SolidWorks

SolidWorks® 98Plus
User's Guide

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SolidWorks Fundamentals

What is SolidWorks 98Plus?

SolidWorks® 98Plus is mechanical design automation software that takes advantage of the familiar Microsoft® Windows™ graphical user interface. This easy-to-learn tool makes it possible for mechanical designers to quickly sketch out ideas, experiment with features and dimensions, and produce models and detailed drawings.
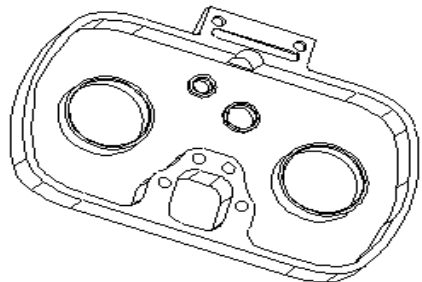
Design courtesy of Barrett Technology

This chapter discusses some basic concepts and terminology used throughout the SolidWorks 98Plus application. It provides an overview of the following topics:

❑ Installing the SolidWorks 98Plus software

❑ SolidWorks terms and document windows

❑ FeatureManager™ design tree

❑ Finding out what is wrong

❑ Right-mouse menus

❑ Selecting items

❑ Time saving tips

❑ Preparation for printing

❑ Rendering models with PhotoWorks™

❑ Feature recognition with FeatureWorks™

❑ Using the SolidWorks API development environment

Installing the SolidWorks 98Plus Software

System Requirements

- ❑ Microsoft Windows NT 4.0 (with Service Pack 3), Windows 95, or Windows 98

- ❑ Pentium®-based computer, or an Alpha-based computer

- ❑ 64 MB RAM minimum recommended

- ❑ Mouse or other pointing device

- ❑ CD-ROM drive

> **NOTE:** You should make backup copies of all SolidWorks documents (parts, assemblies, and drawings) created with a previous version of the software before opening them in SolidWorks 98Plus.

Required Information for Installation

The SolidWorks 98Plus CD installation program guides you through the installation procedure and asks for the following information:

- ❑ The type of installation:

  - • **Individual.** This installation is the most common choice. It should be selected for a computer that will not share its SolidWorks installation with any other computers, whether or not it is on a network.

  - • **Client/Server.** This installation should be selected for a computer that will run SolidWorks, and will also act as a server, sharing its SolidWorks installation with one or more other computers.

> **NOTE: The server and clients must be of the same type platform. There is no cross-platform installation.**

  - • **Server Only.** This installation should be selected for a computer that will *not* run the SolidWorks 98Plus application, but will only act as a server, sharing its SolidWorks installation with one or more other computers (SolidWorks clients). (You must reinstall if you want to change this selection.) With this option, the server does not need to be licensed to run the SolidWorks application, but each client must be licensed.

> **NOTE:** Before starting a SolidWorks update installation, it is important to ensure that *no* SolidWorks clients are running.

❑ **SolidWorks Serial Number.** For new SolidWorks customers or non-subscription customers, the serial number is located on the back of the SolidWorks CD-ROM case. Subscription customers should continue to use the serial number on the CD-ROM case from their first SolidWorks installation.

❑ **SolidWorks Registration Code:**

- If you are a SolidWorks subscription customer, the registration wizard provides a registration code during the SolidWorks 98Plus installation.

- If you are a new SolidWorks customer, you are asked to use the registration wizard during installation. This generates a registration form that you can email or fax to SolidWorks Corp. A registration code will be provided quickly by return email or fax. You have up to 30 days to use the product without the registration code.

- If you are a continuing, non-subscription customer, you must have a registration code to update to the SolidWorks 98Plus software.

❑ **PhotoWorks Serial Number.** If you are a new customer who has purchased the PhotoWorks photo-realistic rendering application, you need to enter the PhotoWorks serial number, located on the back of the CD-ROM case. Subscription customers should continue to use their original PhotoWorks serial number.

❑ **FeatureWorks Serial Number.** If you purchased the FeatureWorks feature recognition software, you need to enter the FeatureWorks serial number, located on the back of the CD-ROM case.

❑ **Enable or disable MCD.** This option applies only to Windows NT 4.0. You should enable MCD (Mini-Client Driver) if you are using a graphics card that has Mini-Client Driver support so that SolidWorks can take advantage of the card's ability to accelerate 3D OpenGL.

❑ **Allow or disallow model changes from drawing.** This option makes it possible for a user to disallow the ability to make changes to part or assembly dimensions from the drawing. (You must reinstall the SolidWorks software to change this option.) The default, and most common configuration, is to allow the part and assembly dimensions to be changed from the drawing.

Installing the SolidWorks 98Plus Software

The installation steps for an individual user, a server, or a client/server installation are the same. The answers to the questions asked during the installation may differ. For instance, a serial number is not required for a computer that is used exclusively as a server.

To install the SolidWorks software:

1   Insert the CD-ROM in the drive.

2   Within a few seconds the install wizard will begin the installation and guide you through the few questions that you have to answer.

The Client/Server Installation

The computer designated as the file server shares its SolidWorks installation with one or more other computers. However, before each of the clients can run the SolidWorks application, it is necessary to perform a SolidWorks client installation to prepare the client computer for sharing the application. The actual SolidWorks program files will not be installed on the client, but some setup procedures must take place. See the client installation discussed below.

The server and clients must be the same platform type; there is no cross-platform installation.

> **NOTE:** A Serial Number and Registration Code are required for each SolidWorks Client and Client/Server.

To perform a SolidWorks client installation:

1   Working on the SolidWorks server computer, make sure that the folder containing the SolidWorks installation is shared so that it is accessible to the client computers on the network.

2   Working on the SolidWorks client computer, browse to the subfolder under the SolidWorks installation directory called:

On Alpha Windows NT: **setup\alpha**

On Intel Windows NT, Windows 95, or Windows 98:   **setup\i386**

3   Double-click on the file named **setup.exe.**

4   The installation will begin and guide you through the few steps needed to complete the installation.

SolidWorks Service Packs

If you are a SolidWorks subscription customer, you can take advantage of SolidWorks service packs that are regularly posted on the SolidWorks web site. These service packs contain software updates and enhancements to the SolidWorks 98Plus software. To check for a new service pack, click **Help, Service Packs**, and click the **Check** button. Click the checkbox if you want the software to automatically check the SolidWorks web site for a new service pack once a week.

Converting Older SolidWorks Files to SolidWorks 98Plus

The format of SolidWorks files has changed somewhat with the development of SolidWorks 98Plus. For this reason, opening a SolidWorks file from an earlier release may take more time than you are used to experiencing. However, once the file has been opened and saved, subsequent opening time will return to normal.

The SolidWorks Conversion Wizard provides a way for you to automatically convert all of your SolidWorks files from an earlier version to the SolidWorks 98Plus format. Depending on how many files you have, the conversion process may take a while, but once it is done, you will not experience the slow initial opening of your files.

To access the Conversion Wizard, click the Microsoft **Start** button, select **Programs**, and select the **SolidWorks 98Plus** program group. Click **Conversion Wizard**. When the conversion utility begins, it offers you the choice of backing up all of your files before the conversion. If you choose to backup your SolidWorks files, the Conversion Wizard copies the files to a sub-folder named "Solidworks Conversion Backup." The wizard asks you for the location of the files to be converted, and leads you through the simple process.

At the end of the conversion process, two report files exist in the folder to which you directed the conversion. *Conversion Wizard Done.txt* contains a list of files that converted; *Conversion Wizard Failed.txt* contains a list of files that did not convert. Double-click the file names to open them in Notepad.

Visiting the SolidWorks Web Site

If your PC is connected to the Internet, you can visit the SolidWorks web site after your installation is complete. You can learn more about the SolidWorks company and product by clicking in the **Help** menu on the main SolidWorks window.

To access the SolidWorks web site:

1 Click **Help, About SolidWorks 98Plus**.

2 Click the **Connect** button to visit the SolidWorks web site.

You can learn about **What's New**, **Technical Support**, **VARs and Distributors**, and you can visit the SolidWorks **Design Gallery**, among other things.

Basic Concepts

❑ **A SolidWorks model consists of parts, assemblies, and drawings.**

- Typically, you begin with a sketch, create a base feature, and then add more features to your model. (You can also begin with an imported surface or solid geometry.)

- You are free to refine your design by adding, changing, or reordering features.

- Associativity between parts, assemblies, and drawings assures that changes made to one view are automatically made to all other views.

- You can generate detail drawings or assemblies at any time in the design process.

Section A-A

MainSheet
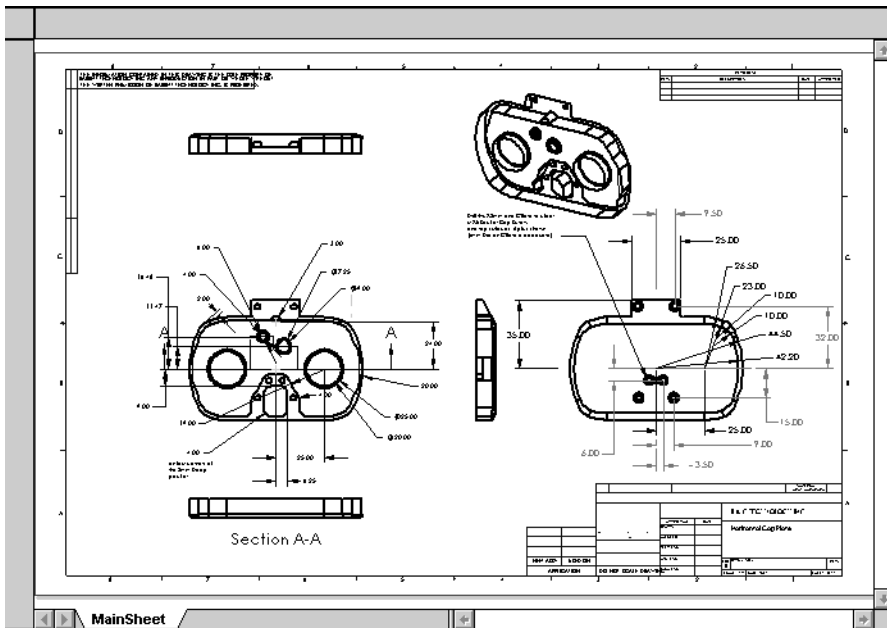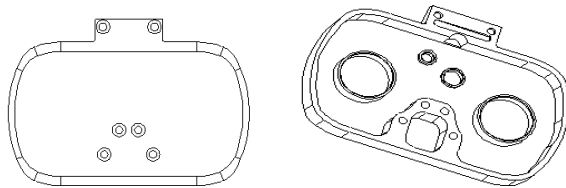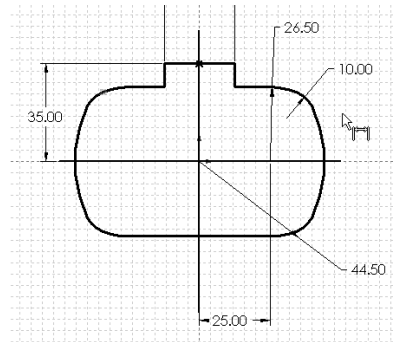
❑ **The SolidWorks 98Plus application lets you customize functionality to suit your needs.**

Click **Tools, Options** on the top menu bar to display tabbed pages of available options. The options pages are described in detail in Appendix A; the following describes some ways in which you can customize the SolidWorks 98Plus application:

Option page name:	Available options on page:
Color	Colors for lines, faces, features; drawing and drawing view borders; colors associated with the radii of curvature on models
Crosshatch	Crosshatch pattern and its scale and angle
Detailing	The dimensioning standard, dimension precision, and tolerances; detailing options such as arrow, witness line, balloon style or virtual sharp representation; default annotations and their properties; font size and style; text placement, and so on
Drawings	Drawing sheet size, scale, projection type, model edge display, automatic update of BOM and model views, and inferencing behavior when dragging detail items
Edges	Display of hidden edges types, tangent edge display, edge display while in shaded mode, repainting and highlighting behavior
Export	Options for IGES, Parasolid, DXF/DWG, ACIS, VRML, TIFF and STL output
External References	Preferences for the way part, assembly, and drawing documents are opened and referenced; defines search paths for external documents, palette features, and palette parts
General	Optional behaviors when in a sketch or model, FeatureManager design tree behavior, angle increment and speed of view rotation, and behavior of over defined dimensions, number of backup copies per document, AutoRecover, and automatic rebuild of assembly on load
Grid/Units	Grid display, spacing, and snap behavior; length unit types and angular unit types; spin box increments
Import	Default disposition of problems that may occur when importing IGES files or files in other formats
Line Font	Style and weight of edge lines for selected edge types
Material Properties	Material properties for the current part
Performance	Display quality for shaded, wireframe, and transparent models; curvature generation on demand or automatic; behavior of lightweight parts in assemblies
Reference Geometry	Define the display filter for reference geometry, and specify the default names for planes in part or assembly documents

❑ **The Registry Copy Tool lets you share customized options with other SolidWorks users.**

After you customize your SolidWorks 98Plus software, you can use the **Copy Options Wizard** to copy the options information set by one user and specify the same options on the machines of other users.

***To use the* Copy Options Wizard:**

1 From the **Programs** list, select **SolidWorks 98Plus**, **Copy Options Wizard.**

2 To capture your SolidWorks options information from the registry, click **An existing user** and select your name from the scroll list. Click **Next**.

3 Specify the destination for the options information:

- **Users or Computers on the network.** As a system administrator on Windows NT, you can copy the SolidWorks options directly to other selected users/computers on the network.

– or –

- **Create a Registry file.** You can create a *.reg file that you can use with the standard Windows NT, Windows 95, or Windows 98 "regedit" utility to load the SolidWorks options on the computers of other users.

❑ **The SolidWorks application saves your work for you.**

The **AutoRecover** option automatically saves information about your active part, assembly, or drawing document so you will not lose your work in case of a system crash. To set this option, click **Tools, Options, General** and select **Save AutoRecover info every (*n*) changes**. Use the scroll arrows to specify the number of changes that should occur before information is automatically saved. See *Appendix A*, **General Options** on page A-15.

Getting Help

If you have questions while you are using SolidWorks 98Plus, you can find answers in several ways:

❑ SolidWorks online help provides assistance while you are working.

- Click [?] or the **Help, SolidWorks 98Plus Help Topics** on the main toolbar to access the help system. Use the help system's **Table of Contents**, **Index**, or **Find** word search tool to locate the topic for which you need help.

- To get help about a specific, active dialog box, click the **Help** button in the dialog box or press the **F1** key.

❑ For helpful hints, click **Help, Tip of the Day**. To see a tip each time you start SolidWorks 98Plus, click **Show Tips at Startup** in the **Tip of the Day** dialog box.

❑ **Tooltips** are available to provide information about icons on toolbars or dialog boxes. When you hold your cursor over an icon for a brief time, a tooltip appears to display the name of the icon.

❑ As you move your cursor over an icon or click a menu item, the **Status Bar** at the bottom of the SolidWorks window provides a brief description of the function of the icon or menu item.

❑ The *Learning to Use SolidWorks 98Plus* tutorial provides much detailed information about using SolidWorks to create parts, assemblies, and drawings.

Opening a SolidWorks Part, Drawing, or Assembly

This section describes how to open a new or existing part, drawing, or assembly document.

Opening a New SolidWorks Document

To open a new part, drawing, or assembly document:

1 In a SolidWorks window, click[1] 🗋 on the Standard toolbar, or **File, New.**

2 From the **New** menu, select **Part, Assembly,** or **Drawing** and click **OK**.
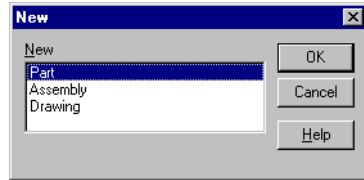
Opening an Existing SolidWorks Document

To preview and open an existing part, drawing, or assembly document:

1 In a SolidWorks window, click **File, Open**. Use the browser to select the part, drawing, or assembly.

2 Select **Open as read-only** if you want the document to be opened in read-only mode. This allows other users to have write-access to the part at the same time.

3 Select the **Preview** option to see a preview of the part.

4 Select **Configure** to open the part in a specified configuration. See **Working with Part Configurations** on page 5-15 for more information.

5 Click **View-Only** to open the document only for viewing. (Only documents saved in SolidWorks 98 or SolidWorks 98Plus may be opened in **View-Only** mode.) If you are in a part or assembly document, you can change to editing mode by pressing the right-mouse button in the graphics area and selecting **Edit**.

6 If you are opening an assembly or drawing document, you can change the pathname from which referenced parts are taken by clicking **References**. In the dialog box that appears, select and enter the **New pathname,** and click **Replace**. See **Editing Referenced File Locations** on page 6-12.

7 Click **Open** to open the document.

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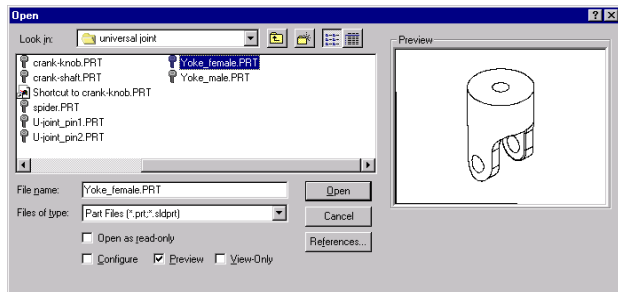
1 In this manual, "Click..." means point the cursor and press the *left* mouse button

Multi-Threaded Retrieval

Documents saved in SolidWorks 98 and SolidWorks 98Plus are retrieved in multi-threaded mode.

When you retrieve a large part, drawing, or assembly document, the document immediately displays in a View-Only state while the actual document and all its components are being retrieved in the background. During the View-Only state, you can use all functions supported in the SolidWorks Viewer (Zooming, Rotation, etc.), but you cannot switch to another document or start opening another document. After the retrieval of all necessary documents is completed, SolidWorks changes to the normal Edit state.

Opening an Existing Part from Explorer

You can preview and open a part, drawing, or assembly document directly from the Microsoft Explorer.
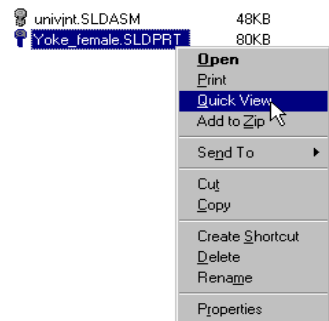
To view the part, drawing, or assembly without opening the document:

1 Right-mouse click the name of the part, drawing, or assembly in Explorer.

2 Select **Quick View** from the right mouse menu.

Quick View displays the part in a **SolidWorks Viewer** window.

You can also view thumbnail images of SolidWorks parts and assemblies in the Explorer as well as in the **File, Open** and **File, Save As** dialogs. The graphic is based on the view orientation of the model when the document was saved. To enable this functionality, click **Tools, Options**, and select the **General** tab. Select the option **Show thumbnail graphics in Windows Explorer**. See *Appendix A*, **General Options** on page A-15.
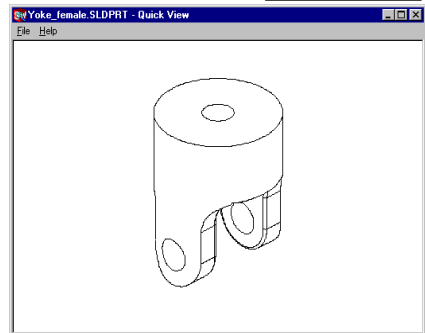
To open the part, drawing, or assembly from Explorer, do one of the following:

❑ Double-click on the name of the part, drawing, or assembly in Explorer.

❑ Right-mouse click on the name of the part, drawing, or assembly in Explorer and select **Open** from the right mouse menu.

❑ Drag and drop a part from Explorer to an open SolidWorks assembly window, an open and empty SolidWorks part or drawing window.

❑ Drag and drop an assembly from Explorer to an open SolidWorks assembly window.

❑ Drag and drop any SolidWorks document from Explorer into an empty area of the SolidWorks window, not occupied by another document window.

Dragging and Dropping Files from Internet Explorer

You can drag and drop hyperlinked SolidWorks part files from Internet Explorer, version 4.0 or higher, to:

- Feature Palette window

- new, empty part

- drawing or assembly document

- empty area of a SolidWorks window

If you drop the file into a drawing or an assembly, you are prompted to name the new file on your local disk.

SolidWorks Terms

This section illustrates some common SolidWorks terms.

Menu bar →

Toolbars →

Toolbars

Part document window

Drawing document window

Status bar

FeatureManager design tree

Graphics area

Model

Vertex

Edge

Axis

Origin

Plane2

Axis1

Plane

Face

Housing - Sheet1

Housing
- Annotations
- Sheet1
  - Template1
  - Drawing View1
  - Drawing View2
  - Drawing View3
- Sheet3
- Sheet4

FeatureManager
design tree

Drawing
views

Drawing
sheet

Sheet1 / Sheet3 / Sheet4

Sheet tabs

Document Windows

In the SolidWorks application, each part, assembly, and drawing is referred to as a document, and each document is displayed in a separate window. (Each drawing document can contain multiple drawing *sheets*, though.)

You can have multiple part, assembly, and drawing document windows open at the same time. Also, you can have multiple views of the same document visible at the same time.

To arrange the windows, you can drag them by the title bar, and resize them by dragging a corner or border. Also, from the **Window** menu, you can select **Cascade**, **Tile Vertically**, or **Tile Horizontally**.
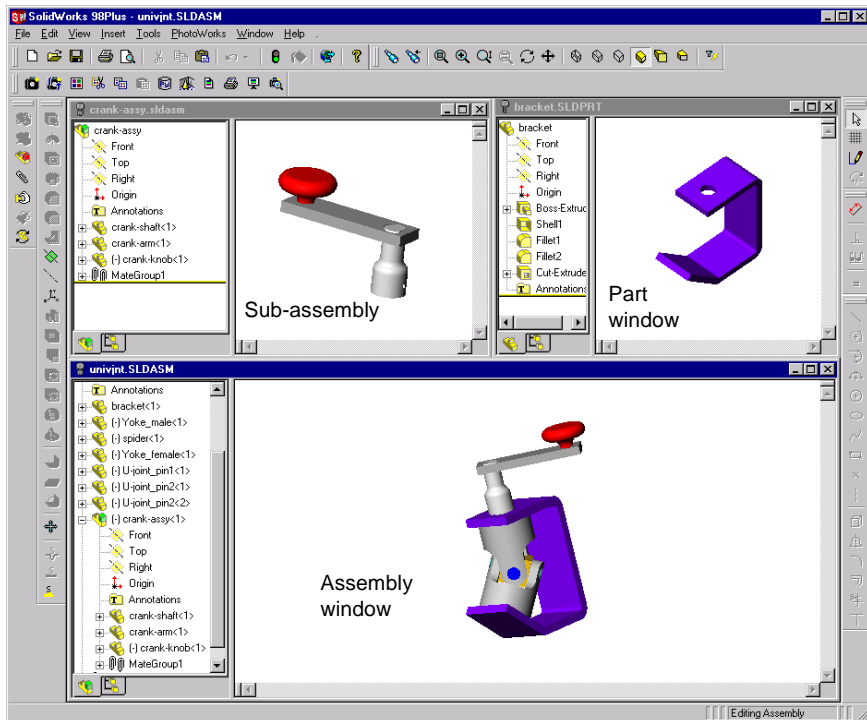
To organize your SolidWorks window, you can iconize open documents. Click the **Iconize** symbol in the upper right-hand corner of the document border. An icon appears in the lower part of the SolidWorks window. (If the icon(s) are not visible, resize the open documents as necessary.) Click **Window, Arrange Icons** to arrange them at the bottom of the SolidWorks window.
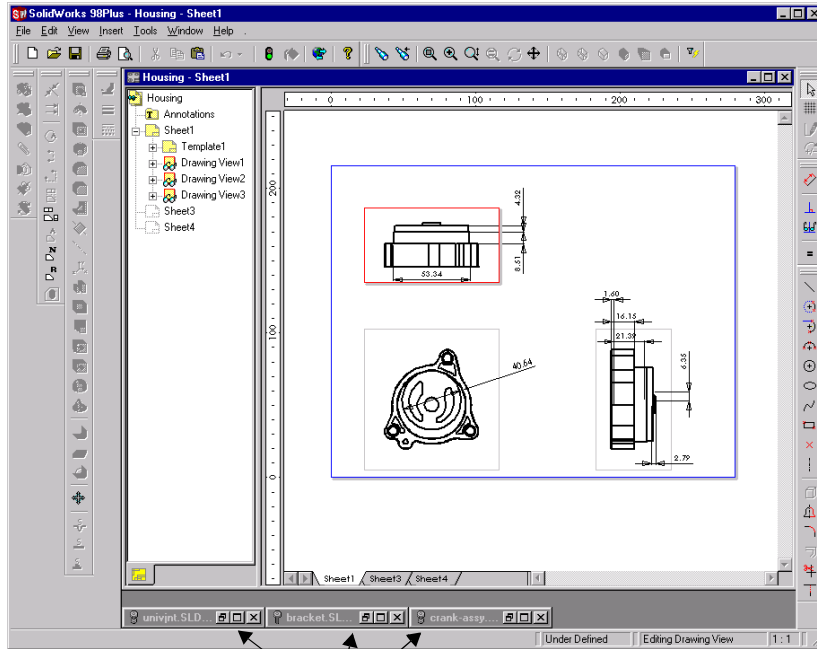
Document icons

Click **Window, Close All** to close all open documents. You are prompted to **Save** unsaved documents.

Toolbars

The toolbar buttons are shortcuts for frequently used commands. Most of the available toolbars are displayed in this illustration of the SolidWorks initial screen, but your SolidWorks window probably will not be arranged this way. You can customize your toolbar display in a way that is convenient for you.

Some toolbars are always displayed; other toolbars display automatically when you open a document of the related type. For example, when you open an assembly document, the assembly toolbar displays.
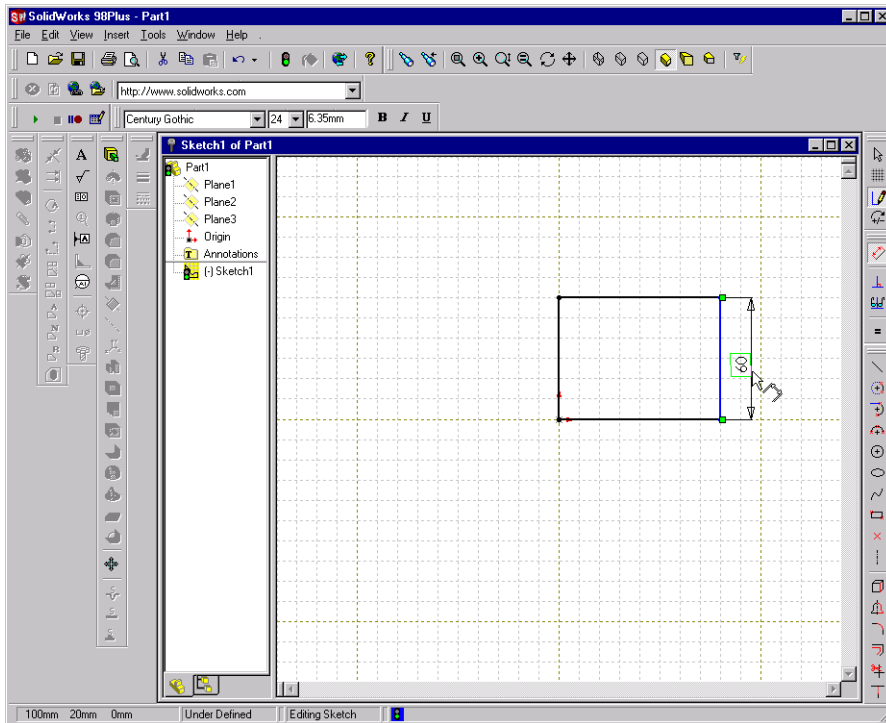
To display or hide individual toolbars:

Click **View, Toolbars**, or right-mouse click the SolidWorks window frame.

A list of all the toolbars is displayed. The toolbars with a check beside them are visible; the toolbars without a check are hidden. Click the toolbar name to toggle its display.

The last item in the toolbar list is **Customize**. Click **Customize** to move toolbar icons to different toolbars, change menus, or reset shortcut keys. See **Customize and Arrange Toolbars** on page 1-18.

Customize and Arrange Toolbars

To customize toolbars:

Click **Tools, Customize**, or right-mouse click the SolidWorks window frame and click **Customize** at the bottom of the toolbar list.

❑ On the **Toolbars** page, you can do the following:

- select individual toolbars to display or hide
- display the toolbar icons in a larger size
- display or hide the tooltips that pop up when the cursor moves over an icon

❑ On the **Commands** page, you can do the following:

- rearrange the command buttons on toolbars
- move buttons from one toolbar to another
- duplicate buttons and place them on more than one toolbar
- delete buttons that you never use by dragging them from the toolbar into the graphics area

❑ On the **Menus** page, you can do the following:

- customize the SolidWorks menus to suit your style of working
- rename, relocate, or remove commands from the menus

❑ On the **Keyboard** page, you can do the following:

- customize the keyboard shortcut keys to suit your style of working
- assign new shortcut keys, remove shortcut keys, or reset the shortcut keys to their original setting

To arrange toolbars:

Click on the space *between* the buttons and drag. If you drag a toolbar to an edge of the SolidWorks window, the toolbar docks to the edge automatically.

If you drag a toolbar away from the window edge, it becomes a floating palette.
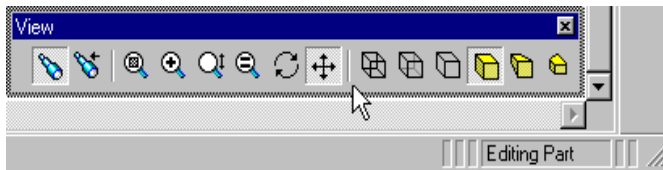
The FeatureManager Design Tree

SolidWorks document windows have two panels:

- A *FeatureManager design tree*, which lists the structure of the part, assembly, or drawing.

- A *Graphics Area*, where you create and manipulate the part, assembly, or drawing.

❑ The FeatureManager design tree makes it easy to:

- Select items in the model by name.

- Do a text search of the FeatureManager design tree for an item. Right-mouse click the name of the part or assembly at the top of the tree and select **Go To** from the menu. Enter the name of the item (or a part of the name) in the edit box and click the **Find Next** button. The name scrolls into view.

- Find an item in the FeatureManager design tree by right-mouse clicking it in the graphics area and selecting **Go To Feature (in Tree)**.

- Identify and change the order in which features are regenerated.

- Display the dimensions of a feature by double-clicking the feature's name.

- Rename features by slowly double-clicking the name and then typing a new name (as standard Windows behavior). You can also right-mouse click a name, select **Properties**, and enter a new name in the **Name** box.

> **NOTE:** Feature names are not case sensitive: "Boss1" and "boss1" are the same. Feature names may not contain the @ character.

- Suppress or hide selected features.

- Temporarily roll the model or assembly back to an earlier state using the *rollback* bar. See **Rollback** on page 5-12.

- Add a new equation, edit, or delete an equation by right-mouse clicking the **Equations** folder Σ, and selecting the action you want. See **Equations** on page 5-6.

- Switch between the features list and the list of named configurations using the tabs at the bottom of the FeatureManager design tree. The tabs also have tool tips.

- Control the display of dimensions and annotations by right-mouse clicking the Annotations folder ⊤ and checking the options. See **Displaying Annotations in Parts** on page 5-26.
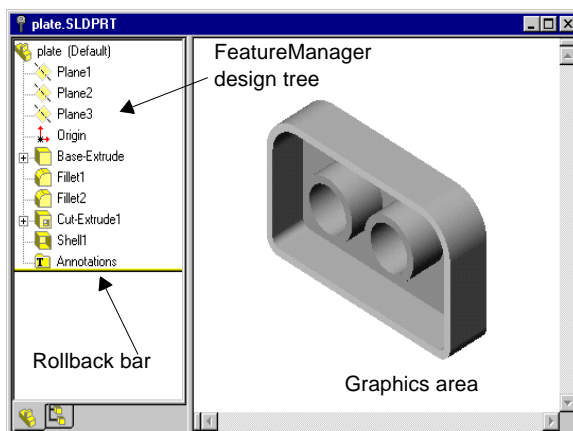
Symbols and Conventions

❑ The FeatureManager design tree gives you information if any part or feature has an *external reference*. (An external reference is a dependency on geometry that exists in another part.)

- If a feature has an external reference, its name is followed by **–>**.

- If a part contains any features with external references, the part name at the top of the design tree list is followed by **–>**.

- If the feature is edited so that it does not contain external references, the **–>** goes away.

- If the feature has an external reference that is currently out of context, the feature name is followed by **->?**.

❑ The FeatureManager design tree uses the following conventions:

- A ⊞ symbol to the left of an item's icon indicates that it contains associated items, such as sketches. Click the ⊞ to expand the item and display its contents.

- Sketches in the FeatureManager design tree are preceded by **(+)** if they are over defined; they are preceded by **(–)** if they are under defined; they are preceded by **(?)** if the sketch could not be solved. For more information about over defined and under defined sketches, see **Fully Defining Sketches** on page 2-35. (There is *no* prefix if the sketch is fully defined.)

- Assembly components in the FeatureManager design tree are preceded by **(+)** if their position is over defined; they are preceded by **(–)** if their position is under defined; they are preceded by **(?)** if their position could not be solved; they are preceded by **(f)** if their position is *fixed* (locked in place).

- Assembly mates are preceded by **(+)** if they are involved in over defining the position of components in the assembly; they are preceded by **(?)** if they could not be solved.

- In an assembly, each instance of the component is followed by a number in angle brackets **<n>** that increments with each occurrence.

FeatureManager Design Tree Options

Select options for the FeatureManager design tree by clicking **Tools, Options** and selecting the **General** tab. (For more information, see **General Options** on page A-15.) Select from the following options:

- **Scroll selected item into view.** Scrolls to display the text that is related to the selected items in the graphics area.

- **Name feature on creation.** When you create a new feature, the feature's name is selected and ready for you to enter a new name.

- **Arrow key navigation.** Lets you use the arrow keys to traverse the FeatureManager design tree, and expand or collapse the design tree and its contents.

- **Dynamic highlight.** The geometry in the graphics area (edges, faces, planes, axes, etc.) highlights when the cursor passes over the item in the FeatureManager design tree.

What's Wrong?

The SolidWorks 98Plus application offers a "What's Wrong" functionality, that allows you to view any regeneration errors of a part or assembly. An exclamation mark (**!**) next to the part or assembly name at the top of the FeatureManager design tree alerts you that there is a problem, and another exclamation mark (**!**) indicates the item responsible for the error.

Right-mouse click the sketch, feature, part, or assembly name and select **What's Wrong** to display the error.

Some common regeneration errors include:

❑ Dangling Dimensions or Relations – dimensions or relations to an entity that no longer exists

❑ Features that cannot be regenerated, such as a fillet that is too large

You can turn off the automatic display of errors by checking the **Display errors at every rebuild** checkbox in the **Rebuild Errors** dialog box. The **Rebuild Errors** dialog box displays when the error is first generated, or you can display the dialog box by right-mouse clicking on the part in the FeatureManager design tree.

> **NOTE:** The **Display errors at every rebuild** dialog box only affects the current session.
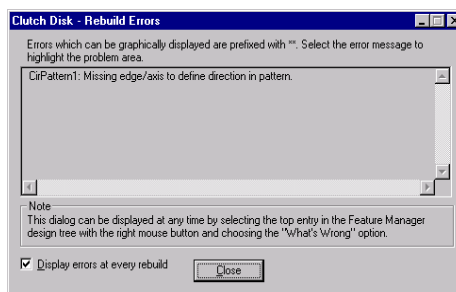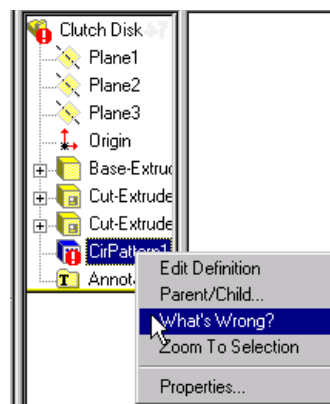
Regenerate Symbol

If you make changes to a sketch or part that require the rebuild of the part, a rebuild icon is displayed next to the part's name as well as in front of the feature and sketch that require the rebuild. Look for the rebuild icon at the right-hand end of the Status Bar, also.

