied directly to the resulting element.
- Byentity—occurs when colors and linetypes are directly applied to the entity and override any Byblock or Bylayer properties. SmartSketch supports this concept for all of its elements and preserves Byentity properties for colors and linestyles.

The following is an entity to element map when you translate files.

SmartSketch	*AutoCAD*
Line	Line
Ellipse	Circle
Arc	Arc
Point	Point
Connector	Polyline
Linestring	Polyline
Bspline Curve	Polyline
Rectangle	Polyline
Complex String	Polyline

Reference	Xref
Text	Text
Leader	Leader
Dimension	Dimension
Group	Entity
Symbol	Block
Smart Label	Entities
Parametric Symbols	Entities
Multi-representation Symbol	Entities
Raster Reference	Raster Reference

Procedures to Explore in On-line Help

- Open an AutoCAD Document
- Place AutoCAD Information in the Document
- Establish Relationships on an AutoCAD Reference File
- Create a Template to Insert an AutoCAD Document

CHAPTER 21

Authoring Symbols

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Planning Your Symbol

After you have determined the need for a symbol, you need to decide how you want that symbol to behave. The goal is to create a symbol that acts the way a user expects it to. .

When you create a symbol, it displays the following default behaviors:

- 90 degree rotation angles
- Automatic alignment with a target element or object when the symbol is placed in a document
- Automatic association with a target element or object.

Tip You can change these default behaviors with commands on the Symbol Authoring Tools toolbar

Checklist for Consideration

When planning your symbol, the first item to consider is what happens when you add text to a symbol. Do you want to add text to the shape? If so, you should consider the position and appearance of labels, text boxes, or balloons. A Smart Label can be associated with attributes that you define. You can also define symbols that resize as a user adds text to the symbol.

The second item you should consider is what template will you want to use with the symbol. The scale of the template affects how the symbol looks when it is dragged in from the Symbol Explorer.

The next item for you to decide is what happens when you manipulate the symbol. Should the symbol size, mirror, flip, and rotate freely? Do you want the symbol to move with an element? You can set different options on the Define Symbol Properties dialog box. You can create special points on a symbol to determine how the symbol attaches to connectors, other symbols, or elements in a document.

How smart should you make the symbol? You can create symbols that have text and parametric attributes. You can create text attributes using SmartText by clicking the SmartText button on the Symbol Authoring Tools toolbar. You can create parametric attributes by adding driving dimensions to the symbol and then adding the dimensions to the Parameters tab on the Define Symbol Properties dialog box.

Do you want users to access special programs when they drop a symbol or drag it? You can set options for these and more on the Define Symbol Properties dialog box.

Do you want users to choose from a list of representations for the same symbol? You can accomplish this by clicking the Symbol Representation button on the Symbol Authoring Tools toolbar.

Tips for Developing Your Symbol

Developing your symbol in an orderly, thoughtful fashion saves you time in the long run. You can consider the following suggestions:

- Make notes on what you want the symbol to do. What are your requirements? If the symbol is to be part of a larger set, define the common requirements for the entire set first.
- Draw a preliminary version of the symbol and format it to look the way you want. Experiment with it. What behaviors do you want when the symbol is rotated or scaled?
- If you want to add parametric attributes, experiment with different formulas in the Variable Table. Create one formula at a time and test its effect on the symbol. Keep notes as you go. You might want to copy the symbol each time you try a new formula to keep a record of your experimentation.
- You should test the symbol for usability by allowing a coworker to play around with the symbol. Does the symbol meet the coworker's expectations?

After you have considered all of these items, you should recreate the symbol from scratch. This is the best way to ensure that the symbol contains no obsolete parameters, attributes, or other behaviors that would clutter the symbol or increase its size.

Using Blank Color and Fill with Symbols

Symbols are actually groups of graphic elements with defined behaviors and properties that are saved in an .SYM document. You can enhance a symbol's display and performance during symbol design and creation. Incorporating a fill with blank color into the symbol allows the symbol, when placed, to mask out elements such as lines or connectors that intersect the symbol. This gives the appearance that the underlying lines have been broken at the edges of the symbol. In reality, the lines still exist beneath the symbol.

You can apply a fill to any closed boundary. Placing many fills in a document can significantly increase the size of the document. A single fill can be used for the symbol by filling an area that represents the outer edges of the symbol and displaying the detailed graphic linework on top of the fill.

The blank color can be used with any graphic object besides fill, so experiment and you might find other uses for this masking capability. .

Controlling How Symbols Expand

You can create symbols that shrink and grow when you enter text. You can also determine how users can resize a symbol by specifying parametric handles or scale handles. Symbols with these types of handles are parametric symbols. You can define parameters with the tools on the Symbol Authoring Tools toolbar. ..

Integrating Programs with Your Symbol

You can kick off special processes from your symbol when a user performs an particular action, such as double-clicking the symbol, dragging it, or even modifying the symbol. The special process can be a .DLL, .OCX, or .EXE file that you created in other programming software. You can select an action from the Standard Actions tab on the Define Symbol Properties dialog box.

You can also specify special processes that you can execute when you select a symbol, click the right mouse button, and then click a custom command on the shortcut menu. The command names of the processes that you define appear at the top of the shortcut menu. You define these processes on the Custom Actions tab of the Define Symbol Properties dialog box.

Controlling Display Properties

You can control display properties for symbols in various ways. You can change the properties of various elements in the shape by applying styles or setting options such as line weight and color on the ribbon bar. You can also determine what handles appear on the symbol by setting options on the Define Symbol Properties dialog box.

On this dialog box, you can also set options for how the symbol displays when a user drags it into the document. For example, if you want the symbol to display as separate elements, and not a unified symbol, you can click the Discrete Objects option on the Behaviors tab of the Define Symbol Properties dialog box.

Controlling Connectors for Symbols

Connect points allow you to determine how a connector line attaches to a symbol. You can define and remove connect points with the Place/Edit Connect Points button on the Symbol Authoring Tools toolbar.

You can specify the angle at which a connector attaches to a symbol. When you attach a connector to a symbol with a connect point, the connector automatically attaches to the point at the angle in radians that you set in the attributes of the connect point. For example, you can place a connect point with an angle value of 1.5708 radians, or pi divided by two. Then, the connector automatically attaches to the connect point at an angle of 1.5708 radians.

You can specify the angle by entering the value that you want in the Current Connect Points Angles box on the Place/Edit Connect Points dialog box.

Creating Flexible Symbols with Handles

Your symbol might need more control than users can implement by simply selecting the symbol and dragging. You can add different types of handles to give the user more control over the symbol.

Scales and Symbols

The scale of a template can affect the visual size of a symbol when you drag the symbol into a document. For example, the scale for the Architecture (metric) template is set to 1:100 mm on the Sheet Setup dialog box. Some symbols that you drag into documents based on that template appear small.

Managing Text in Symbols

You can add text to symbols by creating text boxes and balloons, placing text labels, or creating Smart Labels with Smart Text. Smart Text defines the text attributes for the symbol.

A label with Smart Text attributes is intelligent: Smart Labels with Smart Text read and write properties from an element or object. A label with Smart Text can assume the attributes of an element if the attributes for the element have the same name. Users can then edit the values of the text attributes in the Attribute Viewer. Users cannot change the names of the attributes.

Not all labels assume the text attributes of an element or object. Some labels contain shapes that grow automatically if you add text to the label. These are called text-driven symbols.

Procedures to Explore in On-line Help

- Create a Parametric Symbol
- Define a Text-Driven Symbol