The rebuild symbol also displays when you edit a sketch; when you exit the sketch, the part rebuilds automatically.
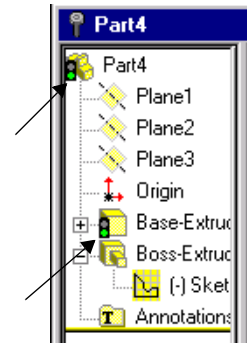
Drag and Drop Features

SolidWorks supports several drag and drop operations for features.

❑ **Reordering features.** You can change the order in which features are rebuilt by dragging them in the FeatureManager design tree. Place the cursor on a feature name, press the left mouse button, and drag the feature name to a new position in the list. (As you drag up or down the tree, each item that you drag over highlights. The feature name that you are moving drops immediately below the currently highlighted item when you release the mouse button.)

If the reorder operation is legal, a ⇤ cursor appears; if it is not legal, a ⊘ cursor appears.

In the following example, the **RoundHole** was cut before the **Shell** feature was added. Later, **Shell** was dragged and dropped before **RoundHole**.
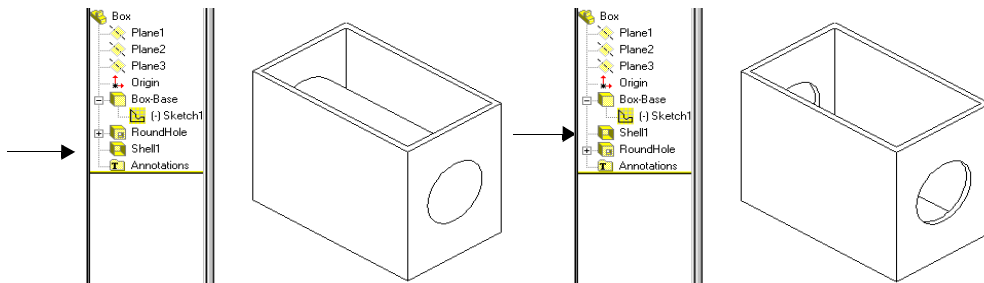
❑ **Moving and copying features.** You can also move features by dragging them in the model. For instance, you can move a hole to a different face.

You can copy or move a fixed-radius fillet or a chamfer using drag and drop.

To create a copy of the feature, hold down the **Ctrl** key while you drag.

For more information about moving and copying features, see **Feature Handles** on page 5-4 and **Copying Features** on page 5-4.

❑ **Dragging and dropping between open documents**. You can drag a part name from the FeatureManager tree of an open part file to insert it in an open assembly document. And, you can drag a part or assembly name from the FeatureManager tree to a drawing document.

Right-Mouse Menus

Whether you are working with a sketch, a model, an assembly, or a drawing, you have access to a wide variety of tools and commands from the right-mouse menu. This gives you an efficient way to do your work without continually moving your cursor to the main pull-down menus or the toolbar buttons.

As you move the cursor over geometry in the model or over items in the FeatureManager design tree, clicking the right-mouse button pops up a menu of commands that are appropriate for whatever you clicked on.

For instance, using the right-mouse button, you can

- Select a tool without moving your cursor to the toolbar

- Open and close sketches

- Change or view the properties of an item

- Give a new name to a feature or dimension using the **Properties** dialog

- **Hide** or **Show** a sketch, plane, axis, or assembly component by right-clicking an item in the FeatureManager design tree

- Open an assembly component for editing

- Access the dimension tools and an annotations menu when in a drawing

- Find an item in the FeatureManager design tree by right-mouse clicking it in the graphics area and selecting **Go To Feature (in Tree)**.

- Change view mode (See **The View Toolbar** on page 1-27.)
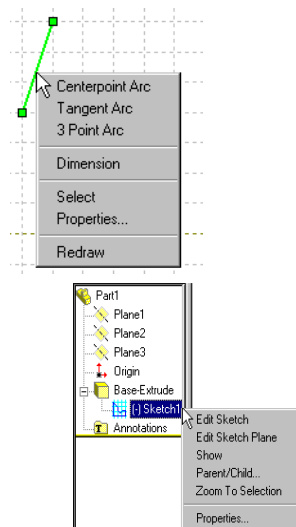
Selection Methods

Selecting from the Graphics Area

Most commands require you to make selections. For instance, to create a fillet, you have to select the model edges or faces you want to fillet.

❑ **Selecting.** Click the ⟨⟩ button (the selection tool), then click the item you want to select.

Notice that items change color as the cursor passes over them. This *dynamic highlighting* helps you locate the item to select.

> **NOTE:** For information about turning dynamic highlighting *off* or *on*, see **Edges Options** on page A-10.

❑ **Selecting with the Selection Filter.** To make it easier to select specific items, you can set the **Selection Filter** to the kind of item that you want to select: faces, edges, and vertices in parts. You can also set it for reference geometry, sketch entities, or dimensions and annotations. With the filter set, the kinds of items that you specified are identified when you pass your cursor over them. Sometimes they are highlighted, and sometimes the cursor changes shape. This makes it easy for you to select only the items that you intend to select.

To toggle the display of the Selection Filter toolbar, click ⟨⟩ on the Standard toolbar, or press **F5**.
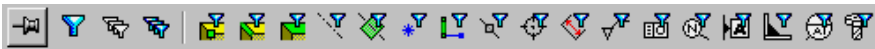
The first four buttons allow you to specify the behavior of the Selection Filter toolbar.

⟨⟩ Keep the **Selection Filter** toolbar open and on top of all other windows.

⟨⟩ Turn the **Selection Filter** on or off.

⟨⟩ Clear all selected filters.

⟨⟩ Select all of the filters that are appropriate for the active function.

The rest of the buttons are filters. Select the filters that match the items you want to select in the graphics area.

Using the Selection Filter:

- Click a function (for example, the **Dimension** tool or the **Fillet** feature), then click ⟨⟩ to select all of the filters appropriate for that function. Click any unwanted filter to deselect it.

- Click a function, then select the filters you want to use. Filters that are grayed-out are inappropriate for the active function and are not selectable.

> **NOTE:** As a reminder to you, while a **Selection Filter** is active, the selection cursor has a small filter icon attached to it. ⟨⟩

❑ **Face and Edge Selection Cursors.** To help you select items, the cursor symbol changes to indicate the kind of item that the cursor is currently pointing to. When you pass the cursor over a face, the cursor symbol is shaped like a flag; when it is over an edge, the symbol is a vertical bar. When you pass the cursor near a vertex, the cursor symbol is a square and the vertex displays two concentric rectangles.

Face
Selection

Edge
Selection

Vertex
Selection
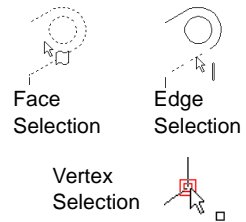
❑ **Selecting midpoint of an entity.** For easy selection of the midpoint of an edge, press the right mouse button over the edge and click **Select Midpoint**. A selection point appears in the middle of the entity.

❑ **Selecting multiple items.** To select more than one item, hold the **Ctrl** key while you click the items. However, if you are selecting multiple items while in a dialog box, it is not necessary to hold the **Ctrl** key.

❑ **Selecting by dragging**. In a sketch or drawing, you can drag a selection rectangle around items to select them. Press the left mouse button and drag the cursor across the area forming a selection rectangle. Entities that are completely within the rectangle are selected and highlighted.
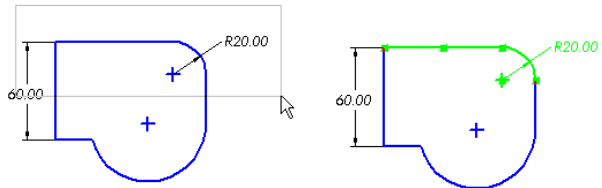
❑ **Selecting loops.** If you select a face, all *loops* (closed collections of connecting edges) on the face are selected, too. To select individual loops, click on the face, then hold down the **Ctrl** key and click on the loops you want to select. (This will deselect the face and leave the loops selected.)

Select
face

Ctrl-click
loop

Only selected
loop is filleted
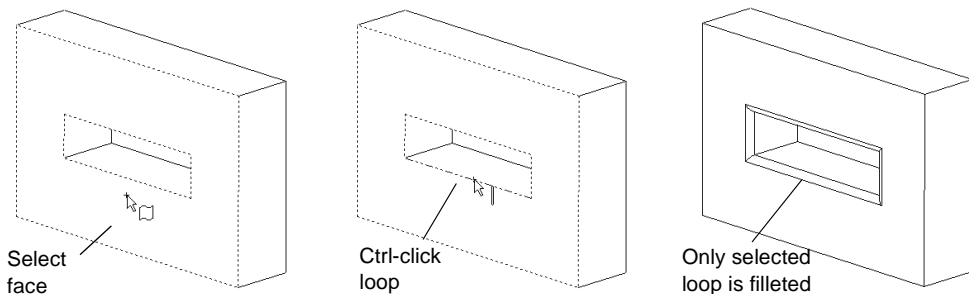
❑ **Changing a selection.** Some dialog boxes show you a list of the selected items. You can change the selection while the dialog is open by clicking a selected item a second time to de-select it. To start over, right-mouse click in the graphics area and choose **Clear Selections**.

Selecting Hidden or Coincident Items

Sometimes the item you want to select is *behind or coincident with* another item.

To select a hidden item:

1 Click the *right* mouse button where you want to make a selection.

2 Click **Select Other** on the popup menu.

Notice the yes/no cursor.

Clicking the *right* button (**N**) highlights each of the items under the point where you clicked, in turn.

3 When the item you want is highlighted, click the *left* button (**Y**) to select that item.

> **NOTE:** You can make selections by clicking items in the graphics area or the FeatureManager design tree, whichever is more convenient.
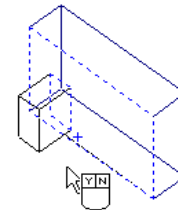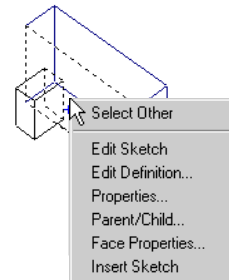
Selecting from the FeatureManager Design Tree

There are several ways to select directly from the FeatureManager design tree.

❑ You can select multiple consecutive items in the FeatureManager design tree by holding the **Shift** key while you select.

❑ You can select multiple items in the graphics area, or select non-consecutive items in the FeatureManager design tree by holding the **Ctrl** key while you select.

❑ You can select features, sketches, planes, and axes in the model by clicking on their names in the FeatureManager design tree.
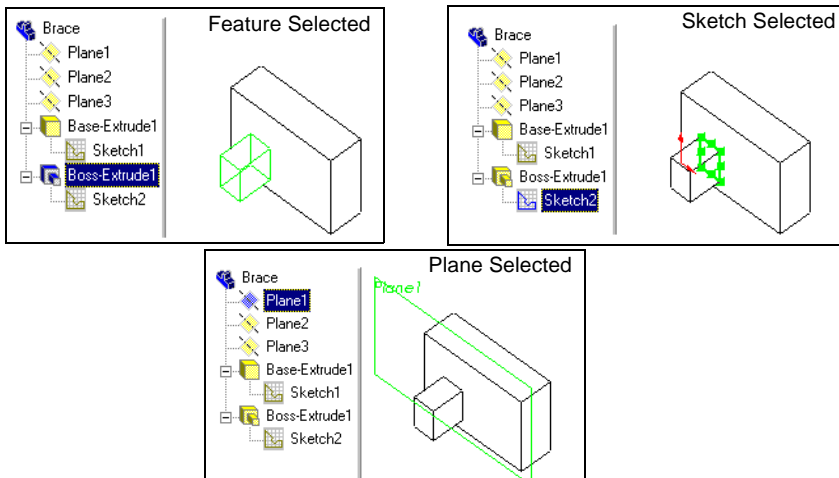
Viewing and Manipulating Parts, Assemblies, and Drawings

The View Toolbar

The View toolbar provides tools for manipulating SolidWorks parts, drawings, and assemblies. (Not all of these tools are appropriate for drawings, however.)

Icon		Description
	View Orientation	Displays the Orientation dialog box. (You can also display the Orientation dialog box by pressing the **Space Bar**.)
	Undo Last Change	Returns to the previous view. You can undo the last 10 view changes.
	Zoom to Fit	Rescales the view so the entire part, drawing, or assembly is visible.
	Zoom to Area	Zooms in on a portion of the view that you select by creating a bounding box. 1. Place the cursor where you want the one corner of the box to be. 2. Drag the cursor diagonally to the opposite corner of the bounding box. 3. Release the cursor. The image enlarges proportionally to the size of the bounding box. 4. To resize the image back to fit the screen, select **Zoom to Fit**.
	Zoom In/ Out	Dynamically changes the scale of the image. • *To enlarge the image*: Press the left mouse button and drag upward. • *To reduce the image*: Press the left mouse button and drag downward.
	Zoom to Selection	Zooms in on a portion of the model, assembly, or drawing that you select. Click an entity to which you want to zoom, or hold the **Ctrl** key and click several entities. To zoom back out, select **Zoom to Fit**.

	Rotate View	Dynamically turns the part or assembly image around a view center as you move the mouse. (Not for drawings.) • *To rotate the image on a vertical axis*: Move the mouse left to right. • *To rotate the image on a horizontal axis*: Move the mouse up and down. • *To rotate the image diagonally*: Move the mouse diagonally. • *To rotate the image about a specific point, line, or plane:* Select the **Rotate** icon, select the entity to rotate about, and use the mouse to rotate, as usual. (Click "air" to deselect the point, line, etc.) You can also use the keyboard arrow keys to rotate the image.
	NOTE: To change the speed of the view rotation, click **Tools, Options**, and select the **General** tab. Use the slider to change the speed of rotation with the mouse; you can change the angle increment used when rotating with the arrow keys.	
	Pan	Dynamically moves the image. Press and hold the left mouse button while moving the cursor around on the screen.
	Wireframe	Displays all the edges of the part or assembly.
	Hidden in Gray	Displays edges that are obscured from view in a light gray line. Visible edges are displayed normally.
	Hidden Lines Removed	Displays only those lines that are visible at the angle the model is rotated. Obscured lines are removed.
	NOTE: To change the default display color of lines in the above three view types, click **Tools, Options, Color**. Select the line type, click **Edit**, and select a color from the color palette. Click **OK**.	
	Shaded	Displays a shaded image of the model. This is the default view mode.
	NOTE: To change the default color of the shaded part, click **Tools, Options, Color**. Select **Shading**, click **Edit**, and select a color from the color palette. Click **OK**. You can also change the color, intensity and direction of the light that strikes the surface of the shaded part. Click **View, Lighting** and choose from the options available for **Ambient**, **Directional**, and **Spot Light.**	
	Perspective	Displays a perspective view of the model. A *perspective* view is the most normal view as seen by the eye or a camera. Parallel lines recede into the distance to a vanishing point. Perspective may be used in combination with any of the view modes.

	Section View	Creates a section view of the model or assembly for display purposes. You cut through the model or assembly using one of the following:
		• one or more standard planes (Plane1, 2, or 3).
		• one or more planes that you create for the purpose (Click **Insert, Reference Geometry, Plane**, and create a new plane.) See **Creating a Construction Plane** on page 3-2.
		• one or more planar face(s) on the model.
		• the viewing plane (The invisible plane that is parallel to the screen you are looking through. The initial position for this plane is the center of the part body bounding box.) See **Section View of a Model** on page 5-27.
	Toggle Selection Filter Toolbar	Shows or hides the Selection Filter toolbar. (You can also show or hide the Selection Filter by pressing the **F5** key.)

NOTE: For information about other SolidWorks toolbars, click **Help** to access online help. In the help **Contents**, select **SolidWorks Basics**, and click **Toolbars**.

The Standard Views Toolbar

The Standard View toolbar provides tools for quickly moving your sketch, model or assembly to any of the standard views. To change the view, click a different view icon.

	Front View		Top View
	Back View		Bottom View
	Left View		Isometric View
	Right View		Normal To – Select a plane or planar face

Using Named Views

You can display a part or assembly using named views with the
Orientation list box.

To display the* Orientation *list box:

- Click 🖸 on the View toolbar.

- Click **View, Orientation**.

- Press the **Space Bar**. The **Orientation** dialog box appears under the
  cursor.

To switch quickly between named views, double-click the name of the
view that you want to see.
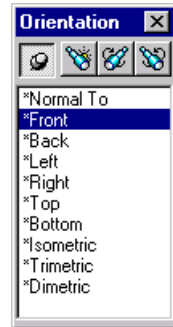
Use the buttons on the **Orientation** dialog box as follows:

The **Push pin** icon keeps the **Orientation** list open and on top of all other
windows. Drag the dialog box to a convenient location on your screen
and click the push pin icon to hold it in place.

If the push pin is holding the **Orientation** list open when you exit the
SolidWorks application, the dialog box is restored to the same location
when you run the application again.

The **Add a new view** icon lets you make additions to the standard list of
view orientations. You can rotate the model or change the zoom, then
click the **Add a new view** icon. Enter a descriptive name for the view,
and click **OK**. The name appears at the top of the list of views, and you
can rotate your model or assembly to the view by double-clicking the
view name.

The **Update Standard Views** icon lets you change the orientation of all
the standard views. Double-click on one of the named views to select the
new orientation, then click the name of the standard view you want to
assign to the current orientation of the model. For example, if you want
what is currently the Left view to become the Front view, double-click
Left, single-click Front, and then click 🖉. This updates all of the
standard views so they are relative to this view.

Click the **Reset** button to return all standard views to their default
settings.

To return your model to a previous view:

Click 🖸 on the View toolbar,

–or –

click **View, Modify, Undo View Change**, or press **Ctrl-Shift-z**.

You can undo the last 10 view changes.

Multiple Views

You can open additional views of the same document. Selecting an item in one view selects it in all views. For instance, when creating a fillet you could select edges on the front of the model in one view and edges on the back in another view.

❑ Click **Window**, **New Window** to open another view in a new window.
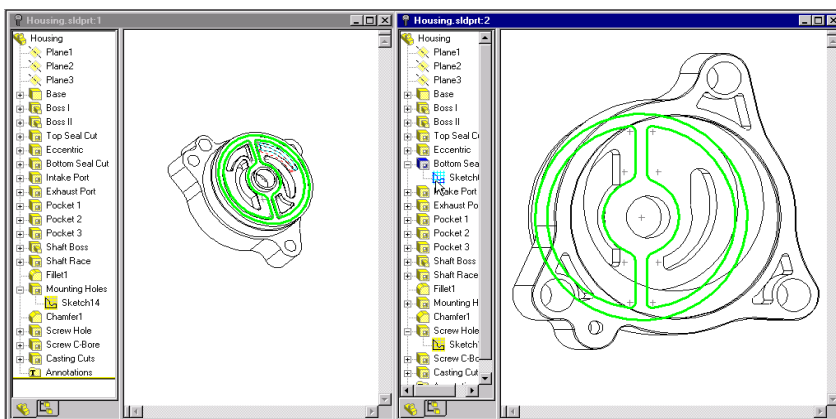
❑ Drag the horizontal and/or vertical **Split** controls to split the window into two or four panes. You can zoom, rotate and set the view mode for each of these views independently. Click in the view, then set the view options.

Horizontal Split control

Window size control

Vertical Split control
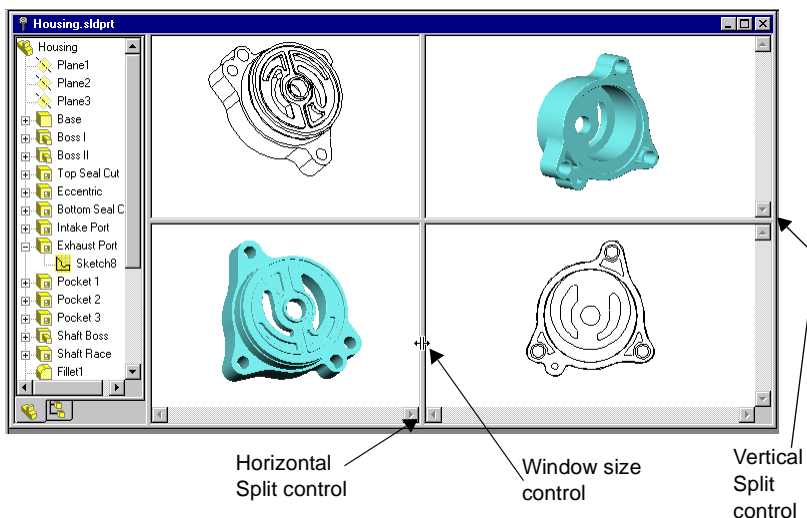
Time Saving Tips

The following tips may help you save time.

❑ **Save snapshots.** Use the **Save As** command to save successive "snapshots" of your design under different names.

❑ **Use tooltips.** To identify a button on a toolbar, just hold the cursor over the button; a *tooltip* will soon pop up.
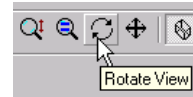
❑ **Use spin box arrows to change values.** If a value box has arrows next to it, you can increase or decrease the value by clicking on the arrows. (You can change the spin box increment value with the **Tools, Options** command. Click the **Grid/Units** tab.)
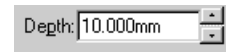
❑ **Modify dimensions with the spin box.** You can increase or decrease dimension values by clicking on the arrows. In addition, you can use the row of buttons below the value window to do any of the following:

    ✓ Accept the current value and exit.

    ✗ Restore the original value and exit.

    🔴 Rebuild the model with the current value.

    ±? Reset the spin box increment value.

❑ **Use the Modify spin box as a calculator.** Enter values and arithmetic symbols directly into the spin box to calculate the dimension.
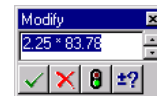
❑ **Check the status bar for prompts.** The *status bar* at the bottom of the SolidWorks window describes commands as you scroll through the menus or point to buttons on toolbars.
It also provides prompts for some commands that require multiple steps.
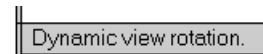
> **NOTE:** You can hide or display the status bar. Click **View, Status Bar**. A checkmark next to the menu item means the status bar is visible.

Keyboard Shortcuts

Keyboard shortcut keys are available for every menu item. Look for the underlined letters in the main menu bar:
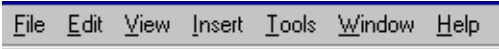
Press the **Alt** key and the underlined letter to display the menu. For example, press **Alt** + **F** to pull down the **File** menu.

Also, look for the underlined letter for each of the menu items. When the menu is pulled down, pressing an underlined letter activates the related command. For example, press **Alt** + **F** to pull down the **File** menu, then press **C** to close your file.

Some commands also have accelerator keys that are displayed on the menu next to the command. For example, the combination **Ctrl** + **N** opens a new file.
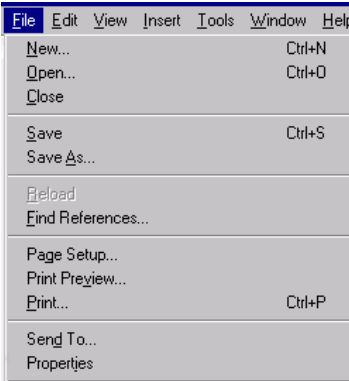
The following table lists the default keyboard shortcuts for rotating and viewing your model.

Action	Key Combination
Rotate the model:	
• horizontally or vertically	**Arrow** keys
• horizontally or vertically 90 degrees	**Shift** + **Arrow** keys
• clockwise/counterclockwise	**Alt** + left/right **Arrow** keys
Scroll the model	**Ctrl** + **Arrow** keys
Zoom in	**Z**
Zoom out	**z**
Rebuild the model	**Ctrl + B**
Force rebuild the model and all its features	**Ctrl + Q**
Redraw the screen	**Ctrl + R**

Additionally, you can customize the keyboard shortcut keys to suit your style of working.

To customize your keyboard:

1 Click **Tools, Customize**.

2 Click the **Keyboard** tab.

3 From the **Categories** box, select the menu that contains the command you want.

4   From the **Commands** box, select the command for which you want to assign or remove a shortcut key.

The **Current Key** box displays the current shortcut key for the selected command.

The **Description** box displays the command description for the selected command.

5   Click **Remove** to remove the shortcut key for the selected command,

– or –

to specify a new shortcut key, enter the new key or key combination in the **Press New Shortcut Key** box, and press **Assign**.

6   If you are not satisfied with the changes you made, press **Reset All** to reset the shortcut keys to their original state.

7   Click **OK** to accept the changes you have made and close the dialog box.

Preparation for Printing

Page Setup

Use **File**, **Page Setup** to set the margins and page orientation that works best with your printer or plotter. This setting applies for all SolidWorks documents that you print, until you change the setup.

You can also use **Page Setup** to create custom headers and footers for individual documents before printing.

To set print options:

1   Click **File, Page Setup**.

2   On the **Printer** tab:

- **Paper Margins.** Set **Use printer's margins** to use the printer's default settings for margins,
  – or –
  Specify margin widths in the appropriate margin boxes (**Top, Bottom, Left,** or **Right**).

- **Line Weights.** Specify the line weight in the appropriate line style boxes (**Thin**, **Normal**, and **Thick**).

- **Page Orientation**. Select either **Portrait** (vertical page orientation) or **Landscape** (horizontal page orientation).

- **Scale.** Select **Scale to Fit** to force the drawing onto the printer's page,
  – or –
  In the **Scale** scroll box, click the arrows to specify the scale to use to print the drawing. (100% = 1:1)
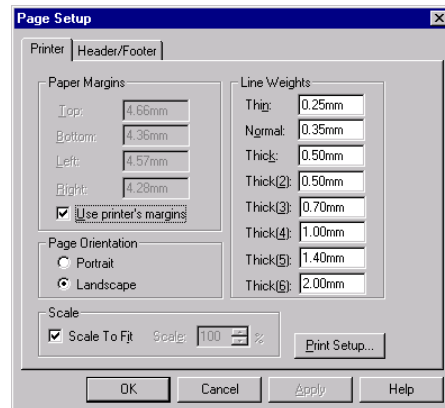
3 Click **Apply** to accept the values displayed. Click **OK** to close the dialog box.

To create a custom header and/or footer for a document:

1 Click **File, Page Setup**.

2 From the **Header/Footer** tab, scroll the **Header** and **Footer** boxes to select predefined headers and/or footers, and view your selection in the **Preview** boxes,

– or –

Click **Custom Header** or **Custom Footer** to define your own header and/or footer. Click **Font** to select a font style and size.
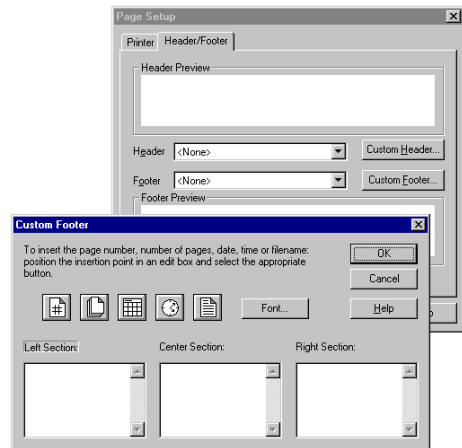
3 Construct the header or footer by clicking a region (**Left Section**, **Center Section**, or **Right Section**) and then clicking the icon that represents the kind of information to put in that region: **Page Number**, **Number of Pages**, **Date**, **Time**, and **Filename**.
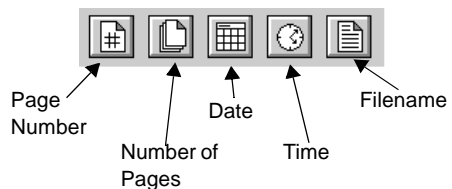
4 Click **OK** and view text in the preview box.

5 Click **Cancel** if you are not satisfied; click **Apply** to use the header or footer that you constructed.

6 Click **OK** to close the dialog box.

7 To view your changes before printing the document, click **File, Print Preview**.

Rendering Models with the PhotoWorks Application

PhotoWorks™ is a photo-realistic rendering application that lets you create realistic images directly from SolidWorks models. PhotoWorks is a separately purchased and licensed application that can be ordered from the dealer from whom you purchased SolidWorks. When it is installed, PhotoWorks is available as an *add-in* to the SolidWorks 98Plus application and is accessible by clicking **Tools, Add-Ins**.

Using the PhotoWorks application, you can specify model surface properties such as color, texture, reflectance, and transparency. It is supplied with a library of surface textures (metals, plastics, and so on) and, in addition, you can scan in and use your own bit-mapped surface textures, materials, scenery, and logos. You can find detailed instructions on how to use the PhotoWorks application in *Learning to Use SolidWorks 98*, Chapter 15.

Identifying Features with FeatureWorks

FeatureWorks™ is an application that recognizes a limited set of features on an imported solid in a SolidWorks part document. Once these features have been recognized, they are added to the FeatureManager design tree to be modified similar to any other SolidWorks features. FeatureWorks is primarily intended for machined parts and sheet metal parts.

FeatureWorks is a separately purchased and licensed application that can be ordered from the dealer from whom you purchased SolidWorks. Fully integrated with SolidWorks, FeatureWorks is supplied as a SolidWorks dynamic link library (.DLL) *add-in* to the SolidWorks 98Plus application. It is accessible by clicking **Tools, Add-Ins**.

You access all the controls for the FeatureWorks interface from the FeatureWorks item on the main SolidWorks menu bar. This menu bar is displayed whenever a SolidWorks part is open. For more information, see **FeatureWorks** on page 13-25.

SolidWorks Application Programming Interface

The SolidWorks Application Programming Interface (API) is an OLE programming interface to SolidWorks. The API contains hundreds of functions that can be called from Visual Basic, VBA (Excel, Access, and so forth), C, C++, or SolidWorks macro files. These functions provide the programmer with direct access to SolidWorks functionality. Refer to *Appendix B* in this *User's Guide* for an overview of the SolidWorks API.

For a detailed description of the API and a list of all the new functionality added to the API since the last release of the SolidWorks software, refer to the API online help file. This help file, **API_HELP.HLP**, is in the ..**\SAMPLES\APPCOMM** subdirectory of your SolidWorks installation. There is also a detailed description of the API functions on the SolidWorks web page (http://www.solidworks.com/html/Products/api).

For a list of the new functions added to the API for SolidWorks 98Plus, refer to the SolidWorks online help.

Sketching with SolidWorks

Most SolidWorks features begin with a sketch. Becoming comfortable with using the sketch tools is, therefore, very important. The discussion of sketching in this chapter is enough to get you started, but you should also do the "40-Minute Running Start" in the *Learning to Use SolidWorks 98Plus* tutorial. Creating the parts in the rest of the tutorial chapters will acquaint you with many sketching techniques, as well. For information about using sketches to create solid model geometry, see Chapter 4, "Creating Features."

This chapter describes:

❑ Starting a new sketch

❑ Accessing and using the sketch tools

❑ Dimensioning the sketch

❑ Adding and deleting geometric relations

❑ Fully defining sketches

❑ Taking advantage of inferencing

❑ Sketching on the face of a part

❑ Editing and exiting a sketch

❑ Deriving a new sketch from another

❑ Selecting a starting plane for a sketch

❑ Deciding how complex the sketch should be

Starting a New Sketch

To open a new sketch, you can either

- Click the **Sketch** tool ![sketch icon], or

- Click **Insert, Sketch** on the main menu

A new sketch opens on Plane1 (the default plane) and activates the three sketch toolbars: *Sketch*, *Sketch Tools*, and *Sketch Relations*. These toolbars and tools are described on the next few pages.
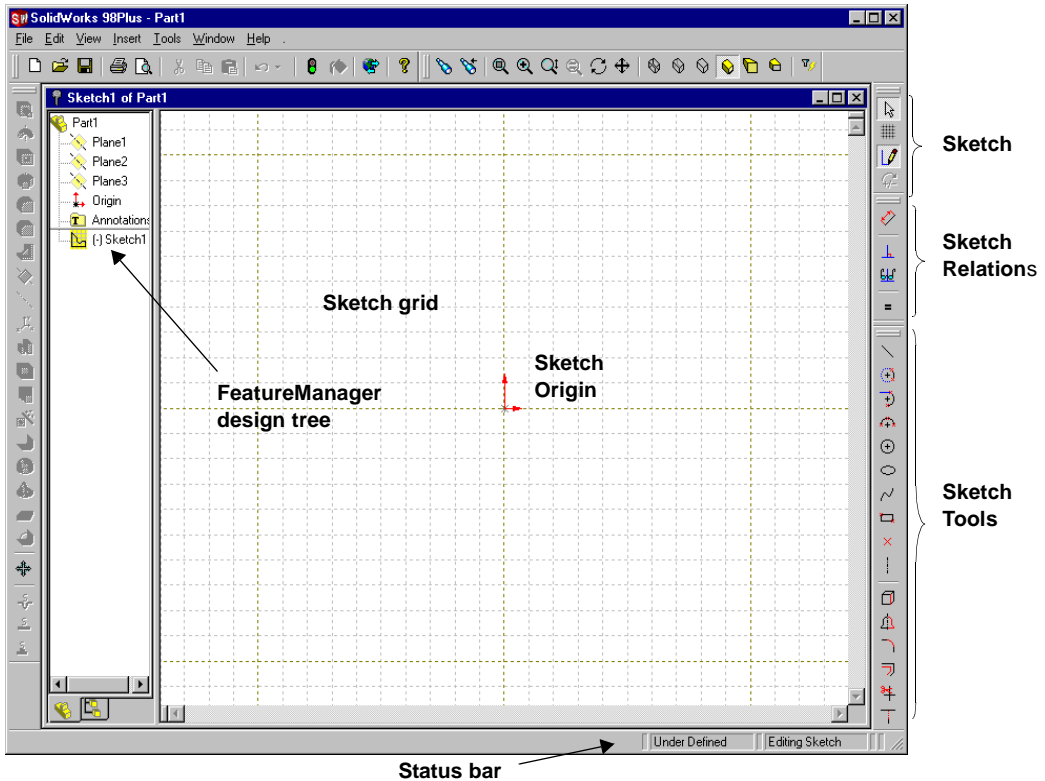
For information about options that you can select to customize the sketching functionality, refer to *Appendix A*, **General Options** on page A-15.

The Sketch Toolbar

The tools on the Sketch toolbar act on the sketch as a whole, rather than on the individual sketch entities.

Select is a most versatile tool that is used throughout the SolidWorks application. To access **Select** from the menu, click **Tools, Select**, or right-mouse click and choose **Select** from the right mouse menu.

In a sketch or drawing, use **Select** to

- select sketch entities
- drag sketch entities and/or endpoints to reshape the sketch
- select a model edge to use with **Convert Entities** or **Offset Entities**
- select multiple entities while pressing the **Ctrl** key
- drag a selection rectangle around multiple sketch entities
- select a dimension to drag to a new location
- double-click a dimension to that you want to modify

Grid gives you access to grid options such as grid visibility, grid spacing, or snap behavior. Use the **Grid** button, or click **Tools, Options**, and select the **Grid/Units** tab. See **The Sketch Grid** on page 2-37.

Sketch opens and closes a sketch. **Sketch** is on the **Insert** menu.

Modify moves, rotates, or scales a sketch. **Modify** is on the **Tools, Sketch Tools** menu. See **Modify Sketch** on page 2-19.

The Sketch Tools Toolbar

The tools on the Sketch Tools toolbar create individual sketch entities or act on the sketch entities in some way.

You can access the sketching tools from:

- the **Sketch Tools** toolbar
- the **Tools** menu
- the right mouse button menu (Only those sketch tools and sketch relations tools that are appropriate at the time of the right-mouse click are available from the right-mouse menu.)

The **Sketch Tools** toolbar and the **Tools**, **Sketch Entity** menu provide the following sketching tools. (Not all menu items have corresponding toolbar icons.)

Line creates a line.

Rectangle creates a rectangle.

Circle creates a circle.

Centerpoint Arc creates an arc from a centerpoint, start point, and end point.

Tangent Arc creates an arc, tangent to a sketch element.

3 Pt Arc creates an arc through three points (start, end and midpoint).

Ellipse creates an ellipse.

Centerpoint Ellipse creates a partial ellipse from a centerpoint, start point, and end point (menu item only).

Parabola creates a parabola.

Spline creates a spline.

Point creates a reference point.

Centerline creates a reference line. Used as construction geometry, and to make symmetrical sketch elements, mirror features, and revolved parts.

Text creates text in a sketch on a model face (menu item for parts only). See page 2-12.

Construction Geometry converts sketched entities (lines, arcs, splines, ellipses, etc.) in a drawing to construction geometry (menu item for drawings only). See **2D Sketching** on page 8-35.

If icons are missing from your toolbars, click **Tools, Customize**.

For descriptions of creating and using sketch entities, see **Using the Sketch Entity Tools** beginning on page 2-7.

To use sketch entities as construction geometry, see **Converting Sketch Lines to Construction Geometry** on page 3-7.

The **Sketch Tools** toolbar and the **Tools, Sketch Tools** menu provide the following tools: (Not all menu items have corresponding toolbar icons.)

Convert Entities creates one or more curves in a sketch by projecting an edge, loop, face, external sketch curve, external sketch contour, set of edges, or set of external sketch curves onto the sketch plane. (Available from an open sketch in an active part or assembly document only.)

Mirror creates copies of sketch elements by mirroring them around a centerline. When you create mirrored elements, a *symmetric* relation is applied between corresponding pairs of sketch segments or sketch points (the ends of mirrored lines, the centers of arcs, and so on). If you change a mirrored element, its mirror image also changes.

Fillet creates a tangent arc at the intersection of two sketch elements, trimming away the corner.

Offset Entities creates sketch curves offset from a selected model edge, loop, face, external sketch curve, external sketch contour, set of edges, silhouette edge, or set of external sketch curves, by a specified distance.

An offset entity relation is created between each original entity and the corresponding sketch curve. If the entity changes, when you rebuild the model the offset curve also changes. (Available from an open sketch in an active part or assembly document only.)

Trim trims or extends a sketch element.

Extend extends a sketch element.

Insert Spline Point places a point on a spline to use as a handle, or to which you can add dimensions or relations.

Automatic Relations automatically creates relations as you add sketch entities.

Automatic Solve automatically does the computation to solve the sketch geometry in the part as you create it. When you are changing many dimensions in an active sketch, you may want to turn off **Automatic Solve** temporarily.

Align Grid aligns the sketch grid with the selected model edge.

Detach Segment on Drag allows you to detach a sketch segment from other entities to which it is attached.

Override Dims on Drag lets you override dimensions by dragging sketch entities.

Close Sketch to Model closes an open profile sketch using existing model edges.

Check Sketch for Feature checks sketch for errors that would prevent its use in creating the specified feature.

For descriptions of using the Sketch Tools, see **Using the Sketch Tools** beginning on page 2-14.

The Sketch Relations Toolbar

The Sketch Relations tools have to do with dimensioning and defining the sketch entities.

The **Sketch Relations** toolbar and the **Tools, Dimensions** menu provide tools for dimensioning.

The **Sketch Relations** toolbar and the **Tools, Relations** menu provides tools for adding and deleting geometric relations. (Not all menu items have corresponding toolbar icons.)

Dimension creates dimensions. The type of dimension (point-to-point, length or angular) is determined by the items you click. See **Dimensioning Sketches** on page 2-22.

Add Relations creates geometric relations (like tangent or perpendicular) between sketch elements, or between sketch elements and planes, axes, edges, curves, or vertices. See **Geometric Relations** on page 2-28 and **Add Relations Table** on page 2-30.

Display/Delete Relations displays relations that were either manually or automatically applied to sketch elements, and lets you delete relations that you no longer want. You can also repair entities that have errors by replacing the listed reference. See **Display/Delete Relations** on page 2-32.

Scan Equal displays elements with equal lengths and/or radii, and lets you create equal length/radii relations between the elements. See **Scan Equal** on page 2-34.

Constrain All sets constraints on the sketch entities of an unconstrained imported .DXF or .DWG drawing. See **Constrain All** on page 2-34.

Using the Sketch Entity Tools

This section discusses how to create and use sketch entities.

Line

To sketch a line:

1 Click the **Line** tool. ◻

2 Press the left mouse button where you want to begin sketching the line and drag the cursor.

   A horizontal or vertical line automatically snaps to the grid points if grid snap is on.

3 To manipulate the line, click the **Select** tool. ◻

To change the length of the line:

1 Point the cursor to one of the end points and press the left mouse button.

2 Drag to lengthen or shorten the line.

To move the line:

1 Point the cursor to the line and press the left mouse button.

2 Drag the line to another position.

To change the angle of a diagonal line:

1 Point the cursor to an end point and press the left mouse button.

2 Drag to lengthen, shorten, or change the angle of the line.
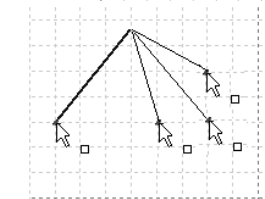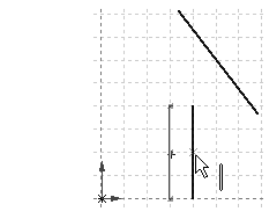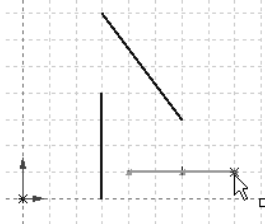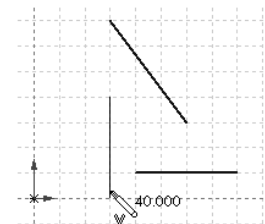
Rectangle

To create a rectangle:

1 Click the **Rectangle** tool. ◻

2 Place the cursor where you want one corner of the rectangle to appear.

3 Drag the cursor and release when the rectangle is the correct size and shape.

   While sketching the rectangle, its dimensions are dynamically displayed.

Circle

To sketch a circle:

1 Click the **Circle** tool.

2 Press the left mouse button where you want to place the center of the circle and drag outward to set the radius.

3 To manipulate the circle, click the **Select** tool.

To move the circle:

Point the cursor at the circle center and drag to another position.

To change the radius:

Point the cursor at the circumference and drag out or in.
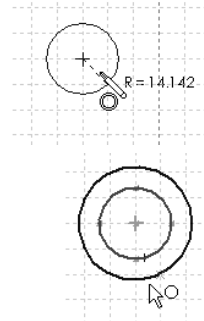
Arcs

To sketch an arc from a centerpoint, start point, and end point:

1 Click the **Centerpoint Arc** tool.

2 Press the mouse button where you want to place the center of the arc and drag to where you want to place the start point of the arc.

3 Release the mouse button. The circumference guideline remains.

4 Press the mouse button and drag to set the length and direction of the arc, then release the mouse button.
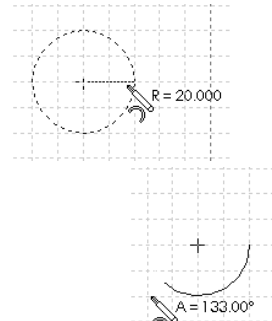
To sketch an arc, tangent to a sketch element:

1 Click the **Tangent Arc** tool.

2 Press the mouse button over the end point of a line, arc, or spline.

3 Drag out the arc to the desired shape.

4 Release the mouse button.
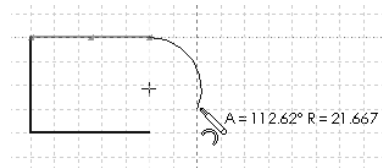
To sketch an arc through three points:

1 Click the **3 Point Arc** tool.

2 Point where you want the arc to start.

3 Press the mouse button and drag to where you want the arc to end.

4 Release the mouse button.

5 Drag the arc to set the radius and to reverse the arc, if necessary.

6 Release the mouse button.
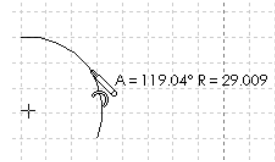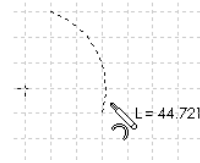
Ellipses

To create an ellipse:

1 Click the **Ellipse** tool.

2 Point where you want to place the center of the ellipse.

3 Press the mouse button and drag to set the major axis of the ellipse.

4 Release the mouse button.

5 Press the mouse button again over the ellipse curve and drag the minor axis of the ellipse.

To create a centerpoint ellipse:

1 Click **Tools, Sketch Entity, Centerpoint Ellipse**.

2 Point to where you want to place the center of the ellipse.

3 Press the mouse button, drag and release to define one axis of the ellipse.

4 Press the mouse button and drag to define the second axis.

5 Release the mouse button. The circumference guideline remains.

6 Press the mouse button and drag to set the length and direction of the ellipse.

Parabola

To create a parabola:

1   Click the **Parabola** tool. $\boxed{\cup}$

2   Place the cursor at the point where you want the focus of the
    parabola, press the left mouse button, and drag the cursor
    away from the focus to enlarge the parabola.

3   Release the cursor and then drag the cursor over the part of
    the parabola's guide lines that you want to use.
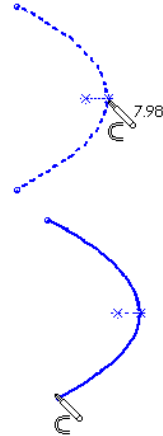
To modify the parabola:

Click the **Select** tool $\boxed{\text{\scriptsize R}}$ and drag the vertex
to shape the curve.

- To flatten the curve, drag the vertex away
  from the focus.

- To make a sharper curve, drag the vertex
  closer to the focus.

Focus   Vertex
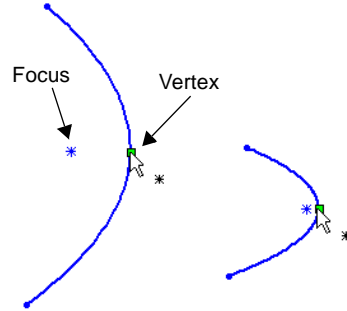
Spline

To create a spline:

1   Click the **Spline** tool. $\boxed{\sim}$

2   Press the mouse button and drag out the first segment.

3   Press over the end point and drag out the second segment.

4   Repeat until the spline is complete.

> **NOTE:** If **Alternate Spline Creation Method** is selected using **Tools,
> Options, General** (see Appendix A), click on each through point to
> create the spline, then double-click when the spline is complete.

To reshape a spline and add relations or dimensions:

1 Right-mouse click the spline and select **Insert Spline Point**.

2 With the spline point cursor, click one or more places on the
spline where you want to insert point(s).

You can:

- use spline points as handles to pull the spline into the shape
  you want

- add dimensions between spline points or between spline points and other entities

- add relations to spline points

- drag the endpoints to reshape the spline

A newly created spline is *non-proportional* by default; you can dimension and add
relations to points of a non-proportional spline. You can change its shape by dragging a
spline point or endpoint.

A *proportional* spline retains its shape when you drag the endpoints; the entire spline
resizes proportionally. You cannot dimension or add relations to its points.

To change the proportional status of a spline, right-mouse click the spline and select
Properties. Click to check or uncheck the **Proportional** checkbox.

To delete spline points:

Select a point and press the **Delete** key.

To simplify a spline:

The **Simplify Spline** option simplifies the definition of the spline by removing through
spline points. The spline is re-fitted to the tolerance of the original spline shape that you
specify in the **Tolerance** edit box.

1 Select **Simplify Spline** from the right mouse menu while the mouse is over a spline, or
click the spline and then click **Tools, Sketch Tools, Simplify Spline**.

2 Enter a higher value in the **Tolerance** edit box, and click **Apply**.

The **Number of spline points** box reports the number of spline points in the original
sketched spline and also reports the number of points in the simplified curve.

3 Observe the spline preview that is displayed. If necessary to produce the spline curve
that you need, reduce or increase the value in the **Tolerance** edit box and click **Apply**
again.

4 When you are satisfied with the shape of the curve, and the reduced number of spline
points, click **OK**.

NOTE: The **Simplify Spline** option is useful for increasing the performance
of a model with complex spline curves.

Point

To create a sketch point (reference point):

1   Click the **Point** tool. $\boxed{\times}$

2   Click where you want to create the sketch point.

To create a sketch point at the virtual intersection point of two entities:

1   **Ctrl**-click the two entities.

2   Click the **Point** tool. $\boxed{\times}$

A virtual sharp symbol indicates the point of intersection and retains its dimensions, even if the actual intersection no longer exists, such as when a corner is removed by a fillet.

There are several options for the appearance of the virtual sharps. To select the appearance of the point, click **Virtual Sharp** on the **Tools, Options, Detailing** page.

Centerline

To create a centerline:

1   Click the **Centerline** tool. $\boxed{\vdots}$

2   Point where you want the line to start.

3   Press the mouse button and drag to where you want the line to end.

4   Release the mouse button.

Text

The **Text** tool creates text on a part as either an extruded boss or a cut.

NOTE: The text aligns with the x-axis of the sketch plane and proceeds in the direction of the increasing x value.

To insert text on a model face:

1   Click a model face where you want to insert text, and open a new sketch.

2   Click **Tools, Sketch Entity, Text.**

The cursor becomes a "Note" cursor.

3   Click on the model face where you want the text to begin.

The **Text** dialog box opens.

4   Type your text into the edit box.

5   Click the checkbox to use the document's default font, or click the **Font** button and choose a font style and size. The text displays in the fonts that you select.

6   Click the arrows next to the **Scale** box to increase or decrease the scale of the text.

7   Click **OK** when you are satisfied with the appearance of the text.

8 You can locate the text sketch on the face of the model by dimensioning or by adding relations to the *text locating point* in the text sketch.

Text locating point

– or –

You can specify the orientation and location of the text by using the **Modify Sketch** tool. (For more information, see **Modify Sketch** on page 2-19.)
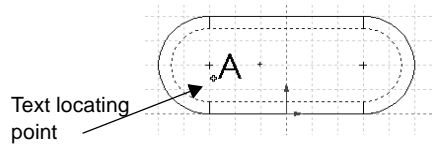
9 Click **Insert, Boss, Extrude** or **Insert, Cut, Extrude**, and set the **Depth** value.

10 Click **OK** to extrude the text.

To change text orientation and location after making the extrusion:

1 Right-mouse click the text sketch in the FeatureManager design tree and select **Edit Sketch**.

2 Edit the dimensions or change the relations that were specified originally,

– or –

Click **Tools, Sketch Tools**, **Modify Sketch** or and use the mouse to position the text.

3 Exit the sketch or, click **Edit, Rebuild** or .

To edit the text:

1 Right-mouse click the text sketch in the FeatureManager design tree and select **Edit Sketch**.

2 Right-mouse click on the text and select **Properties** from the menu.

3 Change the text or the font and click **OK**.

4 Exit the sketch and click **Edit, Rebuild** or .

Using the Sketch Tools

Convert Entities

The **Convert Entities** tool creates one or more curves in a sketch by projecting an edge, loop, face, external sketch curve, external sketch contour, set of edges, or set of external sketch curves onto the sketch plane. See also **Using Silhouette Edges** on page 2-27.

To convert an entity:

1   With a sketch active, click on a model edge, loop, face, external sketch curve, external sketch contour, set of edges, or set of external sketch curves.

2   Click ▣ or **Tools, Sketch Tools, Convert Entities**.

The following relations are created automatically:

- An **On Edge** relation is created between the new sketch curve and the entity, causing the curve to be updated if the entity changes.

- A **Fixed** relation is created internally on the end points of the sketch entity so the sketch can remain fully defined. This internal relation is not displayed when you use **Display/Delete Relations**. Remove the **Fixed** relation by dragging the end points.

Mirror Sketch Elements

The **Mirror** tool creates copies of sketch elements mirrored around a centerline.

When you create mirrored elements, the software applies a *symmetric* relation between corresponding pairs of sketch segments or sketch points (the ends of mirrored lines, the centers of arcs, and so on). If you change a mirrored element, its mirror image also changes.

To mirror existing items:

1   In a sketch, click the **Centerline** tool ▯ and draw a centerline.

2   Hold down the **Ctrl** key and select the centerline and the items you want to mirror.

3   Click ⚠ or **Tools, Sketch Tools, Mirror**.

To mirror items as you sketch them:

1   Select a centerline around which to mirror.

2   Click ⚠ or **Tools, Sketch Tools, Mirror**. Symmetry symbols appear at both ends of the centerline to indicate that automatic mirroring is active.

3   Sketch the geometry. The elements you sketch will be mirrored automatically as you draw them.

4   To turn mirroring off, click ⚠ again.

Fillet (Sketch)

The **Fillet** tool creates a tangent arc at the intersection of two sketch elements, trimming away the corner.

You can also fillet non-intersecting entities, if they are not dimensioned. The selected entities are extended and then the corner is filleted away.

To create a fillet in a sketch:

1 Hold the **Ctrl** key and select two intersecting sketch entities. You can also make your selections after the dialog box appears.

2 Click ⬜ or **Tools, Sketch Tools, Fillet**.

A **Sketch Fillet** dialog box appears.

3 In the **Radius** box, enter a fillet radius value or use the arrows to scroll to a new value.

4 If the corner has dimensions or relations, and you want to keep the virtual intersection point, click the **Keep constrained corners** checkbox.

A virtual sharp symbol indicates the point of intersection and retains its dimensions. To change the display of the point, click **Virtual Sharp** on the **Tools, Options, Detailing** page.

5 If you pre-selected sketch entities, click **Apply** to create the fillet.

– or –

 If you did not pre-select, click the corner that you want to fillet, or select two sketch elements that would form a corner if extended.

NOTE: Consecutive fillets with the same radius are not dimensioned individually; they have an automatic **Equal** relation with the first fillet of that radius.

The **Sketch Fillet** dialog box remains open so you can continue creating fillets until you close the dialog box.

6 Click ⤺ to cancel the last fillet.

7 Click **Close** to close the dialog box.

Offset Entities

The **Offset Entities** tool creates sketch curves offset from a selected edge, loop, face, external sketch curve, external sketch contour, set of edges, or set of external sketch curves by a specified distance.

An offset entity relation is created between each original entity and the corresponding sketch curve. If the entity changes when you rebuild the model, the offset curve also changes. See also **Using Silhouette Edges** on page 2-27.

To create a sketch curve offset from a model edge:

1   In a sketch, select a face or one or more connected edges.

2   Click [icon] or **Tools, Sketch Tools, Offset Entities**.

3   Set the value in the **Offset** box.

4   If necessary, select **Reverse** to change the direction of the offset.

5   Click **OK**.
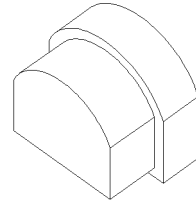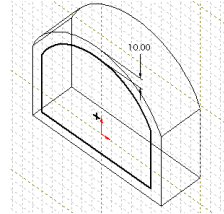
To change the size of the offset:

Change the amount of the offset by double-clicking the offset's dimension and changing the value.

> **NOTE:** If the preview of the offset disappears, the offset is not valid and you must change the offset dimension or direction.

Trim

Use the **Trim** tool to:

- Trim a line, arc, ellipse, circle, spline, or centerline to its intersection with a line, arc, circle, ellipse, spline, or centerline.

- Delete a line, arc, circle, ellipse, spline, or centerline.

- Extend a sketch segment until it is coincident with another entity.

To trim a sketch element:

1   Click the **Trim** tool. [icon]

2   Pass the cursor over a sketch until the sketch segment that you want to trim (delete) is highlighted in red.

3   Click the highlighted segment to delete it.

The segment is deleted up to its intersection with another sketch segment (line, arc, circle, ellipse, spline, or centerline).

The entire sketch segment is deleted if it does not intersect with any another sketch segment.

To use the Trim tool to extend a sketch element:

Drag the endpoint of a sketch element until it is coincident with another entity. You cannot use the **Trim** tool to extend a sketch segment that is already coincident with another entity.

Extend

Use the **Extend** tool to extend a line, arc, or centerline to meet a line, arc, circle, ellipse, spline, or centerline. You can activate the **Extend** tool multiple times to extend a sketch segment until there are no other entities with which it can intersect.

To extend a sketch element:

1 Click the **Extend** tool ⊤, or **Tools**, **Sketch Tools**, **Extend**.

2 Move the cursor over the sketch element that you want to extend (a line, arc, or centerline).

The selected element displays in red, and a green line (or arc) previews the direction in which the element will extend.

3 To accept the preview line or arc, click the sketch element.

– or –

If the preview extends in the wrong direction, move your cursor to a position on the other half of the line or arc and observe the new preview. Click the sketch element to accept the new preview line or arc.

The sketch element extends to intersect with the next sketch element that is available.

Detach Segment On Drag

Detach Segment On Drag lets you detach a sketch segment (line, arc, ellipse, or spline) from the other entities it is attached to in a sketch.

NOTE: If the entity has dimensions or relations, **Detach Segment On Drag** does not delete them, so the entity may not drag or detach as you expect. Delete any relations before you use **Detach Segment On Drag**. See **Display/Delete Relations** on page 2-32.

To detach a sketch segment:

1 If you have a sketch tool active, click it off (click its icon again) before trying to detach a sketch segment.

2 Click **Tools, Sketch Tools, Detach Segment On Drag**.

3 Select the entity that you want to drag away from another sketch entity and drag it to a new location.

The dragged entity detaches from any entities it shares points with.

Override Dims on Drag

The **Override Dims on Drag** tool lets you override dimensions by dragging sketch entities. The sketch dimensions update at the end of the drag. They remain driving dimensions and update in any assemblies and drawings in which they occur.

To override dimensions by dragging:

1   If you have a sketch tool active, click it off (click its icon again) before trying to drag a sketch element.

2   Click **Tools, Sketch Tools, Override Dims on Drag**.

When **Override Dims on Drag** is enabled, you can drag sketch entities and the dimensions will update automatically.

3   Click ⬛ or ✏ to rebuild the part and exit the sketch.

Automatic Solve

A check mark next to the **Automatic Solve** menu item indicates that you want the SolidWorks software to automatically do the computation to solve the sketch geometry of your part as you create it. **Automatic Solve** *on* is the default

To turn automatic computation on or off:

Click **Tools, Sketch Tools, Automatic Solve**. A check mark means that computations are solved automatically.

> **NOTE:** When you are in the process of changing many dimensions in an active sketch, you may want to turn off the **Automatic Solve** capability temporarily.

Align Grid

The **Align Grid** tool aligns the sketch grid with the model edge that you select.

To align the grid to a model edge:

1   **Open** a sketch on a model face.

2   Click **Tools, Sketch Tools, Align Grid**.

3   Click the edge to which you want to align the grid.

4   Click **Apply** to apply the change.

The grid shifts to align with the edge you selected and the origin is placed at the end of the edge closest to where you clicked.

5 Click **OK** to accept the change and close the dialog box.

– or –

Click **Close** to close the dialog box. If you previously clicked **Apply**, the change is saved; if you did not click **Apply**, the dialog closes without change.

Modify Sketch

The **Modify Sketch** tool moves, rotates, or scales a sketch.

The **Translate** and **Rotate** functions move the entire sketch, including its origin. Subsequent changes are based on the updated coordinate system.

To modify the position of a sketch:

1 Open a sketch or select a sketch in the FeatureManager design tree.

2 Click ⏚ or click **Tools, Sketch Tools, Modify**.

3 In the **Translate** region of the **Modify Sketch** dialog, you can enter specific values in the **X value** and **Y value** boxes, and then press **Enter**.

4 Click **Position Selected Point** if you want to select a point on the sketch and move that point to the specified coordinates.

5 In the **Rotate** region of the **Modify Sketch** dialog, you can enter a specific rotation value, and then press **Enter**.

– or –

You can use the mouse cursor ⬚ to move and rotate the sketch, as follows:

- Press the right mouse button to rotate the sketch around the black sketch origin symbol.

- Press the left mouse button to move the sketch.

- Point at the end points or center of the black origin symbol to display one of three *flip* symbols on the right mouse button. Press the flip symbol to flip the sketch on the **X** axis, the **Y** axis, or both.

- Point at the center point of the black origin symbol to display a point symbol on the left mouse button. Press the left button to move the center of rotation independently of the sketch.

NOTE: You cannot move a sketch if it has multiple external references. (The mouse cursor has a **?** on the left button.) If the sketch has only one external point, you can rotate the sketch around this point.

To scale a sketch:

Enter a decimal value in the **Scale** box, and press **Enter**.

Scale applies a uniform scale about the origin of the sketch.

NOTE: You cannot scale a sketch if it has external references.

Close Sketch to Model

Closes a sketch with an open profile, using existing model edges.

To use an open profile sketch to extrude using existing model edges:

1   Open a sketch on a model face.

2   Sketch an open profile with endpoints that are coincident with the model edges on the same face boundary.

3   Click **Tools, Sketch Tools, Close Sketch to Model**.

The **Close Sketch with Model Edges** dialog appears.

Notice that an arrow points in the direction in which the profile will close.

4   Check **Reverse direction to close sketch** if it is necessary to close the sketch in the other direction.

5   Click **Yes** when the arrow points in the correct direction; click **No** to exit the dialog without specifying a direction.

Then you can use the closed sketch to create another feature such as an extruded boss or an extruded cut. The following examples show the effect of extruding a boss with sketches closed first in one direction and then the other.
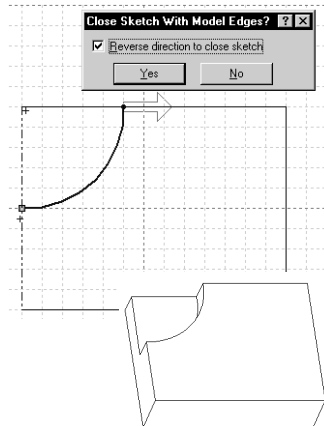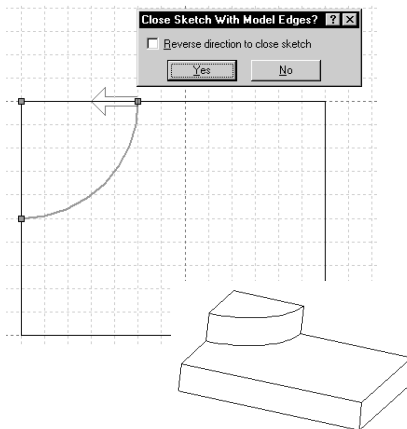
Check Sketch for Feature

Lets you check a sketch for errors that would prevent it from being used successfully in creating the type of feature you specify.

To check a sketch:

1 With a sketch open, click **Tools, Sketch Tools, Check Sketch For Feature**.

   The **Check Sketch For Feature Usage** dialog appears. For each **Feature Usage** type, its required **Contour Type** is displayed. If you select a new **Feature Usage** from the pull-down list, the corresponding **Contour Type** changes dynamically. Each **Contour Type** forces a different set of checks on the sketch in addition to the general checks that are common to all contour types. (See the table of **Legal Contour Types** that follows.)

   • If the sketch has been successfully used to create a feature, the feature type is displayed in the **Feature Usage** box.

   • If the sketch has not been used to create a feature, **<none>** is displayed in the **Feature Usage** box and the contour is checked for errors that are common to all contour types.

2 Click the **Check** button. If the sketch passes the checks, a "**No problems found**" message is displayed. If there is an error, a message describing the error is displayed and, if appropriate, the area of the sketch containing the error is highlighted. It reports one error for each check.

3 To check the sketch for possible use in other feature types, select a new **Feature Usage** type from the pull-down list and click **Check**.

4 Click **Reset** to return to the original **Feature Usage** type.

5 Click the **Close** button to close the dialog. The dialog also closes when you exit the sketch, rebuild, or change to a new document.

Legal Contour types	Features that use this contour type
General	<none> (The contour is checked for errors that are common to all contour types.)
Multiple Contained Closed (If there is more than one contour, one contour must contain the rest.)	Base Extrude
Multiple Disjoint Closed	Boss Extrude, Cut Extrude
Single Open or Closed	Loft, Loft Section, Surface Loft Section, Loft Guide, Sweep Path or Guide, Surface Sweep Section, Cut Extrude Thin, Surface Revolve, Boss Revolve Thin, Base Revolve Thin, Surface Extrude, Boss Extrude Thin, Base Extrude Thin
Single Open	Rib, Cut Extrude
Single Closed	Base Revolve, Sweep Section, Cut Revolve Thin, Cut Revolve, Boss Revolve

Dimensioning Sketches

Usually, you dimension your sketch as you go along. However, it is not *necessary* to dimension sketches before you use them to create features. When you add or change dimensions in a sketch, the changes are reflected in the part and the drawing views of the part.

There are two methods of dimensioning a sketch:

❑ You can select the dimension tool for the specific type of dimension you want.

- Click ✐ , and right-mouse click to select either **Horizontal**, or **Vertical** (the default dimension is **Parallel**)

  – or –

- Click **Tools**, **Dimensions** and select either **Parallel**, **Horizontal**, or **Vertical**.

❑ You can click ✐ and preview the dimension before placing it.

For linear dimensions, the preview automatically switches between horizontal, vertical and slanted, depending on the cursor location. For diameter dimensions, the preview switches between a radial dimension and a linear dimension. The behavior is similar for angular dimensions, where the dimension created depends on the quadrant location of the cursor.

To add a specific type dimension to a sketch:

1 Click ✐ and press the right mouse button to select either **Horizontal** or **Vertical**, or click **Tools**, **Dimensions** and select either **Parallel**, **Horizontal**, or **Vertical**.

2 Click the sketch entity, or entities, then click where you want to place the dimension.
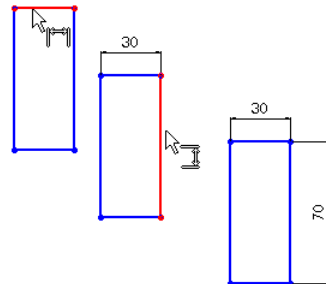
To preview a dimension:

1 Click ✐ and click the sketch entity, or entities, to dimension.

2 Move the cursor around and observe the dimension preview.

3 Click when the preview displays the dimension that you want.
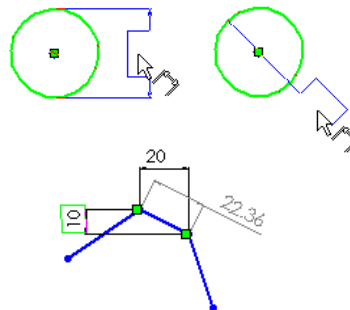
Examples of point-to-point dimensions:
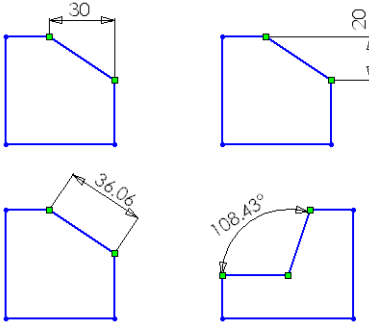
Examples of angular dimensions
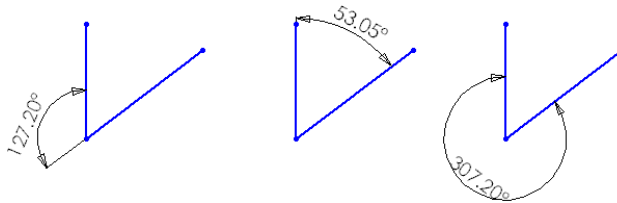
Dimensioning with Equations

You can insert dimensions in a sketch using mathematical relations between dimensions or parameters, using dimension names as variables in the equation. Equations can also be used outside of a sketch.

For a more complete discussion about using and editing equations to control dimensions, see **Equations** on page 5-6.

> **NOTE:** Dimensions that are driven by equations cannot be changed by editing the dimension value in the model.

Changing Dimensions

To change a dimension:

1 Double-click on the dimension.

The **Modify** dialog box appears.

2 To change the dimension value, click the arrows to change the value and press **Enter**, or enter a new value or a calculation and press **Enter**.
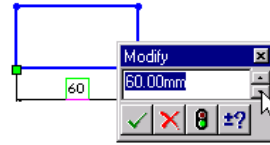
In addition, you can use the row of buttons below the value window to do any of the following:

✓   Accept the current value and exit.

✗   Restore the original value and exit.

🎱   Regenerate the model with the current value. (Not available in a sketch.)

±?   Reset the spin box increment value.
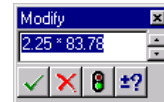
Dimensioning Arcs and Circles

By default, when you dimension to an arc or circle, the dimension goes to the centerpoint of the arc or circle. You can edit the dimension so that it measures to the edge of an arc or circle.

To change dimension options for arcs and circles:

1 Right-mouse click the dimension.

2 Select **Properties** from the menu.

3 From the **Dimension Properties** dialog, select the **First Arc Condition** and the **Second Arc Condition** that provides the dimension that you want.
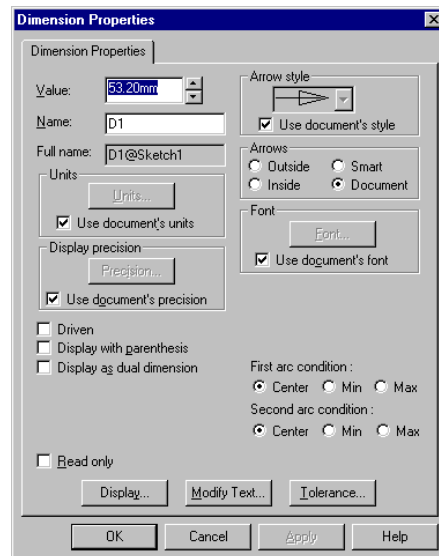
The following illustrations show some of the options for dimensioning between arcs and circles.
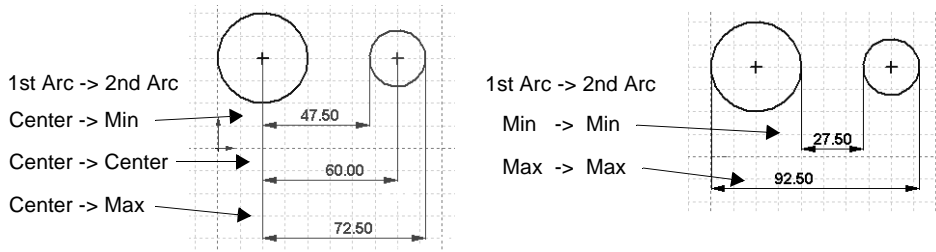
To dimension the true length of an arc:

1 Click [icon], or **Tools, Dimensions**.

2 Click the arc.

3 Click both endpoints of the arc.

4 Click to place the dimension.

An arc length symbol is added above the dimension text.
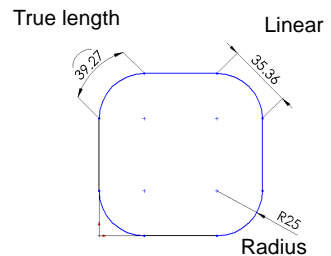
For dimensions on circular features, you have the following options:

- To change a radius dimension to a diameter dimension, right-click the dimension and select **Display as Diameter**.

- To display a diameter dimension as a linear dimension, right-click the dimension and select **Display as Linear**.

- If the linear dimension is not placed at the angle you want, select the dimension, and drag the green handle on the dimension text to set the desired position. The dimension snaps in 15° increments around the circle.
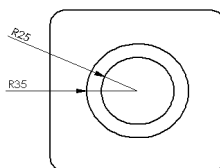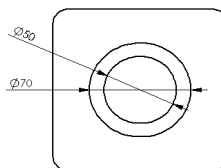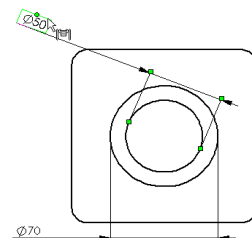
Radius   Diameter   Linear

Dimension Properties

You set the default properties for dimensions on the **Detailing** tab of the **Options** dialog box. See *Appendix A*, **Detailing Options** on page A-4 or **Dimension Properties** on page 9-11 of the *Detailing* chapter for more information.

You can modify properties of selected dimensions at any time.

To modify the properties of a dimension:

1   Right-click the dimension, and select **Properties**.

The **Dimension Properties** dialog box is dynamic and the available options depend on the type of dimension that you clicked (linear, radius, reference, driving, etc.).

2   Change values as described below.

3   Click **OK**.

The following properties are common to all types of dimensions.

- **Value.** The numeric value of a dimension.

- **Name.** The name given to a dimension and used in relations associated with the dimension. To change the name, click in the edit box and enter a new name.

- **Full Name.** The full name of the dimension which consists of the **Name** given above and the name of the sketch or feature in which it is found (for example, *D2@Sketch1*). Used in equations and design tables. This is provided as information only, and is updated when you edit the **Name**.

- **Arrow Style.** Either **Use document's style**, as defined on the **Detailing** tab of the **Options** dialog box, or deselect this checkbox, then choose a different style from the list.

- **Arrows.** Defines placement of arrows with respect to witness lines.

- **Units.** Either **Use document's units**, as defined on the **Grid/Units** tab of the **Options** dialog box, or deselect this checkbox, then click **Units**, and select a new type of length or angular unit.

- **Font**. Either **Use document's font**, as defined on the **Detailing** tab of the **Options** dialog box, or deselect this checkbox, then click **Font**, and select a new style or size.

- **Display Precision**. Either **Use document's precision**, as defined on the **Detailing** tab of the **Options** dialog, or deselect this checkbox, click **Precision**, and specify the precision of the primary and alternate units of measure.

- **Display.** Use the checkboxes to specify which **Witness Lines** and **Leaders** to display.

- **Modify Text**. Lets you add prefix and suffix text and symbols. See page 9-13.

- **Tolerance**. Lets you specify and display dimension tolerances. See page 9-14.

Using Silhouette Edges

You can select the silhouette edge of a rounded surface, such as a cylinder, revolve, loft, sweep, etc., to do any of the following operations: **Convert Entities**, **Offset Entities**, and add **Dimensions** and/or **Relations**.

NOTE: To use silhouette edges, you must first select the **Enable silhouettes** option. Click **Tools, Options**, and select the **General** tab. In the **Sketch** section make sure that **Enable silhouettes** is checked.

To use a silhouette edge:

1 Working with a sketch open on a model face or plane (or working in an active drawing view), move your cursor over a cylindrical face until the cursor changes to include a small cylinder next to the pointing arrow. The cylinder's silhouette edge becomes visible as a dashed line.

NOTE: Silhouette edge visibility is an option that you set by clicking **Tools, Options, Edges**, and selecting **Dynamic Highlight from Graphics View**.

---

2 Select the dashed line to use with the **Convert Entities** or **Offset Entities** sketch tools, or to add **Dimensions** and/or **Relations** to your sketch.

NOTE: To address cases where the generation of silhouettes is too slow, a timer is used in the **Enable silhouettes** option. After 15 seconds, a message appears advising you to use the **Create Silhouettes** right mouse function described below. (You do not have to turn off the **Enable silhouettes** option in **Tools, Options, General**.)

To use silhouettes in a complicated part or large assembly:

1 To improve the rebuild performance of a part or assembly, click **Tools, Options**, and select the **General** tab. Make sure that **Enable silhouettes** is <u>not</u> checked.

2 Working with a sketch open on a model face or plane (or working in an active drawing view), right-mouse click and select **Create Silhouettes** from the right mouse menu. The system tries to create silhouettes for any selected faces.

3 Move your cursor over a cylindrical face until the cursor changes to include a small cylinder next to the pointing arrow.

4 Click on the cylinder's silhouette when it becomes visible as a dashed line.

Geometric Relations

Add Relations ⌞ creates geometric relations (like tangent or perpendicular) between sketch elements, or between sketch elements and planes, axes, edges, or vertices.

When creating relations, at least one of the items must be a sketch entity. The other item(s) can be either a sketch entity or an edge, face, vertex, origin, plane, axis, or sketch curve from another sketch that forms a line or arc when projected on the sketch plane. See also **Using Silhouette Edges** on page 2-27.

Geometric relations can be created two ways:

❑ Automatically created by the SolidWorks software

❑ Manually created by the user

Automatic Relations

Turn **Automatic Relations** on or off for the current document by clicking **Tools, Sketch Tools, Automatic Relations**. A check mark next to the **Automatic Relations** menu item indicates that relations are created automatically as you sketch. For instance, if you draw a vertical line, the SolidWorks software applies a "vertical" relation to it.

To specify **Automatic Relations** for all documents, click **Tools, Options, General**. In the **Sketch** section, select **Automatic relations**.

As you sketch, the cursor changes shape to indicate which relations can be created. Note that the cursor has a small **H** when the line is horizontal; the cursor has a **V** when the line is vertical. If the **Automatic Relations** option is on, the following relations are created automatically.

| Horizontal | Vertical | Coincident | Midpoint |

| Intersection | Tangent | Perpendicular |
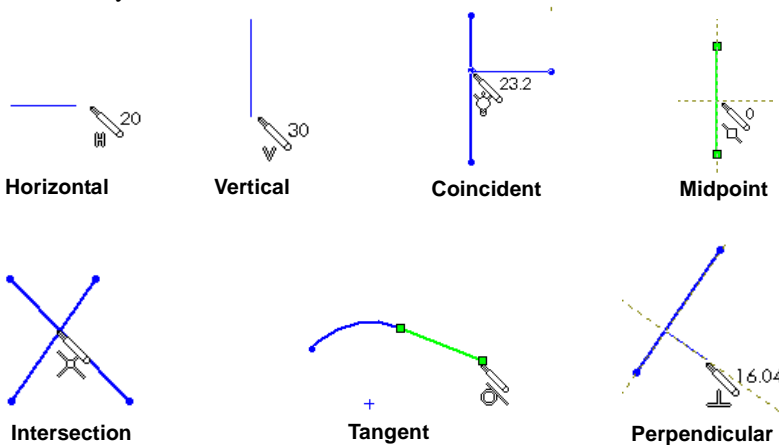
Manually Created Relations

To apply relations manually:

1 Click ⊥, or click **Tools, Relations, Add**.

2 In a sketch, select the item(s) (sketch elements, vertices, edges, and so forth) to relate.

The **Add Relations** dialog selects the best default relation based on the entities you selected, and indicates that selection with a dot in the middle of the radio button. Relations that already exist are marked with a dark circle around the button.

Relations that are not appropriate for the items you selected are "grayed out."

For many relations, you can select more than two entities. At least one of the items must be a sketch element. The other item(s) can be either a sketch element or an edge, face, vertex, origin, plane, axis, or sketch curve from another sketch that forms a line or arc when projected on the sketch plane.

The items that you select are displayed in the **Selected Entities** list box. To remove an item from the list, click it again. To remove all items, right-mouse click in the graphics area and select **Clear Selections**.

3 Select the relation you want to add, then click **Apply**.

To remove the last relation you added, click the **Undo** button, ↶.

4 To turn the **Add Relations** function off, click **Close**.

For a complete discussion of all the sketch relations, refer to the **Add Relations Table** on page 2-30.
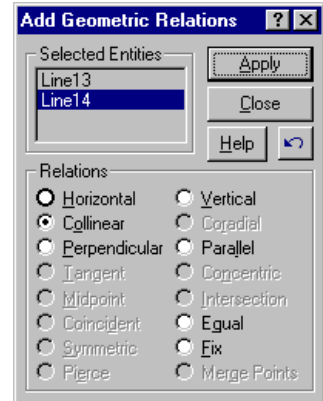
Add Relations Table

Available relations depend on the items you select. Please see the definitions of terms at the end of this table.*

To add this relation:	Select:	Result:
Horizontal or Vertical	One or more lines or two or more points	The lines become horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically.
Collinear	Two or more lines	The items lie on the same infinite line.
Coradial	Two or more arcs	The items share the same centerpoint and radius.
Perpendicular	Two lines	The two items are perpendicular to each other.
Parallel	Two or more lines	The items remain parallel.
Tangent	An arc, spline, or ellipse and another arc, spline, ellipse, or line	The two items remain tangent.
Concentric	Two or more arcs, or a point and an arc	The circles and/or arcs share the same centerpoint.
Midpoint	A point and a line	The point remains at the midpoint of the line.
Intersection	Two lines and one point	The point remains at the intersection of the lines.
Coincident	A point and a line, arc, or ellipse	The point lies on the line, arc, or ellipse.
Equal	Two or more lines, or two or more arcs	The line lengths or radii remain equal.
Symmetric	A centerline and two points, lines, arcs, or ellipses	The items remain equidistant from the centerline, on a line perpendicular to it.
Fix	Any item	The item's size and location are fixed. The end points are still free to move along the underlying infinite line. The end points of an arc or elliptical segment are free to move along the underlying circle or ellipse.

Table continued on next page

Pierce	A sketch point and an axis, edge, line, or spline	The sketch point is constructed coincident to where the axis, edge, or line pierces the sketch plane.
Merge Points	Two sketch points or end points	The two points are merged into a single point.

When you create a relation to a line, the relation is to the infinite line, not just the sketched line segment or the physical edge. As a result, some items may not physically touch when you expect them to.

Similarly, when you create a relation to an arc segment or elliptical segment the relation is to the *full* circle or ellipse.

Also, if you create a relation to an item that does not lie on the sketch plane, the resulting relation applies to the *projection* of that item as it appears on the sketch plane.

Definitions of terms used in the Add Relations table:

Arc	Can be an arc or circle in the sketch, or an external entity that projects as an arc or circle in the sketch.
Ellipse	Can be an ellipse in the sketch, or an external entity that projects as an ellipse in the sketch.
Line	Can be a line or centerline in the sketch, or an external entity (edge, plane, axis, or sketch curve in an external sketch) that projects as a line in the sketch.
Point	Can be a point in the sketch, or an external entity (origin, vertex, axis, or point in an external sketch) that projects as a point in the sketch.
Spline	Can be a spline in the sketch, or an external entity that projects as a spline in the sketch.

Display/Delete Relations

You can view the relations on each entity by selecting the entity and clicking 🔠 or **Tools, Relations, Display/Delete**. If your sketch becomes "over defined" or "not solved," use **Display/Delete** to examine and delete relations as necessary. (For more information about the sketch states, see **Fully Defining Sketches** on page 2-35.)

To view or delete relations:

1   Click the sketch entity about which you have a question. (If no entity is selected, all relations in the sketch are listed.)

2   Click **Display/Delete Relations.**

A dialog box appears that has two tabs:

- **Relations** tells what relations apply and allows you to view or delete the relations. (See **Using the Relations tab** on page 2-32.)

- **Entities** displays information about the entities involved in the active relation and lets you replace them, if necessary. (See **Using the Entities tab** on page 2-33.)

Using the **Relations** *tab*

1   To specify the scope of the relations display, in the **Display Relations By** box select either

- **Entity**, if you want to display relations for the selected entity only, or

- **Criteria**, if you want to display all the relations in the entire sketch that meet the criteria you select from the pull-down list. For example, you can choose **Dangling** to list all the dangling relations in the sketch.

2   Click **Previous** ⬅, or **Next** ➡, to view each relation in turn (if there are more than one).

As each relation is displayed, the related sketch entities are highlighted, and the status is displayed. (The status can be **Dangling**, **Satisfied**, **Over Defining**, or **Not Solved**.)

3   Click the **Suppressed** check box to suppress or unsuppress the current relation.

4   Click **Delete** to delete the current relation; click **Delete All** to delete all of the relations currently selected.

5   To undo an action that you just performed, click the **Undo** button 🔙.

6   You can continue to display relations for any or all entities in the sketch. The dialog box remains open until you click the **Close** button.
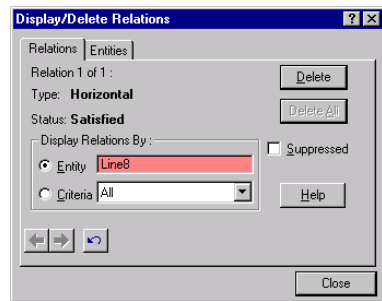
Using the Entities *tab*

1 Click the **Entities** tab.

- The list box displays the name of each entity, the entity's status, and the place where it is defined: **Current Sketch**, **Same Model**, or **External Model**.

- If the entity is defined as existing in the **Same Model**, the **External Information** section lists the external **Entity** to which the selected sketch entity has a relation.

- If the entity is defined as existing in an **External Model**, the **External Information** section lists the **Assembly** name and the **Owner** part within the assembly.

2 To replace an entity, in the list box select an entity that you want to replace.

3 Click an entity in the graphics area that you want to use as a replacement.

This entity's name appears in the selection box under the **Replace** button.

4 Click the **Replace** button.

The software attempts to replace the entity and solve the sketch.

5 If it is necessary to undo the replacement, click the **Undo** button, .

The dialog box remains open until you click the **Close** button.
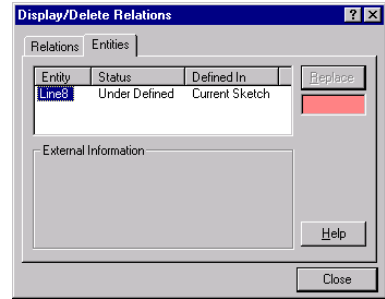
Reattaching Dangling Relations

When there is a single dangling relation, a red handle is displayed at the midpoint of the dangling segment. To reattach the relations, drag the handle onto a sketch or model entity that is suitable to resolve the dangling relation. The cursor displays a ⊘ symbol if the entity is unsuitable.

You must use the **Display/Delete** relations dialog to reattach relations in the following cases:

- if there are multiple dangling relations, or
- if the entity is a sketch point.

Scan Equal

Click **Tools, Relations, Scan Equal** to scan a sketch for elements with equal lengths and/
or radii. **Scan Equal** also provides a way to set an **Equal** relation between sketch elements
that are the same length and/or radius.

To locate equal radii and/or line lengths:

1   In a sketch, click **Tools, Relations, Scan Equal**.

    If sketch elements exist that are equal, the appropriate radio buttons are active. (For
    example, if two or more lines are equal, **Length** is active; if two or more arcs have
    equal radii, **Radii** is active; if an arc has a radius equal to a line length, **Both** is active.)

2   To see the equal sketch elements, click an active radio button.

    • The equal elements corresponding to the radio button that you clicked are
      highlighted in the sketch.

    • The **Value** box displays the length or radii.

    • The **Line Count** or the **Arc Count** box displays the number of equal elements.

3   If there are additional sets of equal elements of the same type, the **Find Next** button is
    active. To highlight the additional sets, click the **Find Next** button.

4   Click **Set Equal** if you want to create an **Equal** relation between the highlighted
    elements.

5   Click **Close**.

Constrain All

The **Constrain All** tool sets constraints on the sketch entities of an unconstrained imported
.DXF or .DWG drawing.

To solve relations in an imported drawing:

1   Import a sketch from a .DXF or .DWG drawing.

2   Click **Tools, Relations, Constrain All**.

The command tries to solve all the apparent relations in the sketch and reports the number
of constraints that were added to the sketch.

Fully Defining Sketches

While you are creating or editing a sketch, you should be aware that a sketch may be in any of five states described below. The state of the sketch as a whole is displayed in the status bar at the bottom of the SolidWorks window.

❑ *Fully Defined* – displayed as black lines. All the lines and curves in the sketch, and their positions, are described by dimensions and/or relations.

❑ *Over Defined* – displayed as red lines. The sketch has too many dimensions and/or relations. For information about viewing and removing conflicting relations, refer to **Display/Delete Relations** on page 2-32.

❑ *Under Defined* – displayed as blue lines. Some of the dimensions and/or relations in the sketch are not defined and are free to change. You can point the mouse at endpoints, lines, or curves and drag them until the sketch entity changes shape.

❑ *No Solution Found* – displayed as pink lines. The sketch was not solved. The geometry, relations, and dimensions that prevent the solution of the sketch are displayed.

❑ *Invalid Solution Found* – displayed as yellow lines. The sketch was solved but will result in invalid geometry, such as a zero length line, zero radius arc, or self-intersecting spline.

With SolidWorks 98Plus it is *not necessary* to fully dimension or define your sketches before you use them to create features. However, it is usually a good idea to fully define your sketches.

> **NOTE:** If you wish to always use fully defined sketches, click **Tools**, **Options**, and select the **General** tab. In the **Sketch** section, click **Use fully defined sketches**.

Sketch Geometry may have any of the following states:

Fully defined – black. The dimensions and relations are fully and correctly described.

Under defined – blue. The dimensions and relations are not defined adequately and lines may move or change size unexpectedly.

Over defined – red. This geometry is constrained by too many dimensions and/or relations.

Dangling – brown, dashed. Only applies to a sketch entity that was added automatically to the sketch in the last known position of the dangling model geometry.

Not solved – pink. This geometry's position cannot be determined using the current constraints.

Invalid – yellow. This geometry would be geometrically invalid if the sketch were solved.

Sketch Dimensions or Relations may have any of the following states:

Dangling – brown, dashed. A dimension that can no longer be resolved (a dimension to a deleted entity, for example).

Satisfied – black. A dimension that is completely and appropriately defined.

Over Defining – red. The dimension or relation is involved in over defining one or more entities.

Not Solved – pink. The dimension or relation results in not being able to determine the position of one or more sketch entities.

Driven – gray. The dimension's value is driven by solving the sketch.

Working in a Sketch

When a sketch is active, the sketch window displays the sketching toolbars, the sketch grid, as well as helpful information in the status bar.
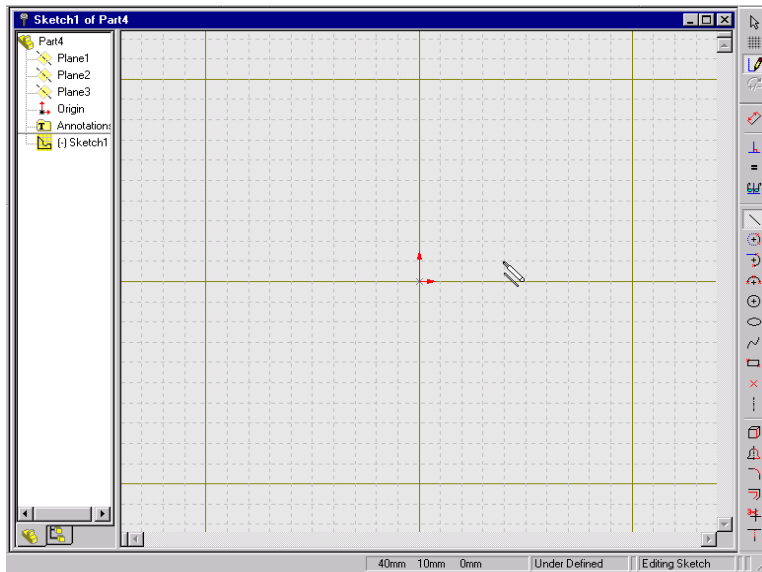
The following information is in the status bar at the bottom of the sketch window.

- The coordinates of your cursor's location.
- The state of the sketch: Over Defined, Under Defined, or Fully Defined. (See **Fully Defining Sketches** on page 2-35 for more information.)
- The text, "Editing Sketch." This makes it obvious that you are in sketch mode even if you choose to turn off the sketch grid while you are working.
- When the cursor points to a menu item or tool button, look for a brief description of the menu item or button on the left side of the status bar.

By default a sketch opens on Plane1, but you can open a sketch on Plane2 or Plane3.

To sketch on Plane2 or Plane3:

1 Click **Plane2** or **Plane3** in the FeatureManager design tree.

2 Click **View, Orientation**, and double-click ***Normal To**.

3 Open a sketch.

To insert an alternate plane, see **Creating a Construction Plane** on page 3-2.

The Sketch Origin

The sketch origin is displayed in red in the active sketch. Right-mouse click the origin in the FeatureManager design tree and select **Show** or **Hide** to toggle the display of the point of origin on or off. Click **View, Origins** in the main menu to toggle the display of origins on or off in part or assembly documents.

You can dimension to and add relations to the point of the sketch origin.

The Sketch Grid

By default, the SolidWorks sketching window provides a grid on which you can sketch. The grid guides you as you create your sketch and is generally a helpful tool. The grid is optional, however, and you may sketch without a grid.

To turn the grid off:

1 Click the **Grid** tool ⊞, or click **Tools, Options** and select the **Grid/Units** tab.

2 Click the **Display Grid** check box to remove the check mark.

The sketching window also provides a **Grid Snap** option to help you create accurate lines and points. When grid snap is on, points that you sketch or drag snap to the nearest intersection of the grid lines. This applies to the end points of lines and the centerpoints of circles and arcs.

To turn grid snap off:

1 Click the **Grid** tool ⊞, or click **Tools, Options** and select the **Grid/Units** tab.

2 Under the **Snap Behavior** heading, deselect **Snap to Points**.

For more information about additional ways to customize the sketch grid, such as changing the grid spacing, see **Grid/Units Options** on page A-19.

Inferencing Cursors and Lines

While sketching, notice that the appearance of the cursor changes to provide feedback about the cursor's current task, position, and the geometric relations that are automatically applied.

❑ When you move the cursor along the length of any sketched curve, it first appears as an *endpoint* cursor, changes to the *on-curve* cursor, and then to a *midpoint* cursor, and so on.
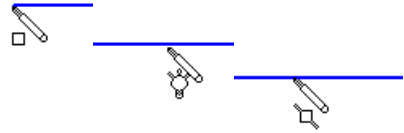
❑ When two lines intersect, the cursor changes to indicate the intersection.
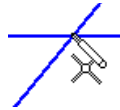
❑ When you sketch an arc, the cursor changes as it moves around the arc.
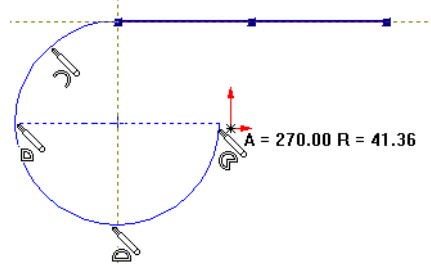
❑ The cursor provides information about dimensions as you sketch lines or arcs, such as the length, angle or radius of the sketch entity.

A = 270.00 R = 41.36

❑ When you select the various sketch or dimension tools, the cursor carries an appropriate symbol with it. Shown are the **Rectangle**, **Circle**, **Spline**, **Point**, **Trim**, **Extend**, and **Dimension** cursors.
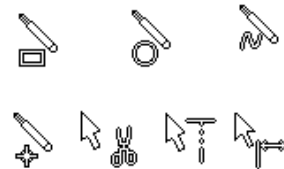
As you sketch, the SolidWorks software monitors your activity and provides *inferencing lines* to help you work more efficiently. While sketching, notice that dashed inferencing lines align your cursor with lines or points that you have already sketched and with existing model geometry.

❑ When the endpoint of any line you are creating aligns with another point that you have already sketched, a dashed inferencing line indicates this alignment. This helps you align endpoints with each other so you can sketch without using a grid.

25.09
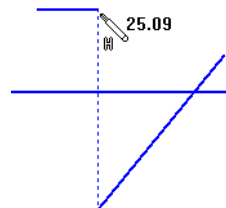
Brown inference lines indicate that a relation is added automatically; blue inference lines indicate that no relation is added.

❑ When you sketch arcs, inferencing lines and dimensions guide you to create the size and shape arc that you want.

A = 90.00 R = 27.16

A = 180° R = 30
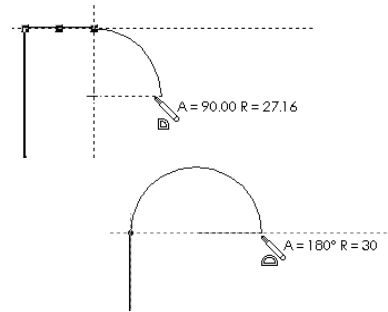
The Right-Mouse Button Menu

As you become more proficient in using the sketch tools, you will find that the right-mouse button menu provides a quick and easy way to change tasks.

For example, after you finish sketching an entity, press the right-mouse button. The menu that comes up lets you select a different sketch tool or perform a different task without moving the cursor to the toolbar or main menu.

The right-mouse menu changes depending on the task that you are performing. Only those actions that are appropriate are listed as menu items.

View	▶	Zoom To Fit
Line		Zoom To Area
Centerpoint Arc		Zoom In/Out
Tangent Arc		Rotate View
3 Point Arc		Pan
Dimension		✔ View Orientation...
Add Relation...		
Display/Delete Relations...		
Create Silhouettes		
Redraw		
Exit Sketch		
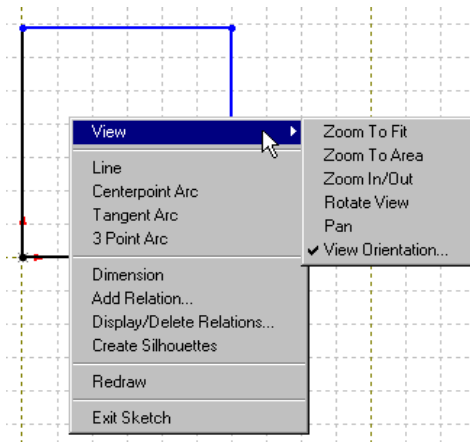
Cutting, Copying, and Pasting Multiple Sketch Entities

You can cut and paste, or copy and paste, one or more sketch entities from one sketch to another, or within the same sketch, by dragging and dropping.

To copy multiple entities, press the **Ctrl** key while selecting them, then drag while using the following modifier keys:

❑ To *copy* within the document or to another document, press the **Ctrl** key while dragging.

❑ To *move* (cut and paste) within the same sketch, press the **Shift** key.

❑ To *move between different documents*, press the **Ctrl** key and drag the sketch into the second document. Release the **Ctrl** key, hold down the **Shift** key, and drop the sketch.

NOTE: Be sure you open a sketch in the target document before you drag and drop.

Additionally, you can copy by selecting one or more sketch entities and clicking **Edit**, **Copy**, or by pressing **Ctrl+C**. Then you can paste the entity(ies) by clicking **Edit**, **Paste**, or by pressing **Ctrl+V**.

Copying and Pasting Entire Sketches

You can copy an entire sketch and paste it on a face in the current part or you can paste it into a different sketch, or part, assembly or drawing document.

To copy and paste a sketch:

1 Select the sketch in the FeatureManager design tree.

2 Click **Edit**, **Copy**, or press **Ctrl+C**.

3 Click in the sketch or document where you want to paste the sketch.

4 Click **Edit**, **Paste**, or by press **Ctrl+V**.

Creating a Sketch on the Face of a Part

To sketch on model faces:

1 Click the model face on which you want to sketch.

2 Click 🖊 or click **Insert**, **Sketch** from the main menu**.**

3 The following happens:

- A grid is displayed on the face that you selected.

- The sketch toolbar is active.

- The message "Editing Sketch" appears on the status line at the bottom of the window.
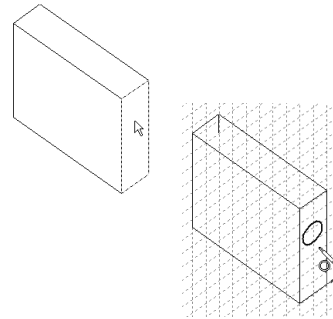
To sketch on a different face, you need to click the new face and open a new sketch. Note how the grid appears on the broad face of the base part in the this illustration.

When you are editing a sketch in **Shaded** mode, you have the option to display the sketch plane as a translucent plane. (By default, models from previous versions will have this checked.)
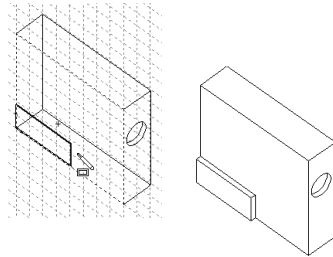
To turn the translucent sketch plane on or off:

1 Click **Tools, Options, General** and select **Display plane when shaded**.

If the display is slow, due to the translucent plane, it may be because of the **Transparency Quality** specified in **Tools, Options, Performance**. With some graphics cards, the display speed will improve if you use **Low (Screen door)** transparency.

2 Click **Tools, Options, Performance** and select **Low (Screen door)** transparency if necessary to improve the speed of the display.

Editing a Sketch

To open a sketch for editing, either select the sketch from the graphics area or the FeatureManager design tree. You can edit a sketch by adding, deleting, and dragging sketch entities, changing dimensions, changing or adding relations, and so on.

To open a sketch for editing using the right-mouse menu:

1   Right-mouse click

   • a face of a single-sketch feature (not a loft, for example)

   • a sketch entity on a part (when the sketch is not active)

   • a sketch name in the FeatureManager design tree.

2   Select **Edit Sketch** from the right-mouse menu.

   The sketch used to create the selected face or entity opens for editing.

3   If necessary, click **Rebuild** to rebuild the part.
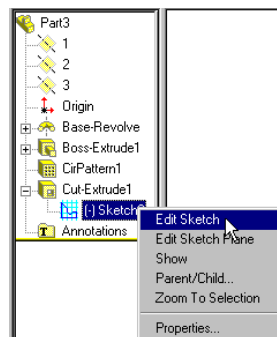
Exiting a Sketch

You exit a sketch when you:

   • Create a feature from your sketched profile. For example, when you extrude a base or boss, or a cut from your sketch.

   • Click **Rebuild** when you are in a sketch.

   • Click the **Sketch** tool [icon] when in a sketch.

   • Click **Insert, Sketch** from the main menu**.**

   • Select **Exit Sketch** from the right-mouse menu.

Edit Sketch Planes

You can change the plane of a sketch to change the location of a feature on a model. This is also useful when you are working with an imported sketch.

To change the plane of a sketch:

1   Right-mouse click the sketch in the FeatureManager design tree, and select **Edit Sketch Plane**.

2   Select a new plane by clicking a plane in the FeatureManager design tree or select a new face by clicking a different planar model face.

3   Click **Apply** or click **Cancel** to exit without making a change.

Derived Sketch

You can derive a sketch from another sketch that belongs to the same part or a sketch from a different part in the same assembly.

Note the following:

❑ If you delete a sketch from which a new sketch was derived, you are prompted that all derived sketches will automatically be underived. Click **Yes** or **No**.

❑ You cannot add or delete geometry in a derived sketch; its shape is always the same as its parent. However, you can re-orient it using dimensions and geometric relations.

❑ When you make changes to the original sketch, the derived sketch updates automatically.

❑ To break the link between the derived sketch and its parent sketch, right-click the name of the derived sketch or part in the FeatureManager design tree and select **Underive** from the menu. (After the link is broken, the derived sketch no longer updates when a change is made to the original sketch.)

To derive a sketch:

1   Select the sketch from which you want to derive a new sketch.

2   Hold the **Ctrl** key and click the face on which you want to place the new sketch.

3   Click **Insert, Derived Sketch.**

   The sketch appears on the plane of the selected face, and the sketch is opened for editing.

4   Position the derived sketch by dragging and dimensioning it to the selected face. (The derived sketch is rigid and drags as a single entity.)

To derive a sketch from a sketch in the same assembly:

1   Right-mouse click the part in which you want to place a derived sketch.

2   Select **Edit Part**.

3   Select the sketch (in the same assembly) from which you want to derive a new sketch.

4   Hold the **Ctrl** key and click the face on which you want to place the new sketch.

5   Click **Insert, Derived Sketch**.

6   Position the derived sketch by dragging and dimensioning it to the selected face. (The derived sketch is rigid and drags as a single entity.)

Does it matter where I start sketching?

When you create a new part or assembly, the three default planes are aligned with specific views. The plane you select for your first sketch determines the orientation of your part.

For example, if you choose ***Front** in the **View Orientation** dialog (or add a front view to a drawing), the view is normal to **Plane1**.
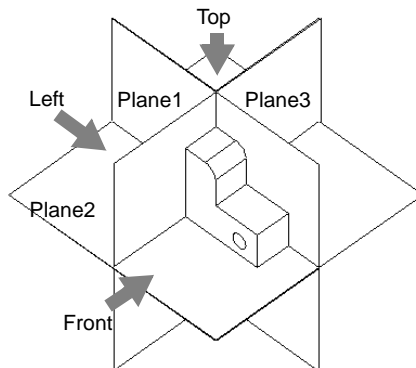
If you open a new part and start sketching *without* selecting a plane, the sketch is on **Plane1** by default and it is a *front* view.
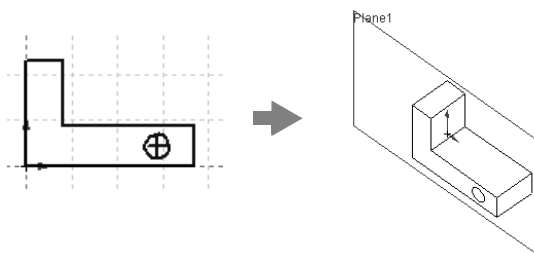
If your first sketch is a top view, you should select **Plane2** in the FeatureManager design tree *before* you click the **Sketch** icon.
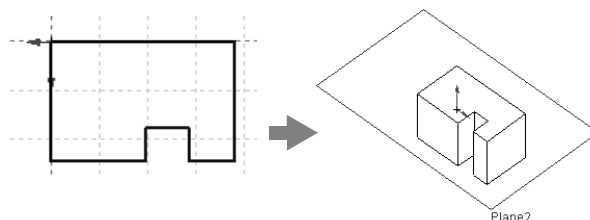
If your first sketch is a left or right view, select **Plane3**.

You do not have to use one of the default planes for your first sketch; you can create a new plane at any angle. The orientation of views is still determined by the default planes, however.

If you make a mistake or change your mind, you can reorient the part (to change "Front" to "Top" for example) with the **Update** button in the **View Orientation** dialog. For more information, see **Using Named Views** on page 1-30.

How complex should sketches be?

In many cases, you can produce the same result by creating an extruded feature with a complex profile, or an extruded feature with a simpler profile and some additional features. (You often face this choice when planning the base feature for a part.)

For example, if the edges of an extrusion need to be rounded, you can draw a complex sketch that contains sketch fillets (**A**), or draw a simple sketch and add the fillets as separate features later (**B**).
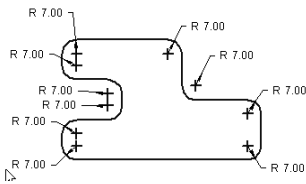
A. Complex sketch
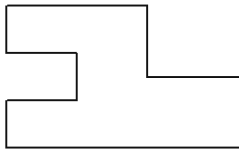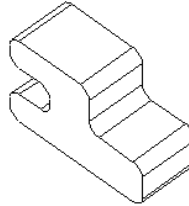
B. Simple sketch
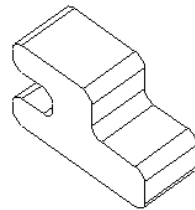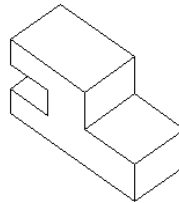
Add fillet features

Here are some things to consider:

❑ **Complex sketches rebuild faster.** Sketch fillets can be recalculated much faster than fillet features, but complex sketches can be harder to create and edit.

❑ **Simple sketches are more flexible and easier to manage.** Individual features can be reordered and suppressed, if necessary.

Reference Geometry

Reference geometry defines the shape or form of a surface or a solid. In Chapter 4, *Creating Features*, you can see how reference geometry is used in the creation of several kinds of features: construction planes are used in lofts and sweeps, split lines are used in some drafts and chamfers, an axis is used in a circular pattern, and so on.

This chapter describes how to create and use reference geometry in the SolidWorks 98Plus software, and describes the following:

❑ Creating and using construction planes

❑ Creating an axis

❑ Creating local coordinate systems

❑ Converting sketch lines to construction geometry

❑ Creating 3D reference curves

❑ Creating and using surfaces

❑ Displaying reference geometry

Creating a Construction Plane

By default, a new sketch opens on **Plane1**. You can also select **Plane2** and **Plane3** for sketching, as discussed on page 2-37. Additionally, you can create a new plane and sketch on it.

To create a plane:

1 Click **View, Planes**.

2 Click ◈ or **Insert, Reference Geometry, Plane**.

3 Select the plane type from the **Specify Construction Plane** dialog, and click **Next**.

The following plane types are available:
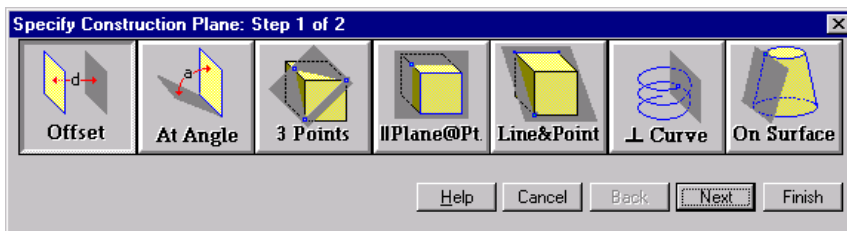
Offset. Creates a plane parallel to a plane or face, offset by a specified distance.

1 Select a plane or a planar face.

2 Specify the offset **Distance**.

3 Click **Reverse Direction**, if necessary.

4 Click **Finish**.

At Angle. Creates a plane through an edge, axis, or sketch geometry at an angle to a face or plane.

1 Select a plane or a planar face.

2 Select an edge, axis, or sketch line.

3 Specify the **Angle** between the new and existing plane.

4 Click **Reverse Direction**, if necessary.

5 Click **Finish**.

Three Point Plane. Creates a plane through three points.

1 Select three vertices, points, or midpoints.

2 Click **Finish**.

NOTE: To select a *midpoint*, right-mouse click an edge then click **Select Midpoint**. A cross appears in the middle of the edge.

Parallel Plane at Point. Creates a plane through a point parallel to a plane or face.

    1 Select a plane or planar face and a vertex, point, or midpoint.

    2 Click **Finish**.

Line and Point. Creates a plane through a line, axis, or sketch line and a point.

    1 Select an edge, axis or sketch line and a vertex, point, or midpoint.

    2 Click **Finish**.

Perpendicular to Curve at Point. Creates a plane through a point and perpendicular to an edge, axis, or sketch curve.

    1 Select an edge, axis, or sketch curve and a vertex or point.

    2 Click **Finish**.

On Surface. Creates a plane on a conical surface.

    1 Select a surface.

    2 Select a plane that intersects that surface.

    3 If no solution is displayed, you need to select an edge on that surface.

    4 Click **Other Solutions**, if necessary.

    5 Click **Finish**.

Moving, Resizing, and Copying Construction Planes

You can move, resize, and copy construction planes using the plane handles and edges.

To display the plane's handles:

- Click the plane's name in the FeatureManager design tree or in the graphics area,

– or –

- Click the edge of the plane.

Using the plane's handles and edges, you can do the following:

- Resize the plane by dragging a corner or edge handle.

- Move the plane by dragging the edge of the plane.

- Copy the plane by selecting a plane in the graphics area. Then hold the **Ctrl** key and, using the *edge,* drag to a new location.

To modify the offset distance, angle, or distance between planes:

    1 Double-click the plane to display the offset distance or angle.

    2 Double-click the dimension or angle and enter a new value in the **Modify** box.

– or –

    1 Right-mouse click the name of the plane in the FeatureManager design tree.

    2 Select **Edit Definition**.

    3 Enter new values to define the plane, and click **Finish**.

Changing Construction Plane Names

To change the names for Planes 1, 2, and 3 in the current document, click-pause-click on the plane's name in the FeatureManager design tree, and enter a new name.

To change the default names used for planes:

You can create *default* plane names that will be used on all new parts and assemblies. For example, you may want to name the planes Front, Top, and Right, instead of Plane1, Plane2, and Plane3.

1   Click **Tools, Options**.

2   Select the **Reference Geometry** tab.

3   In the **Plane Default Names** section, enter new names in the boxes that correspond to the original plane names.

4   Click **OK**.

Hide or Show Planes

You can turn the display of planes on or off.

To turn plane display on or off:

Click **View, Planes**. A check mark next to the menu item means planes are visible (except for planes you have hidden individually).

To hide or show individual planes:

1   Right-click the plane or its name in the FeatureManager design tree.

2   Click **Hide** or **Show**.

> **NOTE:** Individual planes always display when you select them, even when hidden.

Changing the Plane of a Sketch

For information about changing the sketch plane of a model, see **Edit Sketch Planes** on page 2-41.

Creating an Axis

You can use a reference axis (or construction axis) in creating sketch geometry.

Every cylindrical entity has an axis. Temporary axes are those created implicitly by arcs and cylinders in the model. You can use a temporary axis to insert a reference axis in a model.

You can set the default to either hide or show all temporary axes.

To display temporary axes:

Click **View**, **Temporary Axes**.

To turn the display of reference axes on or off:

Click **View, Axes**. A check mark next to the menu item means axes are visible (except for axes you have hidden individually).

To hide or show an individual axis:

1   Right-click the axis or its name in the FeatureManager design tree.

2   Click **Hide** or **Show**.

To create an axis in a part or assembly:

1   Click **Axis** or **Insert**, **Reference Geometry, Axis**.

2   Select from the **Options to make an axis** box:

   • **One Temporary Axis.** Select **View, Temporary Axes**, and then select the axis.

   • **Two Planes.** Select two intersecting construction planes.

   • **Two Points/Vertices.** Select two sketched points or two vertices.

   • **One Line/Edge.** Select one model edge or sketched line.

   • **One Surface.** Select a conical or cylindrical face.

   • **Point and Plane.** Select a surface or plane and a sketched point or a vertex.

3   Verify that the items listed in the **Valid Items for an Axis** box correspond to your selection(s).

4   Click **OK**.

> **NOTE:** To see the new axis, click **View, Axes** to turn on axes display.

Creating a Coordinate System

You can define a coordinate system for a part or assembly. Use this coordinate system with the **Measure** and **Mass Properties** tools; for exporting SolidWorks documents to IGES, STL, ACIS, STEP, and VDA.

To create a coordinate system:

1   Click **Coordinate System** ⬛ or **Insert**, **Reference Geometry**, **Coordinate System**.

2   Select a vertex, sketched point, midpoint of an edge, or the default point of origin on a part (or component, in an assembly). The name of the entity is listed in the **Origin** box.

3   Click in the box for the **X**, **Y**, or **Z** axes, and then click an edge, face, or vertex on the part, or a sketch segment to indicate the direction of the selected axis.

4   Click another box in the **Axes** section and select another edge, face, or vertex to define this axis.

5   If you need to change your selections, right-mouse click in the graphics area and select **Clear Selections**.

6   If you need to reverse the direction of an axis, click the **Flip** check box under its definition.

7   When the coordinate system is defined, click **OK**.

To change the coordinate system definition, right-mouse click its name in the FeatureManager design tree and select **Edit Definition**.

> **NOTE:** When you create a coordinate system, it is a good idea to give it a meaningful name to explain its purpose. Click-pause-click the coordinate system's name in the FeatureManager design tree and enter a new name.

Converting Sketch Lines to Construction Geometry

You can convert sketched lines, arcs, and splines into construction geometry to use in creating model geometry. Use the sketched entity's property sheet to make the conversion.

To open the property sheet of a sketch entity:

1 Right-mouse click the sketch entity.

2 Click **Properties** in the right-mouse menu.

3 Click the **Construction** checkbox in the **Properties** dialog box.

4 Click **OK**.

NOTE: To convert several entities at the same time, hold the **Ctrl** key and right-mouse click the sketched entities. Select **Properties** from the right-mouse menu. In the **Common Properties** dialog, click the **Construction Geometries** box, and then click **OK**.

In a drawing, you can convert sketch entities to construction geometry by selecting the entities and clicking **Tools, Sketch Entity, Construction Geometry**. For more information, see **2D Sketching** on page 8-35.

Curve

A *curve* is a type of reference geometry. You can create several types of 3D reference curves by these methods:

❑ **Projected Curve** - from a sketch projected to a model face or reference surface, or from sketched lines on intersecting planes

❑ **Helix** and **Spiral** - by specifying a circular sketch, pitch, number of revolutions, and height

❑ **Split Line** - from a sketch projected to planar or curved (silhouette) faces

❑ **Curve Through Reference Points** - from user-defined points or existing vertices

❑ **Curve Through Free Points** - from a list of X,Y, Z coordinates for points

You can then use the curves to create solid model features. For example, you can use a curve as the path or guide curve for a sweep feature, as the guide curve for a loft feature, as a parting line for a draft feature, etc.

Projected Curve

You can project a sketched curve onto a model face to create a 3D curve.

To project a curve onto a face:

1 Create a sketch containing a single open or closed curve (made up of lines, arcs, or splines) on a plane or model face.

2 Close the sketch.

3 Select the sketch, hold the **Ctrl** key and select the face where you want to project the curve.

4 Click **Insert, Reference Geometry, Projected Curve**.

The curve appears in the FeatureManager design tree and on the selected face.
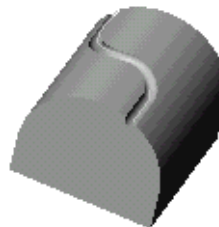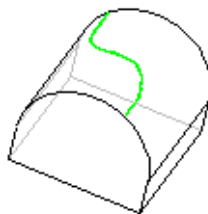
Projected curve used
as a sweep path

You also can create a 3D curve that represents the intersection of two extruded surfaces generated by creating sketches on two intersecting planes.

To create a projected curve using sketches on intersecting planes:

1 Create a sketch on each of two intersecting planes, closing each sketch when you are done.

Align the sketch profiles such that when they are projected normal to their sketch plane; the implied surfaces will intersect, creating the desired result.

2 **Ctrl**-click to select both sketches.

3 Click **Insert, Reference Geometry, Projected Curve**.

The system creates the projected curve.

You can select individual segments of a projected curve if the option **Highlight all of selected features** is <u>not</u> checked on the **Tools, Options, Edges** page. Otherwise, the entire reference curve is selected (highlighted) when any part of the curve is selected.
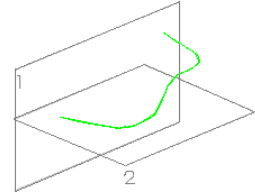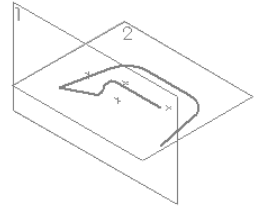
NOTE: During the creation of features such as lofts, if the reference curve is selectable, the whole reference curve is highlighted whether the **Highlight all of selected features** option is checked or not.

Example: The following is an example of using a 3D projected curve to create a swept feature. Note the addition of a profile sketch for the sweep on plane 4.
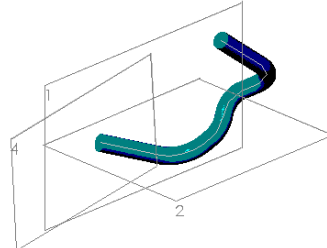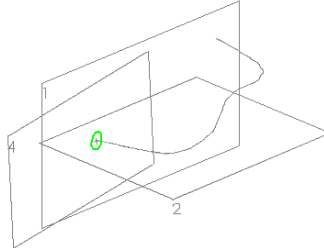
For more information about creating sweep features, see **Sweep** on page 4-10.

Composite Curve

You can create composite curves by combining reference curves, sketch geometry, and model edges into a single curve. Use this curve as a guide curve when creating a loft or a sweep.

To create a composite curve:

1 Select the curves, edges, or sketch entities to use in the composite curve.

2 Click **Composite Curve** ⊠ or **Insert, Reference Geometry, Composite Curve**.

3 Click the items (sketch entities, edges, etc.) that you want to join. The list of the selected items displays in the **Items to join** box.

4 Click **OK** to create the composite curve.

Helix

To create a helix:

1 Open a sketch and sketch a circle. The diameter of this circle controls the diameter of the helix.

2 Close the sketch.

3 Select the circle.

4 Click **Insert, Reference Geometry, Helix**.

5 Select one of the following definitions from the **Defined by** scroll box. (To define a helix, you specify two values, and the third is calculated automatically.)

   • **Pitch and Revolutions**

   • **Height and Revolutions**

   • **Height and Pitch**

6 Depending on the definition, specify these values:

   • **Height** - the parallel distance between the end points of the helix

   • **Pitch** - the parallel distance required for one full revolution

   • **Revolutions** - the number of turns in the helix

7 If you want a tapered helix, click **Taper Helix**, and specify a taper **Angle**. The **Taper Outward** option determines whether the helix tapers inward or outward.

8 If necessary, click **Reverse Direction**, modify the **Starting Angle** (on the plane of the sketched circle), and choose the direction of the turns (**Clockwise** or **Counterclockwise**).
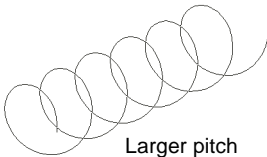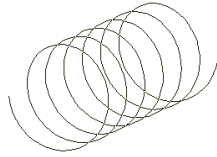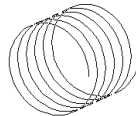
Larger pitch   Smaller pitch   Tapered
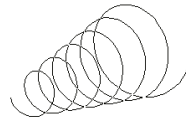
9 Click **OK**.

To create a spiral:

1 Open a sketch and sketch a circle. The radius of the sketched circle is the radius of the spiral at its start point.

2 Close the sketch.

3 Select the circle.

4 Click **Insert, Reference Geometry, Helix**.

5 In the **Defined by** scroll box select **Spiral**.
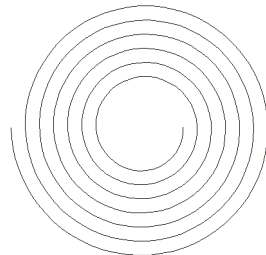
6 Enter a value for the **Pitch** - this determines the rate of change of radius for each revolution.

> **NOTE:** The **Pitch** value must be greater than 0.0 and not greater than 200000.

7 Specify the number of **Revolutions** - the number of turns in the spiral. (This does not have to be a whole number.)

8 Click the **Reverse** box to create an inward spiral.

9 Specify a value in the **Starting Angle** box to indicate where to start the spiral.

10 To choose the direction of the turns, **Clockwise** or **Counterclockwise**, click the corresponding radio button.

11 Click **OK**.

Split Line

Split Line projects a sketched curve onto selected model faces. It divides a selected face into multiple separate faces so that each can be selected. **Split Line** can be used for a variety of purposes:

- When creating a **Draft** feature with the **Parting Line** option. (For more information, see **Parting Line** on page 4-32 or **Creating Molds** on page 7-18).

- When making face blend fillets using a hold line (see **Face Blend with Hold Line** on page 4-27).

- When creating a **Radiate Surface**.

To project a split line onto a planar face:

1 Open a sketch on the face of the part and sketch a line to project as a split line.

2 Exit the sketch.

3 Click **Insert, Reference Geometry, Split Line.**

4 In the **Split Lines** dialog box, select **Projection** and click **Next**.

5 In the **Project Split Line** dialog, click the **Sketch to Project** box and select the sketch in the FeatureManager design tree.

6 Click the **Faces to Split** box, then select all the faces around the perimeter of the part that you want the split line to pass through.

7 Click **Finish**.
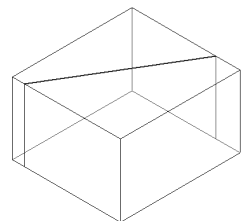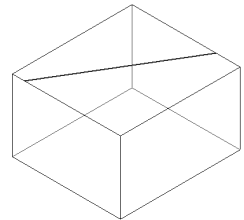
In special cases where there is an interruption in the selected face, you can project the curve in a single direction, and specify the direction of pull.

In this example, the **Face to Split** is cylindrical, and the sketch plane of the **Sketch to Project** lies in the area of a cut.

- Click **Single Direction** to project the curve in one direction only.

- Click **Reverse Direction** if the preview indicates that the curve projects the wrong way.
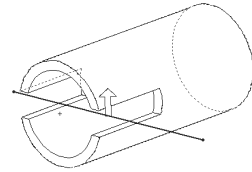
If you do not click **Single Direction**, the curve projects all the way around the cylinder.
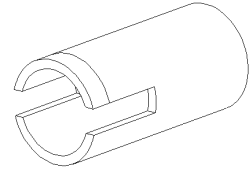
To create a silhouette split line on a part with curved or tangent faces:

1 With a part open, click **Insert, Reference Geometry, Split Line**.

2 In the **Split Lines** dialog box, select **Silhouette** and click **Next**.

3 Click the **Direction of Pull** box.

4 In the FeatureManager design tree, click a plane, edge, face, or axis that extends lengthwise through the model.

5 Click the **Faces to Split** box, then select the curved or tangent face(s) to split.

6 Click **Finish**.
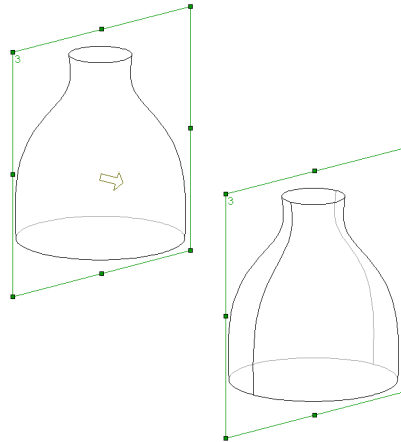
Curve Through Reference Points

Creates a 3D spline through points located on one or more planes. You can use sketch points or model vertices.

To create a curve through reference points:

1 Click **Simple Curve** ⌇, or **Insert, Reference Geometry, Curve Through Reference Points.**

The **Curve** dialog box appears.

2 Select the points in the order in which you want to create the curve.

As you select, the number of sketch items and/or vertices is updated in the **Spline Points** box, and a preview of the curve is displayed.

3 If you want to close the curve, click the **Closed Curve** box.

4 Click **OK**.

Curve Through Free Points

You can create a 3D curve from a set of points. There are two ways to generate a point list for this purpose:

❑ You can create a curve one point at a time, observing the effect on the resulting curve as you go along.

❑ You can create a list of points using a text editor or other tool, such as a spreadsheet program.

To create a curve by specifying coordinate values:

1 Click **Insert, Reference Geometry, Curve Through Free Points.**

2 In the **Curve File** dialog box, double-click the first cell in the first row.

3 Enter the coordinates for **X**, **Y**, and **Z**. Press the **Tab** key to move from cell to cell, and from row to row.

Notice that the value in the first cell in the **Point** column is set to 1. The numbers in the **Point** column define the order in which the points will be connected.
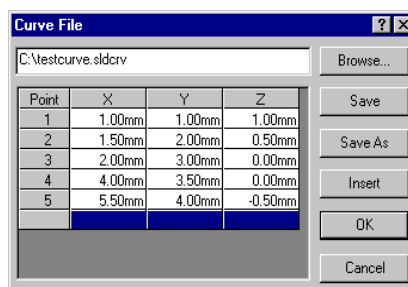
4 Continue adding as many points as needed (one row is one point). The larger the number of points, the greater your control of the resulting curve.

Notice the preview of the curve in the graphics area.

5 If necessary, you can insert a new point between existing points. Click the point number to select the row <u>below</u> where you want the new point to occur, then click **Insert**. A new row is added <u>above</u> the row you clicked. All the points below the selected row move down.

6 Double-click in a previously filled cell to go back and change its value, if necessary.

7 To delete a point, select its row and press the **Delete** key.

8 You can save the curve file for re-use. Click **Save** or **Save As**, navigate to the desired location, and specify the filename. If you do not specify an extension, SolidWorks adds the extension **.sldcrv**.

9 Click **OK** to complete the curve and close the dialog box.

To edit a curve file:

Right-click the curve name in the FeatureManager design tree, or right-click the curve in the graphics area. Select **Edit Definition** from the menu.

To create a curve from an existing file containing coordinate values:

1   Create a point file using a text editor, spreadsheet, or other utility. The format of the file should be a three-column, tab- or space-delimited list of X, Y, and Z coordinates.

   If you use a spreadsheet to create the file, save it as *text*, with either tabs or spaces as the field delimiter.

2   Click **Insert, Reference Geometry, Curve Through Free Points**.

3   Use the **Browse** button to locate the file, then click **Open**.

   The coordinates from the file are displayed in the **Curve File** dialog box. Notice the numbers in the **Points** column, indicating the order in which the points will be connected.

4   Examine the preview of the curve, and edit the coordinates if necessary, until you are satisfied with the result. (Double-click in any cell, then edit the value.)

5   Click **OK**.

Surface

A *surface* is a type of reference geometry. You can create reference surfaces by these methods:

❑ Insert a planar surface from a sketch or from a set of closed edges that lie on a plane

❑ Extrude, revolve, sweep, or loft from sketches

❑ Offset from existing faces or surfaces

❑ Import a file (For information about importing surfaces see Chapter 13, "Importing and Exporting Files.")

You can use reference surfaces in the following ways:

❑ Select edges and vertices and use as a sweep guide curve and path.

❑ Create a solid or cut feature by thickening a surface.

❑ Extrude a solid or cut feature with the end condition **Up to Surface** or **Offset from Surface**.

❑ Create a solid feature by thickening surfaces that have been knit into a closed volume.

Creating Surfaces

The methods you can use to create surfaces are similar to those you use to create solid features in the model. You can extrude, revolve, sweep, or loft a surface from sketches, knit, or offset from existing model face or reference surface. A surface may have either an open or closed profile.

To create a bounded planar surface:

- Create a non-intersecting, closed sketch and select **Insert, Reference Geometry, Planar Surface**. You can select edges and vertices, the same as a model.
  To make changes to this planar surface, you can edit the sketch.

- Select a set of closed edges in a part or assembly, and select **Insert, Reference Geometry, Planar Surface**. (All edges in the set must be on the same plane.)
  You can change this planar surface by right-mouse clicking the planar surface and selecting **Edit Definition**.
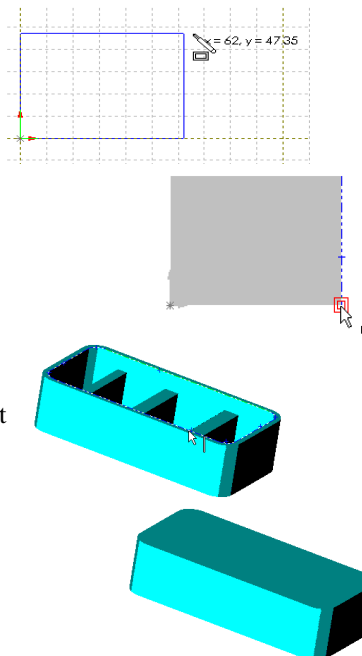
To extrude a surface:

1 Sketch the profile of the surface.

2 Click **Insert, Reference Geometry, Extruded Surface**.

3 In the **Extrude Feature** dialog box, on the **End Condition** tab, choose the **Type**, and specify the **Depth**.

4 Examine the preview. If the offset is in the wrong direction, click **Reverse Direction**.
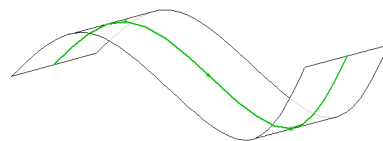
Extruded surface (MidPlane type) from a sketched spline

5 Click **OK**.

See **Extrude** on page 4-2 for more information about extruded features.

To revolve a surface:

1   Sketch a profile and a centerline around which to revolve the profile.

2   Click **Insert, Reference Geometry, Revolved Surface**.

3   Enter the value for **Degrees**, and click **Reverse Direction** if necessary.

4   Click **OK**.

See **Revolve** on page 4-9 for more information about revolved features.
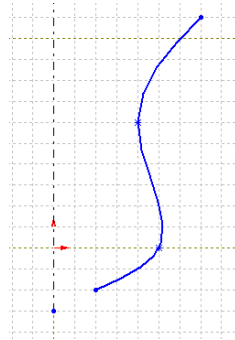
To loft a surface:

1   Create a plane for each profile section of the loft. (The planes do not need to be parallel.)

2   Sketch the section profiles and guide curves (if any are needed) on the planes. Close each sketch after the sketch is complete.

3   Click **Insert, Reference Geometry, Lofted Surface**.

4   Click the profiles in the order in which you want them to be connected. Select corresponding segments on each profile; the vertex closest to the selection point is used to connect the profiles.

5   If you are using guide curves, click the **Guide Curve** box in the **Loft Surface** dialog, then select the guide curves.

6   Choose from the options **Maintain Tangency**, **Close along Loft Direction**, and **Advanced Smoothing** as desired.

7   Click **OK**.

See **Loft** on page 4-15 for more information about lofted features.

Guide curve
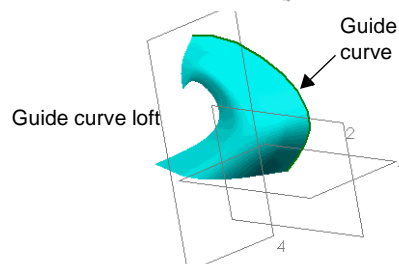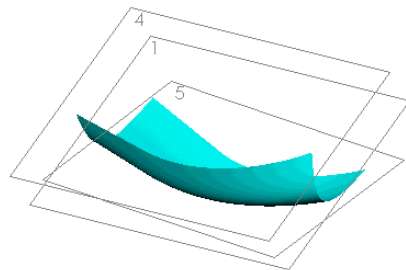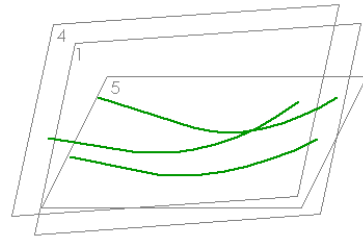
Guide curve loft

To sweep a surface:

1 Create planes as needed for sketching the sweep profile, sweep path, and the guide curves (if any are needed).

Profile   Path

2 Sketch the sweep profile and path on the planes that you created.
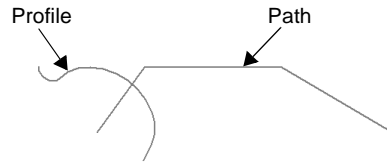
> **NOTE:** You may also sketch the sweep path on a model face or use a model edge for the path.

3 If you are using one or more guide curves, create coincident/pierce relations between the guide curve(s) and the profile.

4 Click **Insert, Reference Geometry, Swept Surface**.

5 Click the **Sweep Section** box, then click the profile sketch.

6 Click the **Sweep Path** box, then click the path sketch or model edge, if that is what you are using for a path.

7 If you are using guide curves, click the **Advanced** tab. Click the **Guide Curve** box, then click the sketch, curve, or model edge for the guide curve(s).

8 Select options as desired under **Orientation/Twist Control**, **Advanced Smoothing**, and **Maintain Tangency**.

9 Click **OK**.
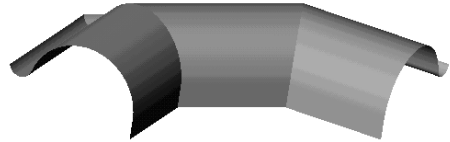
See **Sweep** on page 4-10 for more information about creating swept features.

To offset a surface:

1 Click **Insert, Reference Geometry, Offset Surface**.

2 In the **Offset Surface** dialog, specify the **Offset** value.

3 Select the model surface from which to create an offset. If you want to offset multiple surfaces, hold the **Ctrl** key as you select the model surfaces.

4 Examine the preview, and click **Reverse** if necessary.

5 Click **OK**.
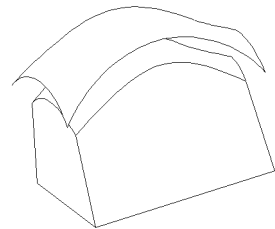
Offset Surface from a lofted model surface

Radiate Surface

You can create surfaces by radiating a parting line, an edge, or a set of contiguous edges inwards or outwards, and parallel to a selected plane.

You can use radiate surfaces to cut a mold into two or more parts. For more information about using radiate surface to cut a mold, see **Mold with Complex Parting Lines** on page 7-21.

To create a surface by radiating a parting line:

1 Sketch a curve on the face of a part.

2 Project the curve as a **Split Line**. (See **Split Line** on page 3-11 for more information.)

3 Click **Insert**, **Reference Geometry**, **Radiate Surface**.

4 Select a reference plane that is parallel to the direction in which you want the surface to radiate.

5 Specify the width of the surface in the **Radiate distance** box.

6 Click the split line as the **Edges to Radiate**. Note the arrow direction. To specify a different direction, click **Reverse direction**.

7 Select **Propagate along tangent faces** if the part has tangent faces and you want the surface to continue around the part.

8 Click **OK**.
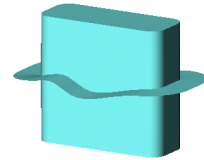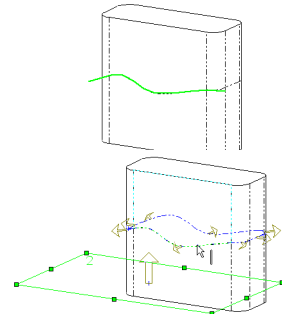
Knit Surfaces

Use the **Knit Surface** function to combine two or more reference surfaces or faces into one. The edges of the surfaces or faces must be adjacent and not overlapping. They do not need to be on the same plane.

This is an illustration of two views of a pair of adjacent surfaces: a lofted surface and a swept surface.

Surface-Loft

To knit surfaces:

Click **Insert, Reference Geometry, Knit Surface**, and select the surfaces or faces in the graphics area. You can also select surfaces in the FeatureManager design tree.

Surface-Sweep

The result is a single surface listed in the FeatureManager design tree as **Surface-Knit**<*n*>. There is no difference in the appearance of the planes after knitting.
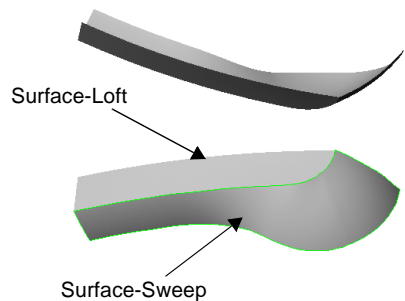
To thicken knit surfaces:

1 Click **Insert, Boss, Thicken**.

2 Specify a value in the **Thickness** box.

3 Select the side to thicken, or select **Thicken both sides**.

4 Click the **Surface to thicken** box, and select the **Surface-Knit** icon in the FeatureManager design tree.

5 Click **OK**.

How to use the Seed face option:

To knit surfaces using the **Seed** option, you must use **Radiate Surface**.

1 Project a split line onto the model and use it to create a radiate surface that completely encompasses the model. (See **Radiate Surface** on page 3-18.)

2 Click **Insert, Reference Geometry, Knit Surface**.

3 In the **Faces and surfaces** list box, select the radiate surface.

4 Click the **Seed** box, and then click any face on the model that you want to knit into a reference surface.
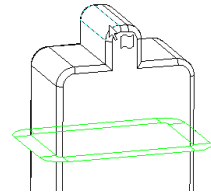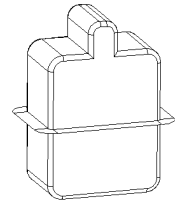
The **Seed** surface propagates the face selection to all the model faces until it reaches the radiate surface. It then knits all those selected faces into a reference surface.

(For this illustration, the **Section View** tool was used to separate the model from the reference surface.)

To fill a volume with a loft between two reference surfaces:

You can create a solid body by lofting between two reference surfaces.

1   Create adjacent reference surfaces.

2   Open a new sketch, sketch a guide curve, and close the sketch.

3   Click **Insert Boss, Loft**.

4   Click the **Profiles** box and select the reference surfaces.

5   Click the **Guide Curve(s)** box and select the guide curve sketch.

The resulting lofted body is solid.
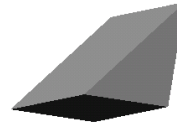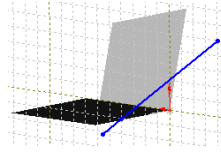
Using Surfaces to Create Features

A reference surface has no thickness. You can thicken a surface to create model geometry, or to cut into existing geometry. You can also use a zero-thickness surface to cut through a model, similar to cutting with a parting line.

To create a solid feature by thickening a surface:

1   Create a surface as described in the previous section, or import an IGES surface.

2   Click **Insert**, **Base**, **Thicken** (if there are no features in the part) or **Insert, Boss, Thicken**.

3   Select the surface.

4   In the **Thicken Feature** dialog, specify the desired **Thickness** for the feature.

5   Specify the side(s) you want to thicken. Examine the preview to see the effect.
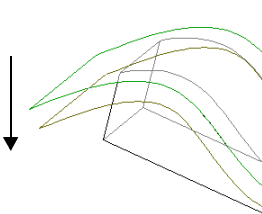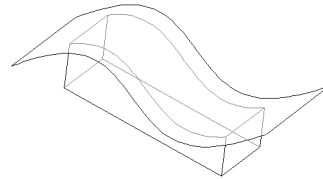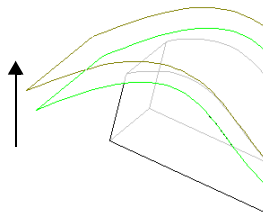
Thicken Side 1   Thicken Side 2   Thicken Both Sides
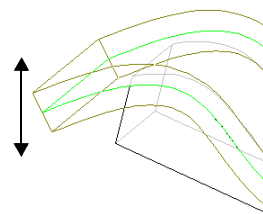
If you are creating a boss, the thickened surface *must* intersect the existing part.

6 Click **OK**.
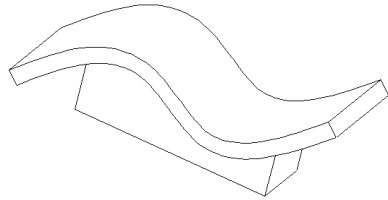
To cut a part with a surface:

1 Create a reference surface in the part model, or import one.

2 Select the surface.

3 Click **Insert, Cut, With Surface**.

Note the preview arrow, indicating which side of the part will be cut away.

4 In the **Surface Cut** dialog box, click **Flip the Side to Cut Away** to reverse the direction of the cut, if necessary.

5 Click **OK**.
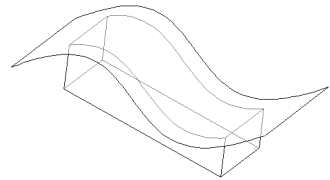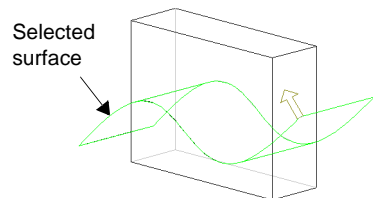
6 To hide the surface, if desired, right-mouse click the **Surface** feature in the FeatureManager design tree and select **Hide**.

> **NOTE:** You can also cut a part with a surface by using the **Up To Surface** option when extruding a cut.
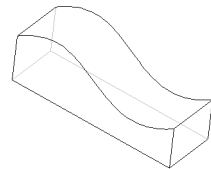
To cut a part by thickening a surface:

1   Create a reference surface in the part model, or import one.

2   Select the surface.

3   Click **Insert**, **Cut**, **Thicken**.

4   In the **Thicken Feature** dialog, enter the desired thickness for the feature.

5   Specify which side to thicken.

6   Click **OK**.

Selected surface
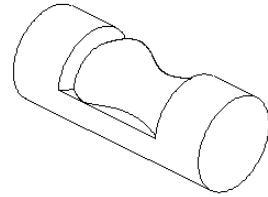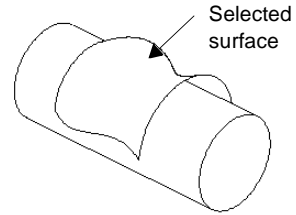
Displaying Reference Geometry

Reference geometry is a necessary tool, but when it is visible, it may make the image of the part appear too complex and hard to understand. You can set system-wide visibility options to filter out various types of reference geometry. Click **Tools**, **Options**, **Reference Geometry** and select the kinds of reference geometry that you want to see, by default. See *Appendix A,* **Reference Geometry Options** on page A-23 for more information.

To set the visibility of reference geometry in the current document, use the options under the **View** menu.

By default, reference geometry is hidden in drawing views. For information about importing reference geometry to drawings, see **Reference Geometry in a Drawing** on page 9-15.

4

Creating Features

Features are the individual shapes that, when combined, make up the part. You can also add some types of features to assemblies. Some features originate as sketches; other features, such as shells or fillets, are created when you select the appropriate menu command and define the dimensions or characteristics that you want. This chapter describes the following:

❑ Features toolbar

❑ Base, Boss, and Cut

❑ Extrude, Revolve, Sweep, and Loft

❑ Fillet/Round, Chamfer, and Draft

❑ Hole - Simple and Hole Wizard

❑ Shell

❑ Rib

❑ Dome

❑ Shape

❑ Pattern - Circular, Linear, and Mirror

The Features Toolbar

For many of the features described in this chapter, there is an icon on the Features toolbar. For others, an icon may be available, but not displayed by default. You can customize the toolbars to include icons for some of the functions you use most.

Click **Tools, Customize**, and click the **Commands** tab. Choose a category, and examine the available buttons. Drag the buttons you want into any toolbar, delete buttons you do not want from any toolbar, or rearrange buttons to your liking.

Base/Boss

The first feature of every part created in SolidWorks is the *base* feature. (There is only one *base* feature.)

A *boss* is a feature that adds material to a part.

A *base* or *boss* may be created by **Extrude**, **Sweep**, **Revolve**, or **Loft** (from one or more sketches), or **Thicken** (from a surface).

Cut

A *cut* is a feature that removes material from a part or an assembly.

A cut may be created by **Extrude**, **Sweep**, **Revolve**, or **Loft** (from one or more sketches), or **Thicken** or cut **With Surface** (from a surface).

Extrude

Extrude extends the sketched profile of a feature in one or two directions as either a *thin feature* or a *solid feature*.

An extrude operation can either add material to a part (in a base or boss) or remove material from a part (in a cut or hole).

See **Surface** on page 3-14 for information about using **Extrude** to create a surface.
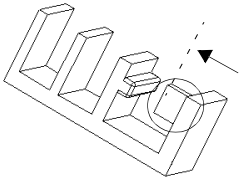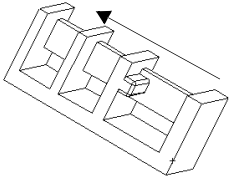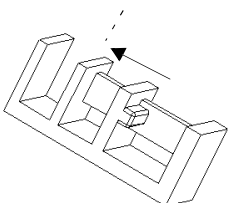
Solid Feature   Thin Feature
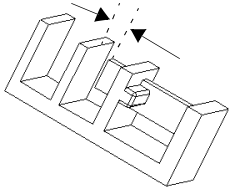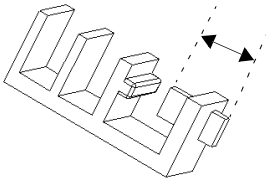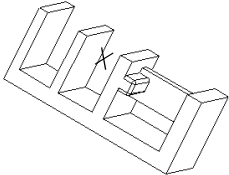
The **End Condition** tab of the
Extrude Feature dialog lets you
define the characteristics of
extruded features created with these
commands on the **Insert** menu:

- **Base, Extrude**
- **Boss, Extrude**
- **Cut, Extrude**
- **Features, Hole, Simple**
- **Features, Hole, Wizard**
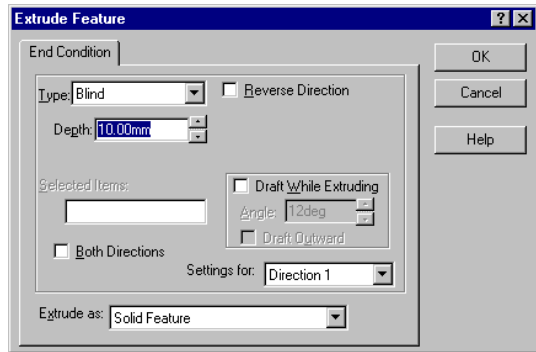
The following table describes the **End Condition** tab of the **Extrude Feature** dialog box
and the various options for extruded bosses and cuts.

Type	Example	Description
Blind		Extends the feature from the sketch plane for a specified distance (**Depth**).
Through All		Extends the feature from the sketch plane through all existing geometry.
Up To Next		Extends the feature from the sketch plane to the next face, or set of faces, that intercepts the entire profile. The *next* face must be on the same part.
Up To Surface		Extends the feature from the sketch plane to the selected face or surface.

Type	Example	Description
Offset From Surface		Extends the feature from the sketch plane to a specified distance from the selected face or surface.
Mid Plane		Extends the feature from the sketch plane equally in both directions. (**Depth** specifies the *total* depth, not the depth in each direction.)
Up To Vertex		Extends the feature from the sketch plane to a plane that is parallel to the sketch plane and passing through the specified vertex.

Specifying End Conditions

❑ **Type** determines how far the feature extends. See the preceding chart for examples.

   If you choose **Blind** or **Mid Plane**, you have to specify the **Depth**.

   If you choose **Offset From Surface**, you have to specify the **Offset**.

❑ **Depth** specifies the depth of the extrusion.

❑ **Reverse Direction** lets you extend the feature in the opposite direction from that shown in the preview in the graphics area.

❑ **Link to Thickness** is used primarily for bosses on sheet metal parts. Checking this option automatically links the depth of an extruded boss to the thickness of the base feature.

❑ **Flip Side to Cut** appears only when you are extruding a cut. By default, material is removed from the *inside* of the profile. Selecting **Flip Side to Cut** removes all material from the outside of the profile.
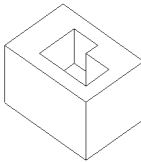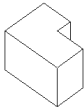
Default cut   Flip side cut

❑ **Draft While Extruding** lets you add draft to a feature while you extrude it. If you check this option, you must set the draft **Angle**, and you can check **Draft Outward**, if needed.

❑ **Selected Items.** If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area. The selection is indicated in the **Selected Items** box.
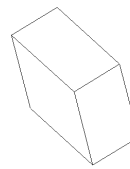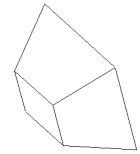
No draft 25° draft angle – inward

❑ **One Direction** or **Both Directions.**

- **One Direction** extrudes the feature in one direction from the sketch plane. This is the default (**Both Directions** *unchecked*).

- **Both Directions** extrudes the feature in both directions from the sketch plane. Specify all the settings for the first direction (**Direction 1**), then select **Direction 2** from the **Settings for** box, and specify the settings for the second direction.

NOTE: Observe the preview to verify the direction and depth of the feature.

❑ **Extrude As** a **Solid Feature** or a **Thin Feature**.

- **Solid Feature** adds (or removes) solid volumes to the model.

- **Thin Feature** adds (or removes) thin-walled volumes to the model. A **Thin Feature** base can also be used as a basis for a sheet metal part. For more information, see Chapter 11, "Sheet Metal."
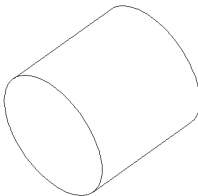
A circle extruded as a solid feature
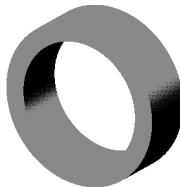
A circle extruded as a thin feature
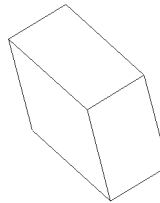
A rectangle extruded as a solid feature

A rectangle extruded as a thin feature, with draft

Thin Features

The following menu items are for thin features only, and appear when you choose the **Thin Feature** tab.

> **NOTE:** Whenever you sketch an open profile, the **Thin Feature** tab appears in the **Extrude Feature** dialog box. When you sketch a closed profile, you need to select **Thin Feature** in the **Extrude As** box to display the **Thin Feature** tab.

❑ **Type** specifies whether to extrude the thin feature in **One-Direction**, **Mid-Plane**, or **Two-Direction**.

- **One-Direction** extrudes the sketch in one direction using the specified wall thickness.

- **Mid-Plane** extrudes the sketch in both directions, dividing the specified wall thickness equally on both sides of the sketch geometry.

- **Two-Direction** extrudes the sketch in both directions, using a different wall thickness on each side of the sketch geometry (as specified for **Direction 1** and **Direction 2**).
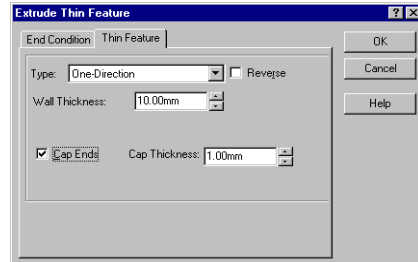
❑ **Wall Thickness** specifies the thickness of the thin feature wall.

❑ **Reverse.** The default is to add the wall thickness to the outside of the sketched profile. Clicking **Reverse** adds the wall thickness to the inside of the sketched profile.

For thin feature base extrusions only, you can specify the following additional options:

❑ If you create a *closed profile* sketch, you can use the **Cap Ends** option.

This option covers (caps) the ends of the feature, creating a hollow part. If you check this option, you must also specify the **Cap Thickness**.

❑ If you create an *open profile* sketch, you can use the **Auto Fillet** option.

This option creates a round at each edge where lines meet at an angle. If you check this option, you must also specify the **Fillet Radius** (the inside radius of the round).

Extruded Base/Boss ![icon]

To extrude a solid or thin feature base (or boss):

1 With a sketch active, click one of the following:

- **Insert**, **Base**, **Extrude**
- **Insert, Boss, Extrude**
- The **Extruded Base/Boss** button ![icon] on the Features toolbar

A sketched rectangle...

...extruded to make a box

The **Extrude Feature** dialog box appears.
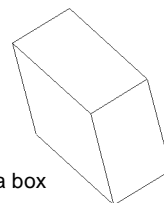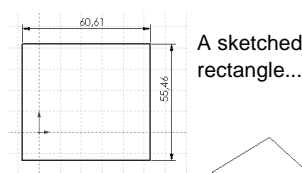
2 Select the **Type** of extrusion. (See the table on page 4-3 for more information.)

3 Set the desired **Depth** of the extrusion.

For bosses on sheet metal parts, click **Link to Thickness** to link the depth of the boss to the thickness of the base feature.

4 To add a **Draft** angle, click **Draft While Extruding**. Enter a value for the draft **Angle**, and click **Draft Outward**, if desired.

5 If the **Type** you specified relies on the selection of a surface or vertex, click that item in the graphics area. The selection is indicated in the **Selected Items** box.

6 Examine the preview and click **Reverse Direction** to extrude in the opposite direction, if necessary.

7 To extrude the feature in two directions from the sketch plane, click **Both Directions**, click **Direction 2**, and enter the settings for the second direction.

8 Select **Solid Feature** or **Thin Feature** in the **Extrude as** box.

If you are extruding a **Solid Feature**, skip ahead to Step 10. If you are extruding a **Thin Feature**, proceed to Step 9.

9 Select the **Thin Feature** tab.

- **a)** Select a wall thickness **Type**. (See the descriptions on page 4-6.)
- **b)** Specify the **Wall Thickness**, and if you want to add the wall thickness to the opposite side of the sketched profile, click **Reverse**.
- **c)** To add fillets to the corners of an open profile thin base feature, click **Auto Fillet**, and specify the desired **Fillet Radius**.
- **d)** To cap the ends of a closed profile base feature, click **Cap Ends**, and specify the desired **Cap Thickness**.

10 When you are satisfied with the preview, click **OK**.

Extruded Cut 🔲

To extrude a solid or thin feature cut:

1 With a sketch active, click **Insert**, **Cut**, **Extrude** or click **Extruded Cut** 🔲 on the Features toolbar.

This sketch for a cut on a cylinder face...
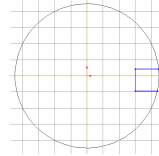
The **Extrude Feature** dialog box appears.

2 Select the desired **Type**. (See the table on page 4-3 for more information.)

3 Specify the **Depth**.

...results in this extruded cut.
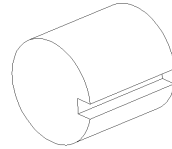
4 Examine the preview and click **Reverse Direction**, if necessary.

5 Select **Flip Side To Cut**, if desired.

6 To add a **Draft** angle, click **Draft While Extruding**. Enter a value for the draft **Angle**, and click **Draft Outward**, if desired.

7 If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area. The selection is displayed under **Selected Items**.

8 To extrude the feature in two directions, click **Both Directions**, and indicate settings for **Direction 1** and **Direction 2**.

9 Specify **Solid Feature** or **Thin Feature** in the **Extrude as** box.

If you are extruding the cut as a **Solid Feature**, skip ahead to Step 11. If you are extruding the cut as a **Thin Feature**, proceed to Step 10.

10 Select the **Thin Feature** tab at the top of the dialog box.

   a) Select a wall thickness **Type**. (See the descriptions on page 4-6.)

   b) Specify the **Wall Thickness**.

   c) To extrude in the opposite direction, click **Reverse**.

11 When you are satisfied with the preview, click **OK**.
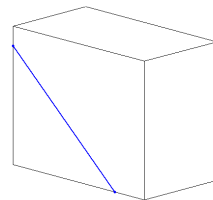
Thin Feature Cut from a sketched line
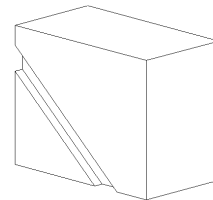
Revolve

Revolve creates a base, boss, or cut by revolving a sketch around a centerline. The default angle is 360°. See **Surface** on page 3-14 for information about using **Revolve** to create a surface.

To make a thin or solid revolved feature:

1 Sketch a centerline and a closed set of curves that do not self-intersect.

2 Click one of the following:

- **Insert**, **Base, Revolve**

- **Insert, Boss, Revolve**

- **Revolved Base/Boss** on the Features toolbar

- **Insert, Cut, Revolve**

- **Revolved Cut** on the Features toolbar

3 Choose a direction **Type** (**One-Direction, Mid-Plane** or **Two-Direction**).

4 Specify the rotation **Angle**.

- If you chose **Type** of **Mid-Plane**, the **Angle** is divided equally on both sides of the sketch plane.

- If you chose **Type** of **Two-Direction**, specify the **Angle** for each direction.

The preview shows the direction of rotation.

5 Click **Reverse** to rotate the feature in the opposite direction.

6 Under **Revolve as**, select **Solid Feature** or **Thin Feature**.

7 For a **Thin Feature** only, click the **Thin Feature** tab, choose a direction (for the **Wall Thickness**) from the **Type** list box, and specify the **Wall Thickness**.

8 Click **OK**.
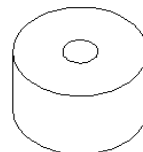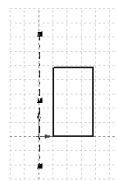
Sweep

Sweep creates a base, boss, or cut by moving a profile (section) along a path, according to these rules:

- The profile must be closed.
- The path may be open or closed.
- The path may be a set of sketched curves contained in one sketch, a reference curve, or a set of model edges.
- The start point of the path must lie on the plane of the profile.
- Neither the section, the path, nor the resulting solid can be self-intersecting.

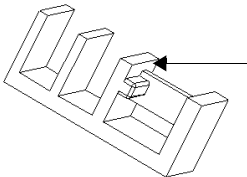See **Surface** on page 3-14 for information about using **Sweep** to create a surface.

Simple Sweep

To create a simple sweep:

Sketch on multiple planes, following these general steps:

1   Sketch a *closed, non-intersecting* profile on one plane.

2   Create the path that the profile will follow. Use a sketch, existing model edge(s) or reference curve(s).

3   Click one of the following:

- **Insert, Base, Sweep**
- **Insert, Boss, Sweep**
- **Insert, Cut, Sweep**
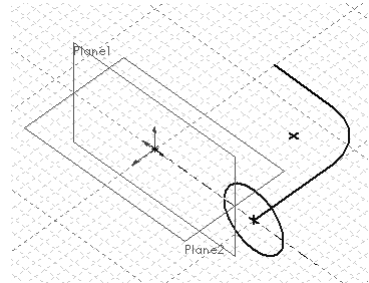
The **Sweep** dialog box appears. For a simple sweep, you only need to use the **Sweep** tab in the dialog box.

4   Click the **Sweep Section** box, then select the section profile either in the model or in the FeatureManager design tree.

5   Click the **Sweep Path** box, then select the sketch, edge, or curve that you want to use as the path either in the model or in the FeatureManager design tree.

6 In the **Orientation/Twist Control** box:

- Select **Follow path** if you want the section to remain at the same angle with respect to the path at all times. In this example, the section is always at 90° to the path.

- Select **Keep normal constant** if you want the section to remain parallel to the beginning section at all times.

7 If you picked an edge as the path, and you want to continue the sweep along all tangent edges, click **Propagate Along Tangent Edges**.

Follow path

section remains parallel

Keep normal constant
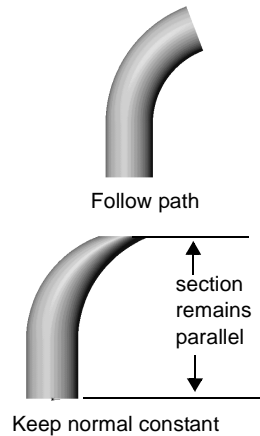
8 If you want to continue the sweep profile up to the last face encountered at the ends of the path, click **Align with End Faces**.

9 Click **OK**.

For examples of creating simple sweep features, refer to the tutorial, *Learning to Use SolidWorks 98Plus*, Chapter 6.
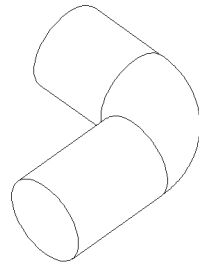
Sweep with Guide Curves

You can use guide curves to control the intermediate profiles as the sketch is swept along the path.

To create a sweep using guide curves:

1 Create the guide curve(s), either as a sketch, a reference curve, or a model edge.

2 Create a path for the sweep. This also can be a sketch, reference curve, or model edge.

> **NOTE:** When using guide curves to create a sweep, the path segments must be *tangent* (no angled corners).

Path

Guide Curve

Section

3 Sketch the sweep section. It is recommended that you sketch the section on a plane that is normal to the end of the sweep path.
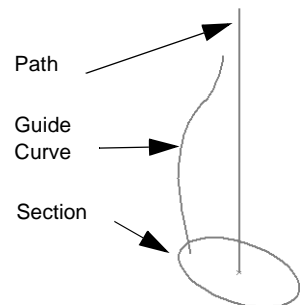
It is important to create the section *after* the path and guide curve(s). The intermediate profiles of the sweep are *dependent* on both the path and guide curves for their definition.

NOTE: To help you with creating the path and guide curves, you may want to create an auxiliary profile sketch first. Then, after the guide curves and path are done, derive a copy of the sketch, then underive it to break the link to the original sketch. Use this as the section in the remaining steps. See **Derived Sketch** on page 2-42 for more information.

As you sketch the section, pay special attention to any relations that may be added automatically, such as **Horizontal** or **Vertical**. These relations influence the shape of the intermediate sections, and may produce results that are not what you want. If necessary, use **Display/Delete Relations** 🔳 to delete the unwanted relations. This allows the intermediate sections to twist as desired.

4   Add **Pierce** relations in the *profile* sketch, where the guide curve(s) intersect the profile. **Pierce** relations cause the section to change size and/or shape along the path. It is important that the *section* be constrained to the *curves*, and not vice-versa.

5   Click **Insert, Base/Boss/Cut, Sweep**. The **Sweep** dialog box appears.

6   On the **Sweep** tab, click the **Sweep Section** box, then select the profile sketch in the FeatureManager design tree or in the graphics area.

7   Click the **Sweep Path** box, then select the path (sketch, curve, or model edge).

8   Under **Orientation/Twist Control**, there are two additional choices besides **Follow path** and **Keep normal constant** (see page 4-11):

- **Follow path and 1st guide curve -** The twist of the intermediate sections is determined by the vector from the path to the first guide curve. The angle between the horizontal and the vector remains constant in the sketch planes of all of the intermediate sections.

- **Follow 1st and 2nd guide curves -** The twist of the intermediate sections is determined by the vector from the first to the second guide curve. The angle between the horizontal and the vector remains constant in the sketch planes of all of the intermediate sections.

GuideCurve

GuideCurve

Section

Path

Follow path and 1st
guide curve

Follow 1st and 2nd
guide curves (twisted)
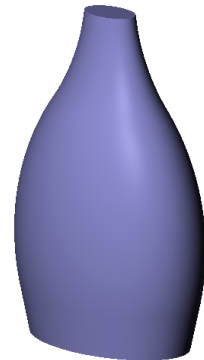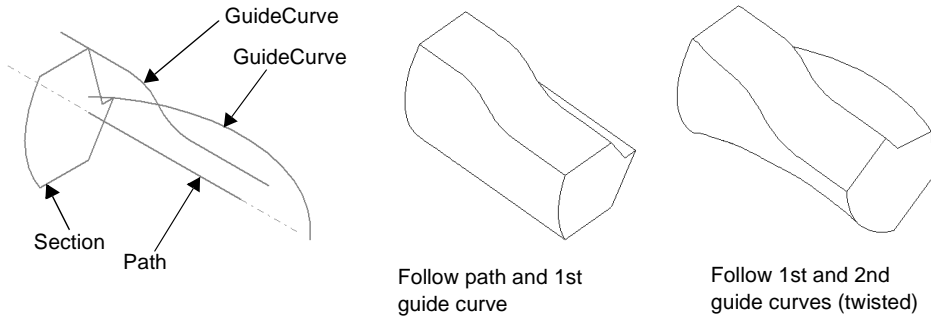
9 On the **Advanced** tab, click the **Guide Curve(s)** box, then select the guide curve(s). Use the **Up** and **Down** buttons to rearrange the order of the guide curves if needed.

10 Under certain conditions, these additional options for sweeps with guide curves are available on the **Advanced** tab:

- **Maintain Tangency** - if the sweep section has tangent segments, checking this option causes the corresponding surfaces in the resulting sweep to be tangent. Faces that can be represented as a plane, cylinder, or cone are maintained. Other adjacent faces are merged, and the sections are approximated. Sketch arcs may be converted to splines.

- **Advanced Smoothing** - if the sweep section has circular or elliptical arcs, the sections are approximated, resulting in smoother surfaces. Sketch arcs may be converted to splines.

11 Specify the tangency options, if desired (the default is **None**). See **End Tangency Options for Sweeps** on page 4-14.

12 Click **OK**.

To view the intermediate sections in the sweep:

You can preview the intermediate sections created along the sweep path. This option is only available for sweeps with guide curves.

1 Right-click the sweep feature in the FeatureManager design tree and select **Edit Definition**.

2 On the **Advanced** tab, click **Show Intermediate Profiles**.

3 Use the up and down arrows beside the **Profile Number** box to scroll through the profiles.

> **NOTE:** You can activate this option before creating the sweep, if desired.

Additional notes on using Guide Curves:

❑ The path and the guide curves may differ in length. The sweep length is determined by the shortest overlapping distance.

- If the guide curves are longer than the path, the sweep is as long as the path.

- If the guide curves are shorter than the path, the sweep is as long as the shortest guide curve.

❑ Guide curves may meet at a common point, which is the apex of the swept surface.

❑ You can use any of the following items as a guide curve: sketched curves, model edges, or reference curves. See **Curve** on page 3-8 for information about creating reference curves.

End Tangency Options for Sweeps

With a simple sweep, end tangency is controlled by choosing either **Follow path** or **Keep normal constant** in the **Orientation/Twist Control** list.

When you use guide curves to create a sweep, you have additional options for controlling the tangency at the ends of the path. You specify the **Start Tangency** and **End Tangency** options on the **Advanced** tab of the **Sweep** dialog box. These tangency options are only effective when you use guide curves in the sweep.

Choose from these options:

❑ **None**. No tangency is applied.

❑ **Path Tangent**. The sweep is normal to the path at the start or end.

❑ **Direction Vector**. The sweep is tangent to a selected linear edge or axis, or to the normal of a selected plane. Click the **Direction Vector** button, then click the edge, axis, or plane.

❑ **All Faces**. The sweep is tangent to the adjacent faces of existing geometry at the start or end. This selection is only available when the sweep is attached to existing geometry.

These tangency options are similar to the tangency options for lofts. Refer to the table on page 4-24 for illustrated examples.

Loft

Loft creates a feature by making transitions between profiles. A loft can be a base, boss, or cut.

You create a loft using two or more profiles. Only the first and/or last profile can be a point.

For a solid loft, the first and last profiles must be model faces or faces created by split lines, planar profiles, or reference surfaces.

See **Surface** on page 3-14 for information about using **Loft** to create a surface.

A circle on plane 1 and a point on plane 4
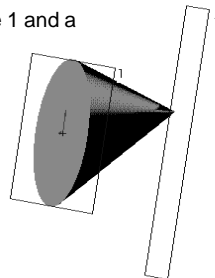
Simple Loft

To create a simple loft:

1 Set up the planes needed for the profile sketches. Use existing planar faces and construction planes, or create new construction planes. Planes do not have to be parallel.

2 Sketch the profiles.

3 Click one of the following:

- **Insert**, **Base, Loft**
- **Insert, Boss, Loft**
- **Insert, Cut, Loft**

4 Select the profiles in order, by clicking a corresponding point on each profile.

You do not have to select the vertices precisely; the vertex closest to the selection point is used. A preview curve connecting the selected entities is displayed.
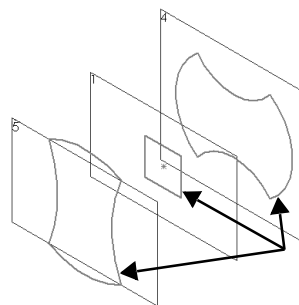
Click corresponding points

5 Examine the preview curve.

- If the preview curve looks wrong, you may have selected the sketches in the wrong order. Use the **Up** or **Down** buttons to rearrange the sketches.
- If the preview curve indicates that the wrong vertices will be connected, click the profile with the wrong vertex connected to deselect it, then click again to select a different point on the profile.
- To clear all selections and start over, right-click in the graphics area, select **Clear Selections**, and try again.

6   Choose from these options as needed:

- Click **Close Along Loft Direction** - to create a closed body along the loft direction. This connects the last sketch and the first sketch automatically.

- Click **Maintain Tangency** to cause the corresponding surfaces in the resulting loft to be tangent if the corresponding lofting segments are tangent. Faces that can be represented as a plane, cylinder, or cone are maintained. Other adjacent faces are merged, and the sections are approximated. Sketch arcs may be converted to splines.

- Click **Advanced Smoothing** to obtain smoother surfaces. This option is available only if the loft sections have circular or elliptical arcs. The sections are approximated, and sketch arcs may be converted to splines.

7   Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-23.

8   Click **OK**.

For examples of creating simple loft features, refer to the tutorial, *Learning to Use SolidWorks 98Plus*, Chapter 7.
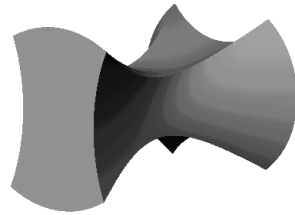
Loft with Non-planar Profiles

To create a loft using non-planar profiles:

1 Create at least one non-planar profile. The non-planar profile can be a model face or a model edge.

2 Set up the planes needed for the profile sketches. Use existing construction planes, or create new construction planes. Planes do not have to be parallel.

3 Sketch the profiles.

4 Click either **Insert, Boss, Loft** or **Insert, Cut, Loft.**

5 Select the face of the non-planar profile and the other profile(s) in order, by clicking a corresponding point on each profile.

You do not have to select the vertices precisely; the vertex closest to the selection point is used.

A preview curve connecting the selected entities is displayed.

6 Examine the preview curve, and use the **Up** or **Down** buttons if necessary to adjust the order of the profiles.

7 Click **Close Along Loft Direction, Maintain Tangency,** and **Advanced Smoothing** if needed. See **Simple Loft** on page 4-15 for more information about these options.

8 Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-23.

9 Click **OK**.
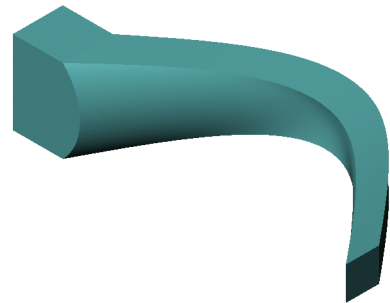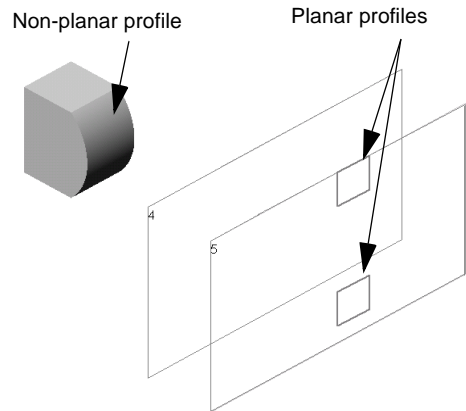
Loft with Split Line

To create a loft using a split line:

1 Use a split line to create a non-planar profile on a model face.

   See **Split Line** on page 3-11 for information about creating split lines.

2 Set up the planes needed for the profile sketches. Use existing construction planes, or create new construction planes. Planes do not have to be parallel.

3 Sketch the profiles.

4 Click either **Insert, Boss, Loft** or **Insert, Cut, Loft**.

5 Select the face created by the split line and the planar profiles.

6 Examine the preview curve, and use the **Up** or **Down** buttons if necessary to adjust the order of the profiles.

Planar profile

Face created by split line

7 Click **Close Along Loft Direction**, **Advanced Smoothing**, and **Maintain Tangency** if needed. See **Simple Loft** on page 4-15 for information about these options.

8 Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-23.

9 Click **OK**.
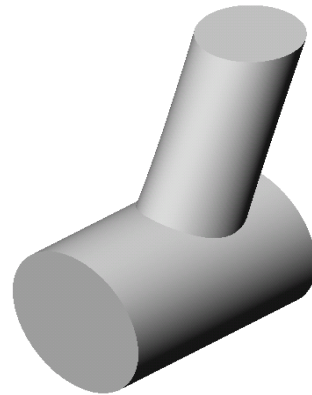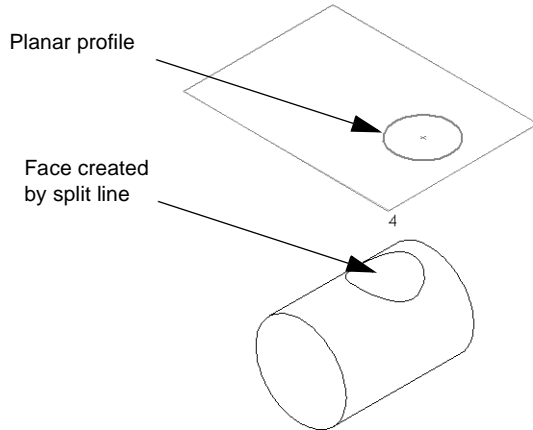
Loft with Guide Curves

You can create a *guide curve loft* by using two or more profiles and one or more guide curves to connect the profiles. Guide curves help you control the intermediate profiles that are generated.

To create a loft using guide curves and planar profiles:

1 Sketch the profiles.

2 Sketch one or more guide curves.

3 Add relations between the guide curve(s) and the profiles. You can use a combination of:

- **Pierce** relations between the guide curve(s) and vertices and/or user-defined sketch points on the profile.

- **Coincident** relations between vertices and/or user-defined sketch points of the guide curve(s) and the profiles.

These relations should be added to the profiles.

Guide curve
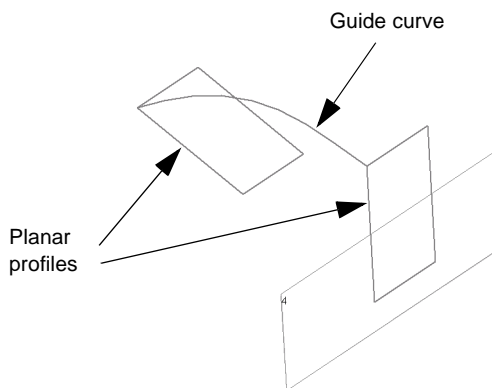
Planar profiles

4 Click one of the following:

- **Insert, Base, Loft**
- **Insert, Boss, Loft**
- **Insert, Cut, Loft**

5 Click the **Profiles** box, then select the profiles in order, in either the FeatureManager design tree or the graphics area.

6 Click the **Guide Curve(s)** box, then select the guide curve(s).

7 Examine the preview, and use the **Up** or **Down** buttons if necessary to adjust the order of the profiles.

8 Click **Close Along Loft Direction**, **Advanced Smoothing**, and **Maintain Tangency** if needed. See **Simple Loft** on page 4-15 for more information about these options.

9 Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-23.

10 Click **OK**.
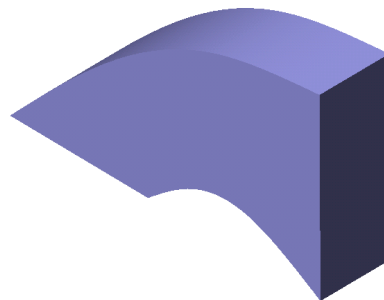
To create a loft using guide curves and non-planar profiles:

1 Use a split line to create a non-planar profile on a model face.

See **Split Line** on page 3-11 for information about creating split lines.

2 Sketch one or more guide curve(s).

3 Add a **Pierce** relation between the guide curve(s) and an edge or vertex on the non-planar face.
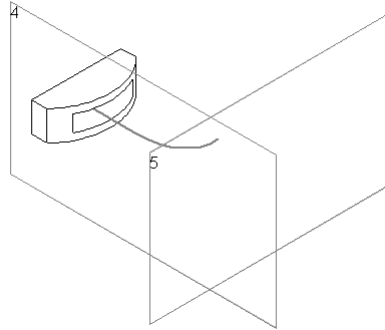
4 Sketch the remaining profiles needed to create the loft. Add **Pierce** relations between the profiles and the guide curve(s).

The relations should be added to the profile sketches.

5 Click either **Insert, Boss, Loft** or **Insert, Cut, Loft.**

6 Click the **Profiles Box**, then select the split line face and the planar profiles in order, in either the FeatureManager design tree or the graphics area.

7 Click the **Guide Curve(s)** box, then select the guide curve(s).

8 Examine the preview, and use the **Up** or **Down** buttons if necessary to adjust the order of the profiles.

9 Click **Close Along Loft Direction**, **Advanced Smoothing**, and **Maintain Tangency** if needed.

10 Click the **Advanced** tab to specify the tangency options, if desired (the default is **None**). See **Start and End Tangency Options for Lofts** on page 4-23.

11 Click **OK**.
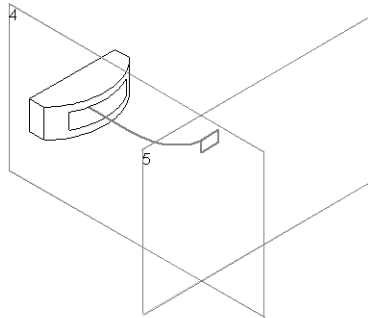
Notes on Creating Lofts with Guide Curves:

❑ Guide curves must intersect all the profiles.

❑ There is no limit to the number of guide curves you may use.

❑ Guide curves can intersect at a point.

❑ You can use any of the following items as a guide curve: sketched curves, model edges, or reference curves of any kind. See **Curve** on page 3-8 for information about creating reference curves.

❑ If a loft fails or twists, you should add a **Curve Through Reference Points** as a guide curve, selecting corresponding vertices of the profiles to create the curve. See **Curve Through Reference Points** on page 3-12.

❑ Guide curves can be longer than the resulting loft. The loft stops at the end of the smallest guide curve.

❑ You can further control the behavior of the loft by creating the same number of segments on all the guide curves. The end points of each segment mark corresponding points for transition of the profiles.

Center Line Loft

You can create a loft that uses a variation of a guide curve that acts as a center line. The sketch planes of all the intermediate sections are normal to the center line. The center line may be a sketched curve or a reference curve.

To use a center line to guide a loft:

1   Create the profiles.

2   Sketch a curve, or create a reference curve. The curve must intersect the area inside each closed profile and any split line faces.

3   Click one of the following, then select the profiles:

- **Insert, Base, Loft**
- **Insert, Boss, Loft**
- **Insert, Cut, Loft**

4   Click the **Advanced** tab, click the **Center Line** box, then click the sketch or curve in the FeatureManager design tree or the graphics area.

To preview the effect of the center line, click **Show Sections**. Use the slider to adjust the **Number of Sections** that are displayed. To view the sections in order, click the up and down arrows beside the **Section Number**.

Center Line

Preview
curve

Sketches

Center Line
OFF

Center Line
ON
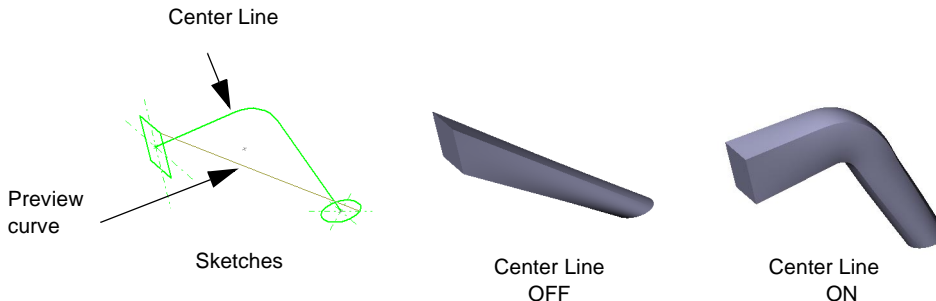
Start and End Tangency Options for Lofts

You have several options for controlling the tangency of a loft at the starting and ending profiles.

You specify the **Start Tangency** and **End Tangency** options on the **Advanced** tab of the **Loft** dialog box. Choose from these options:

❑ **None**. No tangency is applied.

❑ **Normal to Profile**. The loft is normal to the sketch plane of the profile at the start or end.

❑ **Direction Vector**. The loft is tangent to a selected linear edge or axis, or to the normal of a selected plane. Click the **Direction Vector** button, then click the edge, axis, or plane.

❑ **All Faces**. The loft is tangent to the adjacent faces of existing geometry at the start or end. This selection is only available when the loft is attached to existing geometry.

The examples in the following table are created from this base feature and these sketches.

Direction Vector

End Profile   Start Profile
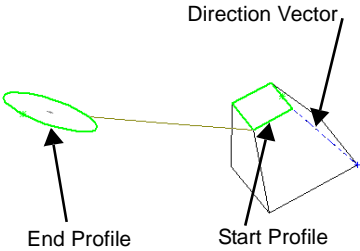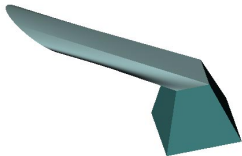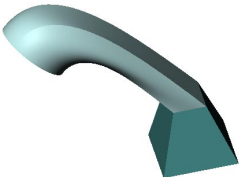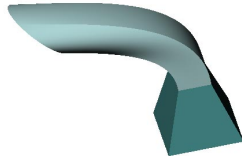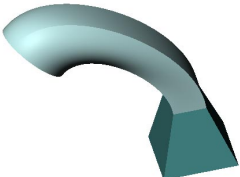
Start Tangency **None** **End Tangency** **None**		**Start Tangency** **None** **End Tangency** **Normal to Profile**	
Start Tangency **Normal to Profile** **End Tangency** **None**		**Start Tangency** **Normal to Profile** **End Tangency** **Normal to Profile**	
Start Tangency **All Faces** **End Tangency** **None**		**Start Tangency** **All Faces** **End Tangency** **Normal to Profile**	
Start Tangency **Direction Vector** **End Tangency** **None**		**Start Tangency** **Direction Vector** **End Tangency** **Normal to Profile**	
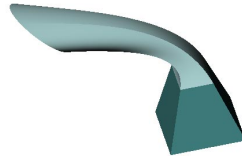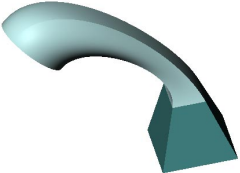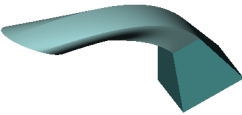
Fillet/Round

Fillet/Round creates a rounded internal or external face on the part. You can fillet all edges of a face, selected sets of faces, selected edges, or edge loops.

In general, it is best to follow these rules when making fillets:

❑ Add larger fillets before smaller ones. When several fillets converge at a vertex, create the larger fillets first.

❑ Add drafts before fillets. If you are creating a molded or cast part with many filleted edges and drafted surfaces, in most cases you should add the draft features before the fillets.

❑ Save cosmetic fillets for last. Try to add cosmetic fillets after most other geometry is in place. If you add them earlier, it takes longer to rebuild the part.

❑ To enable a part to rebuild more rapidly, use a single **Fillet** operation to treat several edges that require equal radius fillets. Be aware, however, that when you change the radius of that fillet, all the fillets created in the same operation change.

Constant Radius

To fillet multiple model edges and/or faces:

1 Click **Fillet** on the Features toolbar, or **Insert**, **Features, Fillet/Round**.

2 Select the faces and edges that you want to fillet or round.

3 Specify the fillet **Radius**.

4 Select **Fillet Type** of **Constant Radius**.

5 Verify that the correct number of edges and faces are shown in the **Items to Fillet, Edge Fillet Items** list.

Selected face

Before and after creating a fillet feature
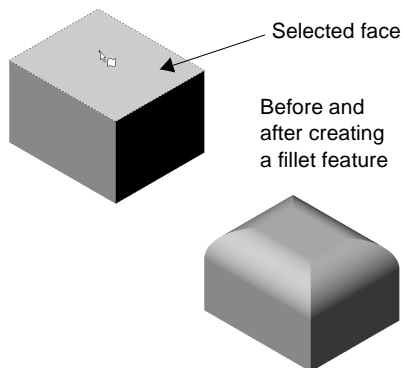
6   By default, the **Propagate to Tangent Faces** option is selected. This extends the fillet to all faces that are tangent to the selected face or edge. If you do not want the fillet to extend along tangencies, *deselect* the checkbox.

Selected edge

7   Select the **Overflow Type**. See **Overflow Type** on page 4-29 for more information.

8   Click **OK**.

Propagate to
Tangent Faces
OFF

Propagate to
Tangent Faces
ON
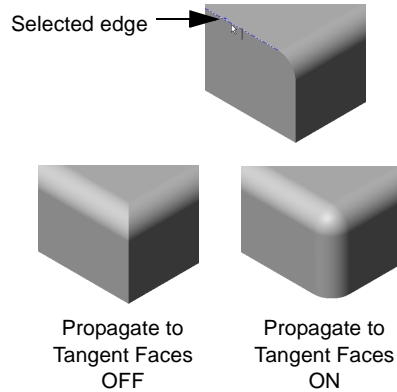
Variable Radius

You can also make variable-radius fillets. You specify a different radius for each vertex of the edge being filleted or rounded.

To insert a variable radius fillet:

1   Click **Fillet** [icon] on the Features toolbar, or **Insert**, **Features, Fillet/Round**.

2   Change the **Fillet Type** to **Variable Radius**.

3   Select the edges that you want to round.

4   Choose a transition type:

   • **Smooth transition** creates a fillet that changes smoothly from one radius to another when matching a fillet edge on an adjacent face.

   • **Straight transition** creates a fillet that changes from one radius to another more abruptly without matching edge tangency with an adjacent fillet.

5   In the **Vertex List** under **Items to Fillet**, highlight **Vertex1**. Notice that the current radius value appears next to the selected vertex.

6   Change the **Radius** to the desired value. Repeat with each vertex until you have changed all the radii as desired.

7   Click **OK**.
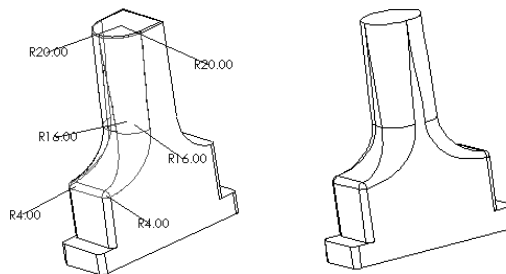
To change the radii of the fillets after they are inserted, double-click the fillet feature, then double-click and modify the dimension displayed on each vertex.

Face Blend

You can blend non-adjacent faces with a **Face Blend** fillet.

To insert a face blend fillet:

Click this face first

1 Click **Fillet** 🔲 on the Features toolbar, or **Insert**, **Features, Fillet/Round**.

2 Change the **Fillet Type** to **Face Blend.**

3 Click the first face or set of faces to be blended. Under **Items to Fillet**, the selection is indicated in **Face Set 1**.

4 Click the **Face Set 2** box, and click the face(s) to blend with.

5 Specify the **Radius**.

Blend with this face

6 If you want the blend to continue along tangencies, click **Propagate To Tangent Faces.**

7 If any of the selected faces has a tangency that results in an ambiguity, you can use a **Help Point** to resolve the ambiguous selection. Click **Use Help Point**, then click an edge or face where you want to insert the fillet.

8 Click **OK**.
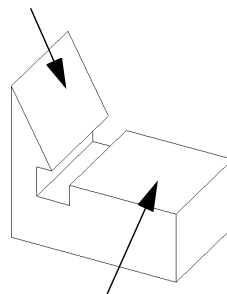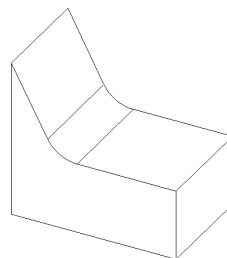
Face Blend with Hold Line

You can determine the radius and shape of a face blend fillet by setting a boundary, or hold line, for the fillet. The hold line can be an edge on the part or a split line projected on a face to be filleted.

In a fillet of this type, the radius of the fillet is driven by the distance between the hold line and the edge to be filleted.

To insert a face blend fillet using an edge:

1   Click **Fillet** 🖼️ on the Features toolbar, or **Insert**, **Features, Fillet/Round**.

2   From the **Fillet Type** box, select **Face Blend**.

3   Click the **Face Set 1** box, then click a model face (or faces) to fillet.

4   Click the **Face Set 2** box, then click the *adjacent* model face (or faces) to fillet.

5   Click the **Advanced Face Fillet** tab and check the **Set fillet boundary** checkbox.

6   Click the model edge or edges that mark the end of the fillet. The names are listed in the **Hold Lines** box.

7   Click **OK**. The faces that share an edge with the **Hold Lines** are removed by the fillet.
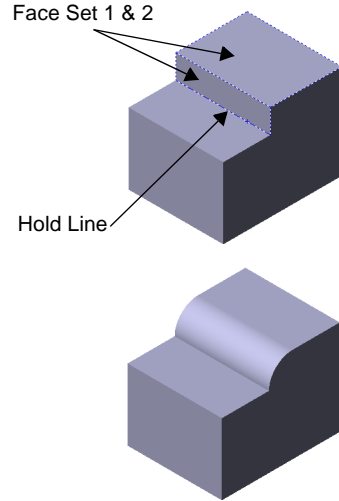
To insert a face blend fillet using a split line:

1   Create a split line on a model face. See **Split Line** on page 3-11 for information about creating a split line.

2   Click **Fillet** 🖼️ on the Features toolbar, or **Insert**, **Features, Fillet/Round**.

3   Select **Insert**, **Features, Fillet/Round**.

4   Under **Fillet Type**, select **Face Blend**.

Do not enter a radius. The radius is determined by the distance between the **Hold Line** and the edge to be filleted.

5   Click one model face to be blended. Under **Items to Fillet** the selection is indicated in **Face Set 1**.

6   Click the **Face Set 2** box, and click an *adjacent* model face.

7   Click the **Advanced Face Fillet** tab, and check the **Set fillet boundary** checkbox.

8   Click the split line. The name is listed in **Hold Lines** box.

9   Click **OK**.
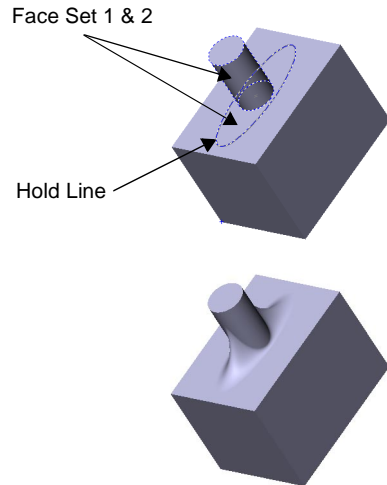
Overflow Type

One of the options on the **Fillet Feature** dialog box is **Overflow Type**. The following describes the available types and the results obtained with each type.

- **Default.** The system chooses one of the following options, depending on the geometry conditions (convexity of edges being filleted and the adjacent edges, and so on).

- **Keep Edge.** Maintains the integrity of adjacent linear edges. However, the fillet surface is broken into separate surfaces, and in many cases the top edge of the fillet may have a dip in it.

- **Keep Surface.** Uses the adjacent surface to trim the fillet. As a result, the fillet edge is continuous and smooth, but the adjacent edge is disturbed.
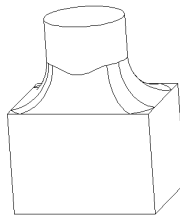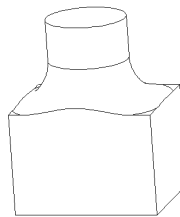
Keep Edge   Keep Surface

Chamfer

Chamfer creates a beveled edge on the selected edges and/or faces.

To create a chamfer:

1 Click **Chamfer** on the Features toolbar, or **Insert**, **Features, Chamfer**.

2 On the model, select the edges, faces, loops, or vertex to chamfer. (You can also do this as the first step.)

3 Look for an arrow on the selected edge; it indicates the direction of the chamfer. If the direction is not what you want, click **Flip Direction**.

4 Examine the **Items to Chamfer** list. If you selected an incorrect item, click it again to deselect, or right-click and select **Clear Selections**.

5 Select a chamfer type from the list, and specify the necessary parameters:

 • If you selected **Distance-Distance**, enter values for the distances on either side of the chamfer edges, or click **Equal Distance** and specify one value.

 • If you selected **Angle-Distance**, enter values for distance and angle.

 • If you selected **Vertex-Chamfer**, enter distance values for the three chamfer edges, or click **Equal Distance** and specify a single value.

6 Click **OK**.
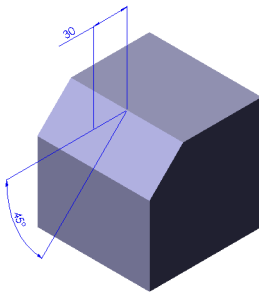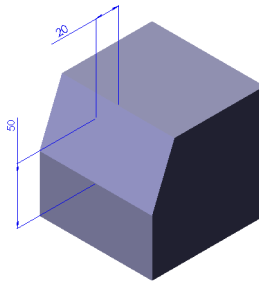
Angle-Distance   Distance-Distance   Vertex-Chamfer
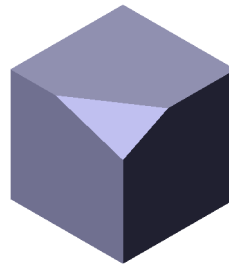
Draft ![Draft icon]

Draft tapers faces using a specified angle to selected faces in the model, to make a molded part easier to remove from the mold. You can insert a draft in an existing part or draft while extruding a feature. You can draft using either a *neutral plane* or a *parting line*.

Neutral Plane

When you draft using a neutral plane, you select a face or reference plane to serve as the neutral plane. The draft angle is measured perpendicular to this plane.

To insert a draft angle in an existing part using a neutral plane:

1 Click **Draft** ![icon] or **Insert, Features, Draft**.

2 In the **Type of Draft** box, select **Neutral Plane**, click in the **Neutral Plane** box, and select the neutral plane in the graphics area.

3 Set the **Draft Angle**. If you want the draft to slant in the opposite direction, click **Reverse Direction**.

4 Click in the **Faces to Draft** box, and select the faces to draft in the graphics area.

5 Choose the **Face Propagation** type that describes how you want the draft to propagate across additional faces.

   - **None**. Only the selected face is drafted.

   - **Along Tangent.** Extend the draft to all faces that are tangent to the selected face. (The faces meet with filleted corners.)

   - **All Faces.** Draft all faces next to the neutral plane and extruded from the neutral plane.

   - **Inner Faces.** Draft all faces extruded from the neutral plane.

   - **Outer Faces.** Draft all faces next to the neutral plane.

6 When you are satisfied with the results, click **OK**.
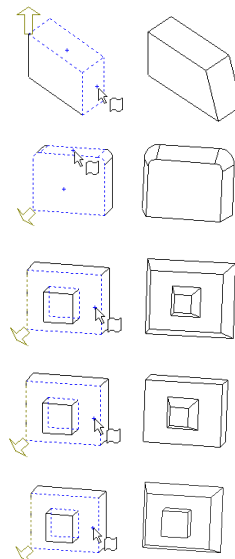
Parting Line

The parting line option lets you draft surfaces around a parting line. The parting line can be non-planar.

To draft on a parting line, you may first divide the faces to be drafted by inserting a **Split Line** (see page 3-11), or you may use an existing model edge. Then you specify the *direction of pull*, that is, the side of the parting line from which material is removed.

To insert a draft angle parting line:

1   Sketch the part to be drafted, and desired parting line(s).

2   Insert a split line curve as described in **Split Line** on page 3-11.

3   Click **Draft** 🔲 or **Insert, Features, Draft**.

4   In the **Type of Draft** box, select **Parting Line**.

5   Specify the **Draft Angle.**

6   Click the **Direction of Pull** box, and select an edge or face in the graphics area to indicate the direction of pull.

   Note the arrow direction, and click **Reverse Direction** if desired.

7   Click the **Parting Lines** box, and select the parting lines in the graphics area.

   Note the arrow direction. To specify a different draft direction for one or more segments of the parting line, click the name of the edge in the **Parting Lines** box, and click **Other Face**.

8   Choose the **Face Propagation** type:

   • **None.** Draft only the selected face.

   • **Along Tangent.** Extend the draft to all faces that are tangent to the selected face.

9   When you are satisfied with the results, click **OK.**

Other Face selected for this edge
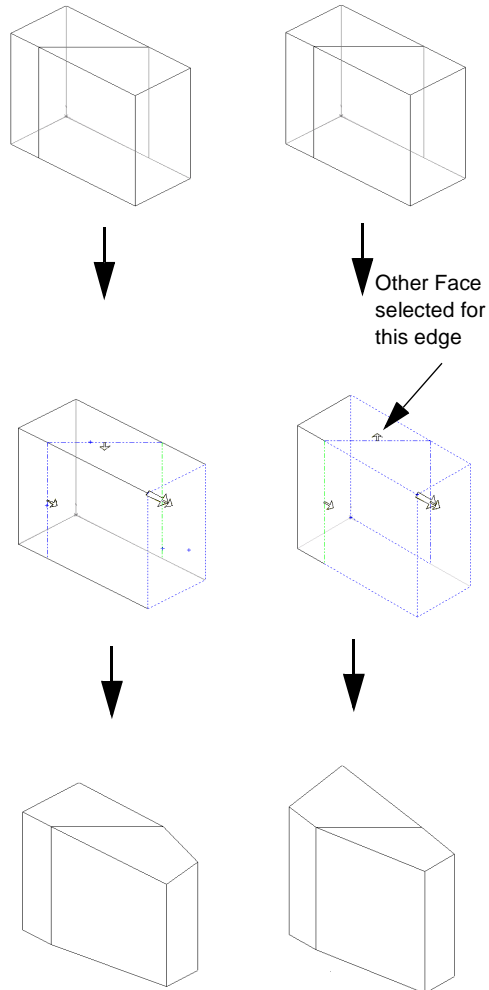
To insert a draft angle using parting line with step draft:

1 Sketch the part to be drafted.

2 Create a reference plane.

3 Create desired parting line. Make sure that:

- At least one parting line segment is coincident to the reference plane on each draft face.

- All other parting line segments are in the pull direction of the reference plane.

- None of the parting line segments are perpendicular to the reference plane.

    Insert a split line curve as described in **Split Line** on page 3-11.
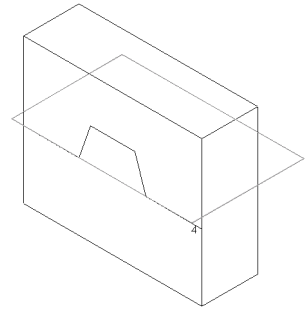
4 Click **Draft** 🔲 or **Insert, Features, Draft**.

5 In the **Type of Draft** box, select **Parting Line**.

6 Click **Step Draft**.

7 Specify the **Draft Angle**.

8 Click the **Direction of Pull** box, and select the reference plane to indicate the direction of pull.

    Note the arrow direction, and click **Reverse Direction** if desired.

9 Click the **Parting Lines** box, and select the parting line segments in the graphics area.

    Note the arrow direction. To specify a different draft direction for one or more segments of the parting line, click the name of the edge in the **Parting Lines** box, and click **Other Face**.
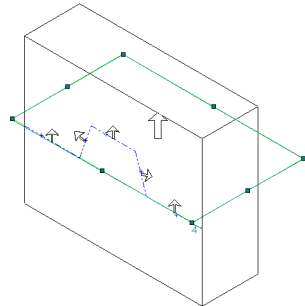
10 Choose the **Face Propagation** type:

- **None.** Draft only the selected face.

- **Along Tangent.** Extend the draft to all faces that are tangent to the selected face (faces that meet with fillets or rounds.)

11 When you are satisfied with the results, click **OK.**
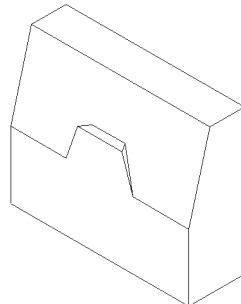
Hole 🔲 🔲

Hole creates various types of hole features in the model. You place a hole on a planar face, then specify its location by dimensioning it afterwards.

❑ **Simple** 🔲 - Places a circular hole of the depth you specify.

❑ **Wizard** 🔲 - Creates holes with complex profiles, such as **Counterbore** or **Countersunk**.

In general, it is best to create holes near the end of the design process. This helps you avoid inadvertently adding material inside an existing hole. Also, if creating a simple hole, use **Simple Hole**; do not use **Hole Wizard**. **Simple Hole** provides better performance than **Hole Wizard** for simple holes.

To insert a* simple *hole:

1 Select a planar face on which to create the hole.

2 Click **Insert, Features, Hole, Simple** or click **Simple Hole** 🔲 on the Features toolbar.

3 Select the **Type**, and specify the **Depth** or **Offset**, if necessary. For information about selecting the **Type**, see **Specifying End Conditions** on page 4-4.

4 Specify the **Diameter** of the hole.

5 If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area. The selection is indicated in the **Selected Items** box.

6 Examine the preview and click **Reverse Direction**, if necessary.

7 To add a draft, click **Draft While Extruding**. Enter a draft **Angle** and click **Draft Outward** if necessary.

8 Click **OK**.

9 To position the hole:

   a) Right-click the hole feature in the model or the FeatureManager design tree and select **Edit Sketch**.

   b) Add dimensions to position the hole. You can also modify the hole diameter in the sketch.

   c) Exit the sketch or click **Rebuild**.
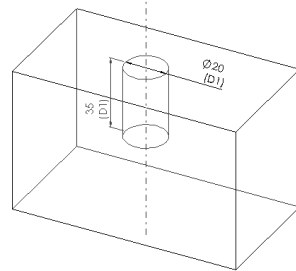
To change the diameter, depth, or type of the hole, right-click the hole feature in the model or the FeatureManager design tree, and select **Edit Definition**. Make the necessary changes, and click **OK**.

To insert a hole using the **Hole Wizard:**

1 Select a planar face on which to create the hole.

> **NOTE:** Do not select a reference plane; select a planar face.

2 Click **Insert, Features, Hole, Wizard** or click **Hole Wizard** 🔲 on the Features toolbar.

3 Select the **Hole type** from the pull-down list (countersunk, counterbore, and so on).

4 In the **Section dimensions** area, click-pause-click a dimension value to select it. Enter the new dimension.

5 Select the **End Condition Type** from the pull-down list.

6 If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area.

7 Click **Next**. The **Hole Placement** dialog appears.

8 Drag the centerpoint of the hole, or add dimensions to place the hole at the desired location. You can use the dimension tool 📐 while the **Hole Placement** dialog is displayed.

9 Click **Finish**.
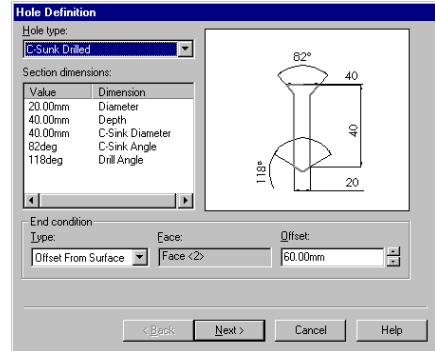
Shell

Shell hollows out the part, leaving open the faces you select, and thin walls on the remaining faces. If you plan on adding fillets to the part, you should fillet the part before you shell it.

To insert a shell:

1   Click **Shell** on the Features toolbar, or **Insert**, **Features, Shell.**

2   In the model, click on the face(s) from which you want to remove material.

   The faces are listed in the **Faces to Remove** box.

3   Specify the wall **Thickness**.

4   Click **Shell Outward** if you want the shell thickness added to the outside.

5   Click **OK**.
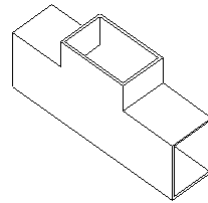
To set a different shell thickness for each face:

1   Select **Shell** or **Insert**, **Features, Shell**.

2   Click on the face(s) you want removed.

3   Click the **Multi Thickness Faces** box.

4   Click on the walls to which you want to apply different thicknesses.

   The faces are listed in the **Multi Thickness Faces** box.

5   Click each face in the **Multi Thickness Faces** box, and enter a **Thickness** value.

6   Choose whether to offset the shell inward or outward.

7   Click **OK**.

MultiThickness Faces
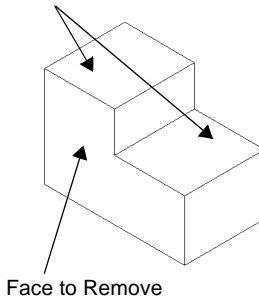
Face to Remove

Rib ![icon]
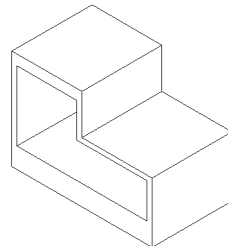
Rib is a special type of extruded feature created from an open sketched contour. It adds material of a specified thickness in a specified direction between the contour and an existing part.

To insert a rib:

1   Using a planar surface that intersects the part, sketch the contour to be used for the rib.

2   Click **Rib** ![icon] on the Features toolbar, or **Insert, Features, Rib**.

3   Select **Mid Plane** to extrude the rib equally in both directions from the sketch plane, or **Single Side** to extrude in one direction.

   If you chose **Single Side**, examine the preview and select **Reverse** if necessary.

   NOTE: For better visibility, use **Isometric** and either **Hidden Lines Removed** or **Hidden in Gray** view.
   ---

4   Enter the **Thickness** of the rib and click **Next.**

5   Note the direction of the arrow in the preview. If necessary select **Flip Side of material** to reverse the direction in which material is added.

6   To add draft, select **Enable Draft** and enter the **Angle**. Select **Draft Outward** if necessary.

7   Click **Finish**.
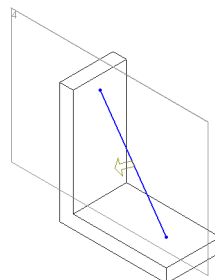
To insert a rib with multiple drafts:

1   Sketch the part and contour lines of the rib.

2   Click **Rib** ![icon] on the Features toolbar, or **Insert, Features, Rib**.

3   Select the **Type**, specify the **Thickness**, and click **Next.**

4   Select **Enable Draft** and enter the draft angle. Select **Draft Outward** if necessary.

5   To select the neutral plane for the draft, click **Next Reference** until the contour that represents the neutral plane is indicated. An arrow cycles around the edges to show you which edge is selected.
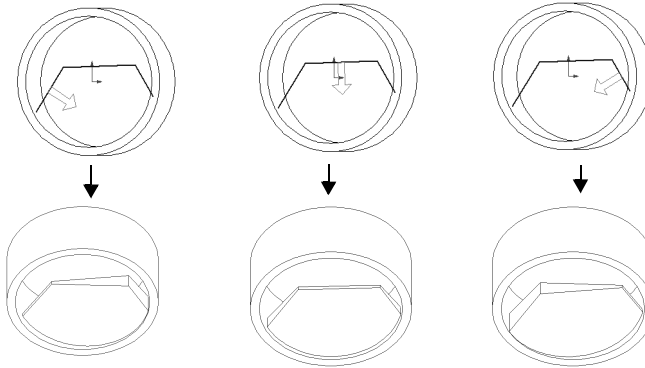
6   Click **Finish**.

Dome 🟢

You can add a *dome* feature to any planar model face.

To create a dome on a planar face:

1   Click **Dome** 🟢 on the Features toolbar, or **Insert, Features, Dome**.

2   Select a planar **Dome Face** in the graphics area.

3   Specify the **Height**, and observe the preview. The height is measured from the centroid of the selected face.

4   Click **Reverse Direction** to create a concave dome (default is convex).

5   If a circular or elliptical face is selected, you can click **Elliptical Dome**. This creates a dome whose shape is a half ellipsoid, with a height equal to one of the ellipsoid radii.

A dome on a circular face may be larger in diameter than the selected face.

6   Click **OK**.
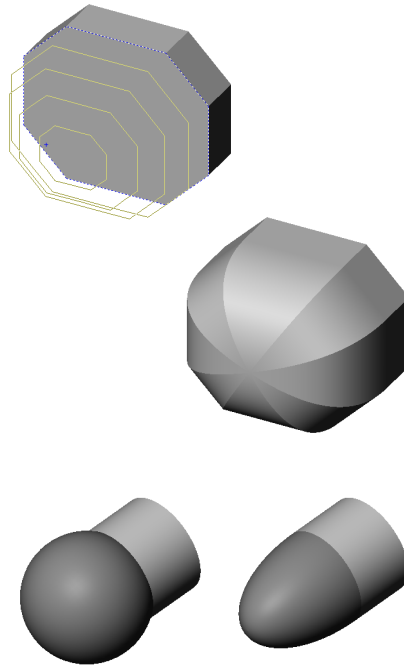
Elliptical Dome OFF   Elliptical Dome ON

Shape ![icon]

Shape creates a deformed surface from a face on a model. A deformed surface is flexible, much like a membrane. It can be expanded, constrained, and tightened using the sliders on the **Shape Feature Controls** dialog box.

If the **Shape** icon does not appear on your toolbar by default, you can add it. Click **View, Toolbars, Customize**, and follow the instructions on the **Commands** tab.

To create a deformed surface:

1 Create a model with a face to shape.

2 Sketch one or more entities with which to constrain the shape. You can create the constraining entities directly on the face or on a construction plane. Valid entities are as follows:

- Points (sketch points, endpoints, vertices, etc.)
- Sketches
- Edges
- Reference curves

3 Click **Shape** ![icon] or **Insert, Features, Shape**.

The **Shape Feature** dialog box appears.

4 Select the face to shape; you can only select one face.

5 Click the **Constrain to** box and select the entities you created to constrain the shape.

6 If desired, deselect **Maintain Boundary Tangents.**

**Maintain
Boundary
Tangents**
deselected
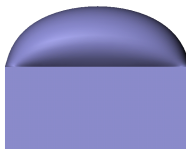
**Maintain
Boundary
Tangents**
selected
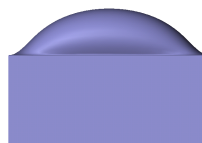
7 Click **Preview**. Rotate the part and examine the shape from different angles.

> **NOTE:** If the selected face does not change shape, use the sliders on the **Controls** tab to adjust the shape.

8   Click the **Controls** tab to modify the shape.

Adjust the sliders and observe the preview. *Experiment* until you are satisfied with the results.

- **Pressure.** Deflates or inflates the shape.

- **Curve influence.** Moves the shape away from or toward the constraining entities. Note that the **Curve influence** control is grayed out if **Constrain to** is not selected on the **Shape Feature** tab.

  The following four illustrations are derived from this base feature and sketch, and show how you can create different shapes by adjusting the sliders.
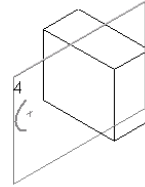
Pressure
decreased
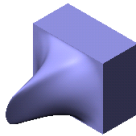
Pressure
increased
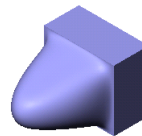
Curve influence
decreased
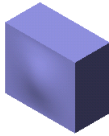
Curve influence
increased
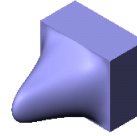
- **Stretch.** Adjusts the degree of stretching.

- **Bend**. Adjusts the degree of bending.

  The following four illustrations show the effects of adjusting the **Stretch** and **Bend** sliders. All four illustrations are derived from this base feature, which was shaped by increasing **Pressure** on the front face.

Stretch decreased	**Stretch** increased
Bend decreased	**Bend** increased
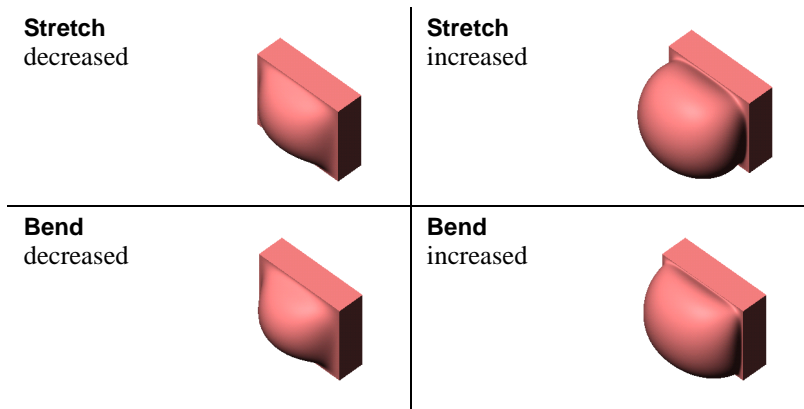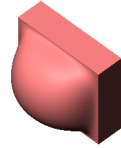
- **Resolution.** Adjusts the precision of the shape by changing how many points on the face are deformed.

  Higher **Resolution** often smoothens the flat areas and sharpens the shape near the constraining entities. Lower **Resolution** provides better performance and a coarser shape; however, some details may be omitted.

Resolution decreased   **Resolution** increased
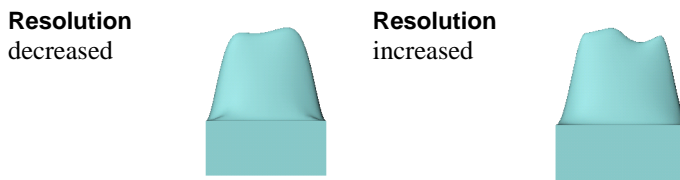
9 When you are satisfied with the shape of the deformed surface, click **OK**.

To change the shape of a deformed surface, right-mouse click the **Shape Feature** in the FeatureManager design tree and select **Edit Definition**. Then repeat Steps 8 and 9.

Pattern/Mirror

Pattern repeats the selected feature(s) in a linear or circular array.

Mirror copies the selected feature(s) or all features, mirroring them about the selected plane or face.

❑ For a **Linear Pattern**, you select the feature(s), then specify the direction, the linear spacing, and the total number of instances.

❑ For a **Circular Pattern**, you select the feature(s) and an edge or axis as the center of rotation, then specify the angular spacing and the total number of instances.

❑ For a **Mirror Feature**, you select the feature(s) to copy and a plane about which to mirror them.

❑ To **Mirror All**, you select a planar face on the model, then mirror the entire model (base and all other features) about the selected face.

You can also create patterns of patterns, and mirrored copies of patterns.

For information about using a pattern of components in an assembly, see **Adding a Component Pattern** on page 7-12.

Linear Pattern

You can use a linear pattern to quickly create multiple copies of a feature or features in one or more directions.

To create a linear pattern:

1   Create a base part and on the base part, create one or more cut, hole, or boss feature(s) that you want to repeat.

2   Select **Insert**, **Pattern/Mirror**, **Linear Pattern**.

3   With the **Items to copy** box selected, click the feature(s) to pattern.

4   Click a model edge or dimension to indicate the pattern direction or driving dimension for the **First Direction**. Notice the direction arrow on the model.

5   Click **Reverse Direction** if the arrow points in the wrong direction.

6   Specify the values for **Spacing** and **Total Instances** of the pattern.
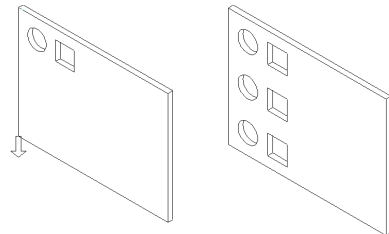
   Spacing is the distance from a point on one feature to the corresponding point on the adjacent copy (*not* edge-to-edge spacing).

   Total Instances includes the original feature(s) on which the pattern is based.

   As you modify the values, a preview of the resulting pattern is displayed.

7 Click **Vary Sketch** if you want the pattern to change as it is repeated. See **Controlling and Modifying Patterns** on page 4-47 for more information about using **Vary Sketch**.

8 Click **Geometry Pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-47.

9 Click **OK** to create the pattern in one direction, or continue with Step 10 to create the pattern in two directions.

10 Select **Second Direction** in the scroll box, and click a different model edge to indicate this direction. Again, you may need to click **Reverse Direction**.

11 Set the values for **Spacing** and **Total Instances** of the pattern in the **Second Direction** and examine the preview.

12 Click **OK**.
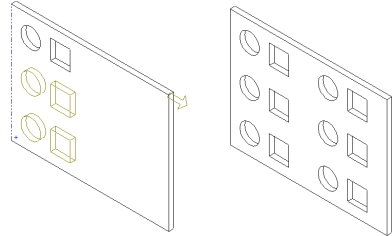
Circular Pattern

You use a circular pattern to create multiple copies of a feature or features in rotation about an axis.

To create a circular pattern:

1 Create a base part and on the base part, create one or more cut, hole, or boss feature(s) that you want to repeat.

2 Create an axis or use an existing linear edge or axis around which to pattern the feature.

> **NOTE:** For any circular object in SolidWorks, there is a temporary axis. If axes are not displayed, click **View, Axes** (for user-defined axes) or **View, Temporary Axes** (for axes created implicitly by the model).

3 Click **Insert, Pattern/Mirror, Circular Pattern**.

4 With the **Items to copy** box selected, click the feature(s) to pattern.

5 Select an axis or angular dimension to define the pattern.

6 Specify the **Spacing** in degrees and the **Total Instances** of the feature. (**Total Instances** includes the original feature(s) on which the pattern is based.)

7 Click **Reverse** if you want to create the pattern in a counter-clockwise direction. Otherwise, the pattern goes in a clockwise direction.

As you modify the values, a preview of the resulting pattern is displayed.
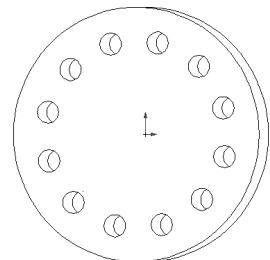
8 Click **Vary Sketch** if you want the pattern to change as it is repeated. (Refer to **Controlling and Modifying Patterns** on page 4-47 for more information on using **Vary Sketch**.)

9 Click **Geometry Pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-47.

10 Click **OK**.

You can use an equation to calculate the spacing angle used in a circular pattern.

To use an equation to control a circular pattern:

1 Create a circular pattern as described in **Circular Pattern**, above. Leave the default values in the **Spacing** and **Total Instances** boxes. It is not important what numbers you use at this time.

2 In the FeatureManager design tree, double-click the circular pattern feature. Two values appear in the model: **Total Instances** and **Spacing** angle.

3 Click **Tools, Equations**, then click **Add** in the **Equations** dialog box.

4 Click the **Spacing** angle dimension in the model. The dimension name is entered in the **New Equation** dialog box.

5 Using the calculator keypad in the dialog box, click the equals sign (=) and click **360 /**.

6 Click the **Total Instances** value in the model. Its full name is added to the end of the equation.

7 Click **OK** to complete the equation, and click **OK** again to close the **Equations** dialog box. Note the **Equations** folder Σ in the FeatureManager design tree.

If you change the number of **Instances** in the pattern, the **Spacing** angle is re-calculated, so that the instances are always regularly spaced around the center.

8 Click **Rebuild** $\boxed{8}$, or click **Edit, Rebuild**.

For more information about using equations, see **Equations** on page 5-6.
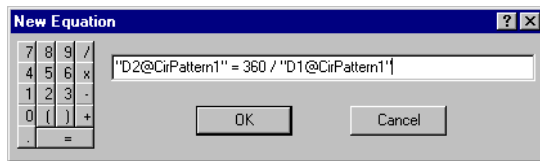
Mirror Feature

Mirror Feature creates a copy of a feature (or features), mirrored about a plane. You can either use an existing plane or create a new one. If you modify the original feature, the mirrored copy is updated to reflect the changes.

To mirror a feature (or features):

1   Click **Insert, Pattern/Mirror, Mirror Feature**.

2   With the **Mirror Plane** box selected, click on a plane or face.

3   With the **Items to copy** box selected, click a feature or features in the model or in the FeatureManager design tree.

   A preview of the mirrored feature(s) is displayed.

4   Click **Geometry Pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-47.

5   Click **OK**.

> **NOTE:** You can also mirror a feature about a face that is perpendicular to the sketch plane. Mirroring will be about the selected face, not about the feature itself.
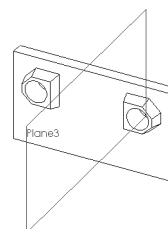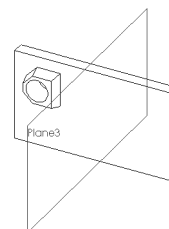
Mirror Pattern

To create a mirrored copy of a pattern:

1   Click **Insert, Pattern/Mirror, Mirror Feature**.

2   With the **Mirror Plane** box selected, select a plane about which to mirror the pattern.

3   With the **Items to copy** box selected, click the *pattern* to be mirrored. You only need to select *one instance* of one element of the pattern in the model.

   A preview of the mirrored pattern is displayed.

4   Click **Geometry Pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-47.

5   Click **OK**.

Mirror All

To create a part that is symmetrical about a planar face, you can use **Mirror All.** You build one half of the part, then mirror the entire model all at once. Any changes you make to the original half are reflected in the other half.

To mirror a part around a planar face:

1 Establish a plane corresponding to the plane of symmetry of the complete part. This can be an existing plane or one you create.

2 Create the features for one half of the part on one side of this plane.

3 Select the face of the part half on the plane of symmetry.

4 Click **Insert, Pattern/Mirror, Mirror All.**

A mirror image of the original part half is joined to it at the selected face to make a complete, symmetrical part.
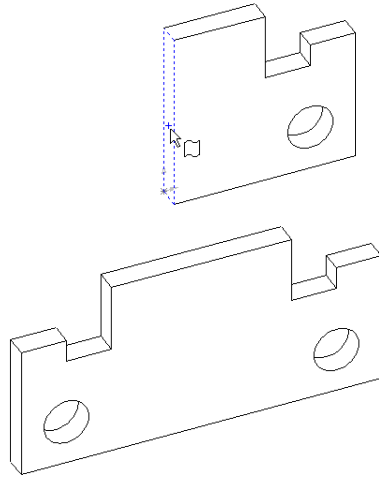
Pattern of Patterns

To create a pattern of patterns:

1 Select **Insert**, **Pattern/Mirror**, then either **Linear** or **Circular** pattern.

2 Select the pattern feature in the FeatureManager design tree, or select a face on a patterned feature in the model. You only need to select one instance of one element of the pattern in the model.

3 Proceed as described in **Linear Pattern** on page 4-42 or **Circular Pattern** on page 4-43.

Controlling and Modifying Patterns

You can control and modify feature patterns in the following ways:

❑ Use the **Geometry Pattern** option to make exact mirrored/patterned copies of features, and to improve system performance when using mirrored/patterned features.

❑ Use the **Vary Sketch** option to adjust the profile of the patterned feature based on its dimensions and relations to other features.

❑ Use a mathematical **Equation** to calculate values that define the pattern.

❑ Use **Delete Pattern Instances** to remove selected instances in a pattern.

Geometry Pattern

You can make mirrored or patterned copies of features using the **Geometry Pattern** option. **Geometry Pattern** (a checkbox on the **Mirror/Pattern** dialog box) copies the faces and edges of the geometry being mirrored or patterned. Any references to other model geometry (such as **Up to Surface**, **Offset from Surface**) used to create the original feature are ignored (not solved).

The **Geometry Pattern** option also improves system performance when using complex mirrored or patterned features.

In the example below, the cut is extruded using the **Offset from Surface** end condition.When the **Geometry Pattern** option is *off*, the copies retain this design parameter. The end of the cut is offset the same distance from the selected face in each instance.

When the **Geometry Pattern** option is *on*, the copies are exact copies of the faces and edges of the original feature. The end condition is ignored, and the distance from the end of the cut to the selected face is not recalculated.

Item to Copy
Offset From Surface
End Condition

Geometry Pattern OFF

Geometry Pattern ON
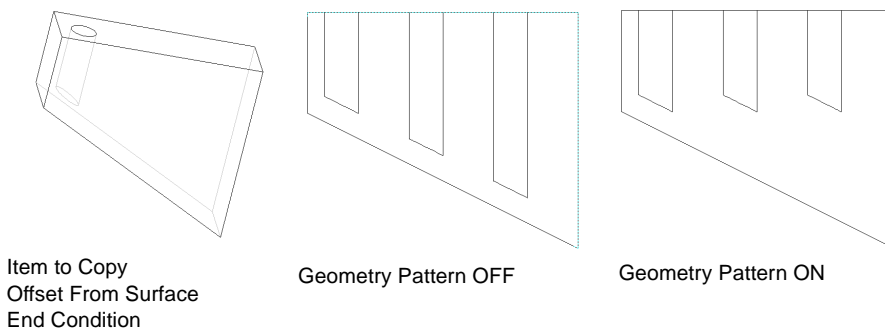
> **NOTE:** You cannot use the **Geometry Pattern** option with any feature that has a face in common with another face on the model.
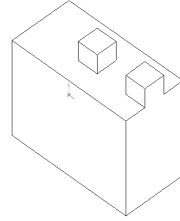
Vary Sketch

Use the **Vary Sketch** option if you want the pattern to change its dimensions as it is repeated. For example, you may want to maintain a specific distance between the edges of the base part and the patterned features.

In this example, the first feature (the cut feature) is patterned three times.

- When **Vary Sketch** is *unchecked*, the pattern remains the same regardless of the defining geometry.

- When **Vary Sketch** is *checked*, the pattern maintains its relationship to the sloping edge, based on the dimensions and constraints of the first instance of the pattern.

Linear pattern *without* Vary Sketch

first feature

Linear pattern *with* Vary Sketch
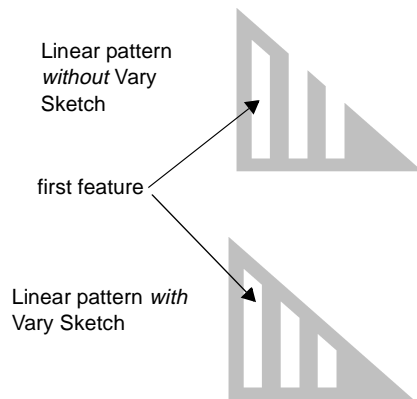
To create a variable pattern:

1 Create a sketch for the first feature in the pattern, observing the following recommendations:

- The feature sketch must be constrained to the boundary that defines the variation of the pattern instances. For example, in the illustrated pattern, the angled top edge of the first feature is parallel and dimensioned to the angled edge on the base part.

- The feature sketch should be *fully defined*.

2 Double-click one of the feature dimensions and click the spinbox arrow to change the dimension.

This gives you a preview of the way this dimension will drive the feature when you create the pattern. Try a different dimension for a different result.
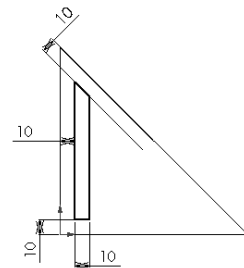
3 When you are satisfied that the dimensions are correct and the pattern will repeat as you wish, click **Insert, Cut, Extrude** (or **Insert, Boss, Extrude**) to create the first feature.

4 In the FeatureManager design tree, select the feature to repeat.

5 Click **Insert**, **Pattern/Mirror**, **Linear Pattern**.

6 Click the dimension that you want to use to drive the pattern. Notice the preview arrow.

7 Click the **Vary Sketch** checkbox.

8 Specify the values for the **Spacing** and **Total Instances** of the pattern.

9 Click **OK** to create the pattern.
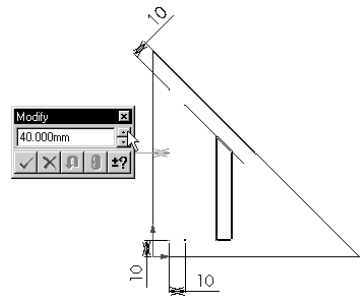
Delete Instance

To delete an instance of a feature from a pattern, or an entire pattern:

1 Select the pattern instance(s) that you want to delete.

2 Press the **Delete** key.

3 In the dialog box, select one of the following:

- **Delete Pattern Instances**. Only the selected instance(s) are deleted. The instance locations are displayed in the **Instances Deleted** box.

- **Delete Pattern Feature**. The complete pattern is deleted, not including the initial feature.

4 Click **OK** to delete, or click **Cancel** to exit the dialog.

To restore deleted pattern instances:

1 Right-click the pattern feature, and select **Edit Definition**.

2 In the **Instances Deleted** box, select the number corresponding to the deleted instance to restore, and press the **Delete** key.

5

Working with Parts

The 3D part is the basic building block of the SolidWorks mechanical design software. This chapter describes parts and some ways to work with them, including:

❑ Editing, moving, and copying features

❑ Using equations

❑ Dependency editing using Rollback, Suppress and Unsuppress

❑ Working with part configurations

❑ Using a design table

❑ Displaying a section view of the model

❑ Annotating parts

❑ Specifying lighting characteristics and material properties

Feature Editing

In a part document, you can perform the following types of feature editing operations:

❑ Edit the definition, the sketch, or the properties of a feature.

❑ View the parent/child relationships of a feature.

❑ Use the feature handles to move and copy features.

❑ Control the access to selected dimensions.

❑ Change the order in which features are reconstructed when the part is rebuilt.
See **Drag and Drop Features** on page 1-22.

❑ Roll back the part to the state it was in before a selected feature was added.
See **Rollback** on page 5-12.

To perform some of these operations, you use the icons in FeatureManager design tree, where the features are listed in the order in which they are created. For more information, see **The FeatureManager Design Tree** on page 1-19.

Edit Definition

You can edit the definition of a feature to change its parameters. For example, you can edit the depth of an extruded feature, or the selected edges for a fillet.

To edit the definition of a feature:

1 Click a feature either in the graphics area or in the FeatureManager design tree, then click **Edit, Definition,** or right-mouse click the feature and select **Edit Definition**. Depending on the selected feature type, the appropriate dialog box appears.

2 Edit the definition in the dialog box by specifying new values or options.

3 Click **Apply** or **OK** to accept the changes; click **Cancel** to discard the changes.

Edit Sketch

For features that are based on sketches, you can edit the sketches to change the geometry of the features. See Chapter 2, "Sketching with SolidWorks."

Feature Properties

Feature properties include name, color and suppression state.

To edit feature properties:

1 Select one or more features, either in the graphics area or in the FeatureManager design tree, then click **Edit, Properties,** or right-mouse click the feature and select **Properties**.

To select multiple features, hold the **Ctrl** key while selecting. If you select multiple features in the FeatureManager design tree, move the cursor to the graphics area to select **Properties** from the right-mouse menu. Otherwise, only the last item remains selected.

The **Feature Properties** dialog box appears.

2 Enter a new **Name** if desired.

3 To suppress the feature, click **Suppressed**.

Suppressing a feature removes it from the model temporarily, without deleting it. This can be used to simplify the model, or to create part configurations. See **Suppressing Features** on page 5-10 and **Working with Part Configurations** on page 5-15.

4 To change the color, click **Color**. In the **Entity Property** dialog, click the **Change Color** button. Select a color from the color palette or define a custom color.

Click **Advanced** to change lighting properties such as shininess and transparency.

To restore the feature to the color of the part, click **Use Part Color**.

5 Click **OK**.

To change the color of a feature face:

1 Right-mouse click one or more feature faces and select **Face Properties.**

2 In the **Entity Property** dialog, click the **Change Color** button. Select a color from the color palette or define a custom color.

3 Click **Advanced** to change lighting properties such as shininess and transparency.

4 To restore the face to the color of the feature it belongs to, click **Use Part Color**. (Face colors override colors specified for the *feature*.)

5 Click **OK**.

Parent/Child Relationships

When features are built upon other features, their existence depends upon the existence of the previously built feature. The new feature is called a *child* feature. For example, a hole is the child of the feature in which it is cut.

A *parent* feature is an existing feature upon which others depend. For example, a boss is the parent feature to a fillet that rounds its edges.

To view Parent/Child relationships:

1 In the FeatureManager design tree or in the graphics area, right-mouse click the feature whose relationships you want to see.

2 Select **Parent/Child** from the menu. You can only *view* the relationships; you cannot edit them.
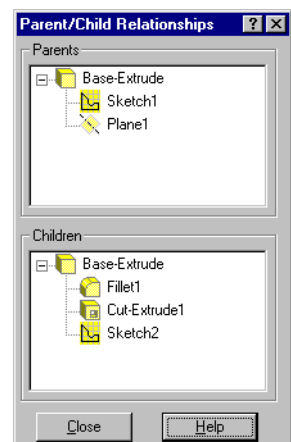
Feature Handles

Extruded and revolved features, and domes have handles that allow you to move, rotate, and resize the features quickly.

To display and use the feature handles:

1 Click the **Move/size feature** tool ⬚ on the Features toolbar. (If this button is not displayed on your toolbar, click **View, Toolbars, Customize**, and follow the instructions on the **Commands** tab to add it to a toolbar.) This button stays enabled while you continue working in the part.

2 Click the feature you want to work with, either in the FeatureManager design tree or in the graphics area. Double-click the feature to display both the handles and the dimensions of the feature.

3 To rotate or resize a feature, drag the appropriate handle. (Domes have only the resize handle.)

4 To move a feature, you can either:

- Drag the feature to a new location. If the feature has locating dimensions or relations that restrict its movement, you are notified that the dimensions will be recalculated, and relations may be deleted. You have the choice to continue or not.

- Hold the **Alt** key as you drag to maintain the dimensions and relations on the feature. The feature is allowed to move only within unspecified degrees of freedom.

5 When you are finished, click ⬚ again to disable the handle display.
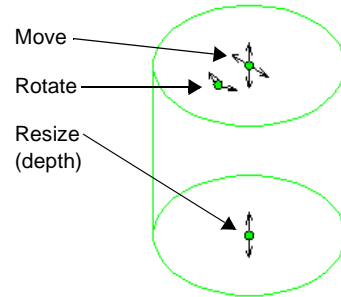
With the **Move/size feature** tool ⬚ disabled, you can still move a feature, by holding the **Shift** key and dragging one of its faces. The behavior is the same as using the move handles, and you can use the **Alt** key the same way.

When you *drag a feature with the move handle*, any child feature moves with its parent. When you **Shift**-drag, an attempt is made to maintain the relative position of any child feature, based on the child's dimensions and relations. However, in some cases (for example, when the child feature is underdefined, or dimensioned to something other than its parent), the child may not move with the parent as expected.

To move more than one feature at a time, press the **Ctrl** key as you select the features. This is useful in the case of underdefined child features.

Copying Features

To copy a feature:

1 Press the **Ctrl** key, then drag and drop the feature where needed. You can drop the feature on the same face, or on a different one. To copy more than one feature at a time, press the **Ctrl** key as you select the features.

Locating dimensions are left dangling, and the feature and sketch are marked with a red exclamation point indicating rebuild errors.

2   Either delete or re-attach the dangling dimensions. Edit the sketch, select the dangling dimension, and drag the red handle to a new attachment point. As you drag, the ⊘ cursor is displayed until you are on an edge or vertex that is suitable to re-attach the dimension.

To copy features from one part to another part:

Tile the windows, then drag and drop features from one window to another, or use the **Copy** 🗐 and **Paste** 🗐 tools on the Standard toolbar.

Controlling Access to Dimensions

For some types of parts, such as sheet metal forming tools and library feature parts, you may want to prevent other users from changing certain dimensions.

When you drag and drop one of these items into a part document (either from the Feature Palette window, or from Windows Explorer, or from a hyperlink in Internet Explorer), the dimensions are displayed and they are available for editing. You can control the access to each dimension, to prevent other users from making unwanted changes when they use these items. See **The Feature Palette Window** on page 10-6 for more information.

You can also use this function to reduce the clutter of unnecessary dimensions in the display.

To control dimension access:

1   Right-mouse click the part icon at the top of the FeatureManager design tree, and select **Edit Dimension Access**.

In the **Dimension Access** dialog box, there are two lists: **User Dimensions** (accessible) and **Internal Dimensions** (inaccessible).

2   Specify the dimensions for each list:

• To move a dimension from one list to the other, click its name, then click the left or right arrow button. As you click each name, the dimension is displayed in the model.

• To move *all* the dimensions at once, click the double arrow button.

3   Click **Finish**.

When you drag and drop an item that has restricted dimension access, only the **User Dimensions** are displayed and available for editing. The **Internal Dimensions** are not displayed, and they are not available for editing.

Equations

You can use equations to define mathematical relations between parameters or dimensions. Equations can include sketch dimensions and feature parameters (such as the depth of an extruded feature, or the instance count in a pattern). Equations can also be used to relate the dimensions of different parts in an assembly, or the values (distances or angles) used in mates.

To add an equation:

1 Click **Tools, Equations**, and in the **Equations** dialog box, click **Add**.

2 Enter the equation by clicking the dimensions or parameters in the model, and the numbers and operators in the calculator keypad, either in the **New Equation** dialog box, or on the keyboard.

Equations are solved left to right (the expression on the left is driven by the expression on the right), *in the order in which they appear in the list.* Therefore, the order in which you enter equations can affect the resulting part.

3 Click **OK**. The equation is listed in the **Equations** dialog box, and the resulting value is displayed in the **Evaluates To** column.

4 If you want to add another equation, click **Add** again; otherwise, click **OK** to close the **Equations** dialog box. Notice the **Equations** folder ⬛ in the FeatureManager design tree.

5 Click **Rebuild** 🔳 to apply any changes caused by solving the equation.

To add more equations:

1 Click **Tools, Equations, Add,** or right-mouse click the **Equations** folder ⬛, and select **Add Equation**.

2 Repeat Steps 2 through 5 above.

To edit an equation:

1 Click **Tools, Equations, Edit All,** or right-mouse click the **Equations** folder ⬛, and select **Edit Equation**.

2 In the **Edit Equations** dialog box, edit the text of the equations as needed, and click **OK**. The revised equation is displayed in the **Equations** dialog box.

3 Click **OK**, and **Rebuild** 🔳 the model.

To delete an equation:

1 Click **Tools, Equations,** or right-mouse click the **Equations** folder ⬛, and select **Delete Equation**.

2 Click the equation number or the status marker (see the next section) for the equation to delete, and click the **Delete** button in the **Equations** dialog box.

3 Click **OK**, and **Rebuild** 🔳 the model.

Equation Status

When you delete a feature or a dimension that is used in an equation, the system asks you if you want to delete the equation also. Because the order of solving equations is important, you may prefer to keep the equation, and edit it later. (You can cut, copy and paste equations in the **Edit Equations** dialog box, if necessary, to modify the order.)

If you choose to keep the equation, it has **Not Solved** status, and is marked with a red exclamation point ❗ in the **Equations** dialog box. Equations with **Solved** status are marked with green checkmark ✔.

Comments in Equations

You can add comments to equations, to provide documentation of your design intent. You can also use comment syntax to prevent an equation from being evaluated.

To add a comment to an equation:

In the **New Equation** dialog box, type a single quote (') at the end of the equation, then enter the comment. You can also add comments in the **Edit Equations** dialog box. Anything after the single quote is ignored when the equation is evaluated.

For example: **"D2@Sketch1" = "D1@Sketch1" / 2 'height is 1/2 width**

If you want to temporarily prevent an equation from being evaluated, but you do not want to delete it, place a single quote *before* the equation. The equation is then interpreted as a comment, and it is ignored.

Operators and Functions

The following operators and functions are supported:

+	(plus sign)	addition
–	(minus sign)	subtraction
*	(asterisk)	multiplication
/	(forward slash)	division
^	(caret)	exponentiation

Function	Name	Notes
sin *(n)*	sine	*n* is angle expressed in radians
cos *(n)*	cosine	*n* is angle expressed in radians
tan *(n)*	tangent	*n* is angle expressed in radians
atn *(n)*	inverse tangent	*n* is angle expressed in radians
abs *(n)*	absolute value	returns the absolute value of *n*
exp *(n)*	exponential	returns *e* raised to the power of *n*
log *(n)*	logarithmic	returns the natural log of *n* to the base *e*
sqr *(n)*	square root	returns the square root of *n*
int *(n)*	integer	returns *n* as an integer
sgn *(n)*	sign	returns the sign of *n*

Measuring Sizes and Distances

You can measure the size of items, and the distance and angle between lines, points, surfaces and planes in sketches, part or assembly models, or drawings.

To measure the size of, or the distance between entities:

1 Click **Tools, Measure**. To keep the dialog in place while you are working, click the push pin icon 📌.

2 Click the entities to measure.

- If you select a single entity, the size of the entity (the length of an edge, the area of a face, etc.) is displayed in the **Measurements** box.

- If you select multiple entities, the distance between them is displayed.
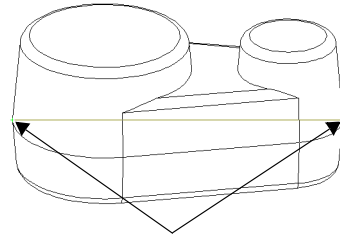
Measurement between these points

3 In the **Projection On** area, select **Screen** to measure the projection to the screen; click **Plane/Face** to measure the projection to a selected plane or planar face.

4 To select different items to measure:

- Click a selected item again to deselect it, then click a new item.

- Click an item in the **Selected Items** list and press the **Delete** key.

- Right-click in the graphics area, and select **Clear Selections** to start again.
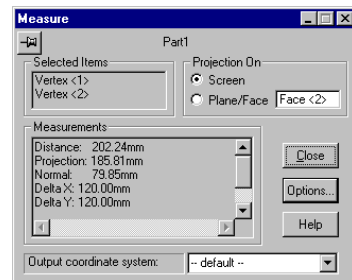
The measurements update dynamically when you change selections.

> **NOTE:** To temporarily turn off the measuring function, right-click in the graphics area, and choose **Select** from the menu. To turn the measuring function on again, click inside the **Measure** dialog box.

5 If you have defined a coordinate system, you can display the results based on that system, instead of the using the document's default coordinate system. Select the name from the **Output coordinate system** list.

See **Creating a Coordinate System** on page 3-6 for information about creating a user-defined coordinate system.

6 Click **Options** if you want to display the results using different units than the units specified for the active document.

In the **Measurement Options** dialog box, select one of these systems of units:

- **View measurement settings**. This is a *temporary* system of units. Changing the units here does not affect the units specified for the document, or the system default units (for new documents). You can specify the **Length unit**, **Angular unit**, and other options as needed (decimal places, display of fractions, etc.).

- **View global settings**. Displays the units specified for the active document.

- **View system defaults**. Displays the system default units for new documents.

  You can view the settings for the active document and the system defaults, but you cannot change them here. (To change them, click **Tools, Options, Grid/Units**.)

When you click **OK**, the values in the **Measurements** box update automatically to reflect the changed units.

7 Click **Close** to close the **Measure** dialog box.
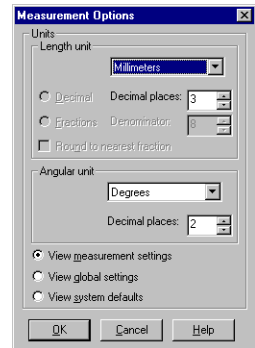
Dependency Editing

The Features toolbar contains the following tools to suppress and unsuppress features of the model.

	Suppress
	Unsuppress
	Unsuppress with Dependents
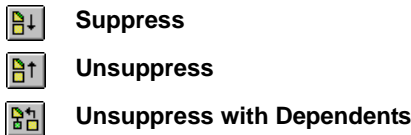
If the buttons do not appear on your toolbar, you can add them. Click **Tools, Customize**, click the **Commands** tab, then follow the instructions in the dialog box.

Suppressing Features

You can suppress a feature so you can work on the model with the selected feature temporarily removed from the model. Suppressing a feature not only removes it from the display, but also from any calculations in which it may be involved. Regeneration of complex models occurs faster when features with a large amount of detail are suppressed.

Suppression is governed by the parent/child relationships of the features. See **Parent/ Child Relationships** on page 5-3 for more information. Examine the illustrations below:

❑ In the first example, the **Tooth Cut** feature (the parent) and the **Tooth Copies** feature (the child, a circular pattern of **Tooth Cut** features) are unsuppressed.

❑ In the second example, **Tooth Cut** (the parent) is suppressed, causing **Tooth Copies** (the child) to be suppressed also (the child is *dependent* on the parent).

❑ In the third example, unsuppressing **Tooth Cut** (the parent) does not automatically unsuppress **Tooth Copies** (the child). However, unsuppressing a child feature causes its parent feature to be unsuppressed.

Tooth Cut	Unsuppressed	Suppressed	Unsuppressed
Tooth Copies	Unsuppressed	Suppressed	Suppressed

To suppress a feature:

1 Select the feature(s) in the FeatureManager tree, or select a face of the feature in the graphics area.

2 Click **Edit, Suppress**, or **Suppress** 🔽 on the Features toolbar.

– or –

1 In the FeatureManager design tree, right-click a feature, and select **Properties**.

2 In the **Feature Properties** dialog box, click **Suppressed**, and click **OK**.

The selected feature is removed from the model (but *not* deleted). The feature disappears from the model view and is grayed out on the FeatureManager design tree.

To unsuppress a feature:

1 Select the suppressed feature on the FeatureManager tree.

2 Click **Edit, Unsuppress**, or **Unsuppress** 🔼 in the Features toolbar.

– or –

1 In the FeatureManager design tree, right-click a feature, and select **Properties**.

2 In the **Feature Properties** dialog box, *deselect* **Suppressed**, and click **OK**.

> **NOTE:** While a feature is suppressed, the only way to select the feature to unsuppress it is through the FeatureManager design tree.

To unsuppress a feature and its dependents:

1 Select the suppressed feature in the FeatureManager design tree.

2 Click **Edit, Unsuppress with Dependents**, or **Unsuppress with Dependents** 🔼 on the Features toolbar.

– or –

1 Select the child feature.

2 Click **Edit, Unsuppress**, or **Unsuppress** 🔼 on the Features toolbar.

The selected feature and any features that are dependent on it are returned to the model.

Rollback

Rollback reverts the model to the state it was in before a selected feature was created. When a part is rolled back, new features are inserted at the point in the FeatureManager design tree where you rolled back, instead of at the bottom of the tree. Use this function to change a part at an earlier point of its development.

To rollback with the rollback bar:

1   Place the cursor over the rollback bar in the FeatureManager design tree. The cursor changes to a hand.

2   Click to select the rollback bar. The bar changes color from yellow to blue.

3   Drag the rollback bar up the FeatureManager design tree until it is above the feature you want rolled back,

– or –

Use the up and down arrow keys on the keyboard to move the rollback bar up or down. (Check **Arrow key navigation** in **Tools, Options, General** to enable this functionality.)

The rolled back icons are grayed out, and cannot be selected.

You can also select a feature in the FeatureManager design tree, and click **Edit, Rollback**. The rollback bar moves above the selected feature.
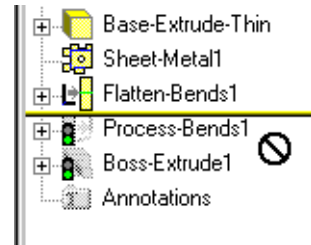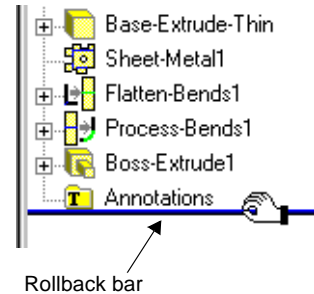
Rollback bar

To roll forward:

Move the rollback bar down the FeatureManager design tree. You can:

• Roll forward all the way by dragging to the bottom of the tree, or a few steps at a time, by dragging the rollback bar to the desired location.

• Roll forward one feature at a time. Select the rollback bar, then use the up and down arrow keys to walk through the regeneration of the features one by one. (Check **Arrow key navigation** in **Tools, Options, General** to enable this functionality.)

Derived Parts

You can create a new part directly from an existing part. The new part, called a *derived part*, has the original part as its first feature, and it is linked to that part by means of an *external reference*. This means that any changes you make to the original part are reflected in the derived part.

When a part has an external reference, its name in the FeatureManager design tree is followed by an arrow **->**. To see the name, location, and status of the externally referenced document, right-mouse click the derived part, and select **List External References.**

Derived part documents have some limitations:

- You can only *add* features in a derived part document.
- You cannot delete or modify the features of the original part in the derived part document. You can only edit the original part in its own window.

There are several types of derived parts.

❑ **Base Part** uses an existing part as the base feature for the derived part.

❑ **Mirror Part** creates a mirrored version of an existing part.

❑ **Derive Component Part** creates a part from an assembly component. Derived parts created this way include any features created in the context of the assembly. For example, you can create a cavity in a mold assembly, then derive and cut the pieces of the mold. The derived parts maintain the necessary references to the cavity component in the assembly. See **Creating Molds** on page 7-18 for more information.

To create a derived part from an existing part document:

1   Open a new part document.
2   Click **Insert, Base Part.**
3   Browse to the original part document (**.sldprt**) that you want to use as a base feature, and click **Open** (or double-click the part document).

    The derived part is opened in a new part window.

To create a mirrored, derived part:

1   In an open part document, click a model face or plane about which to mirror the part.
2   Click **Insert, Mirror Part**.

    The mirrored part is opened in a new part window.

To create a derived part from an assembly component:

1   In an assembly document, select a component.
2   Click **File, Derive Component Part**.

    The derived part is opened in a new part window.

To edit the original part from which another part is derived:

1   Right-mouse click the derived part, and select **Edit In Context**.

- If the part was derived using **Base Part** or **Mirror Part**, the original part document is opened.

- If the part was derived from an assembly component, the assembly document is opened.

2   Change the original as needed.

The changes are reflected in any derived parts that have an external reference to the original part.

File Management with External References

When you save a part that has an external reference to another document, the path and the internal ID of the referenced document are saved with the current part. The software needs this information to locate and verify the original document the next time you open the derived part.

Under **Tools, Options, External References**, the **Load referenced documents** option lets you specify whether to load referenced documents or not. Choose from these options:

- **Always.** Referenced documents are always opened, in a minimized window.
- **Never.** Referenced documents are never opened.
- **Prompt**. A dialog box appears each time you open a part with external references, and you can decide then whether to open the referenced documents or not. If you click **Don't ask me again** in the dialog box, the option is updated to reflect your choice (**Yes** changes the option to **Always**, **No** changes the option to **Never**).

Suppose you create a derived part, then you inadvertently move or rename the original document. When you try to open the derived part, the software notifies you that the externally referenced document (the original) was not found, and you are given a chance to look for it. If you choose not to look, the operation is cancelled. If you decide to look for the referenced document, you have the following choices:

❑ If you locate and select the *original* document (in a different directory, or with a new name), the derived part opens using the new name or path. Because the internal ID matches, the external reference updates as expected. When you save the part, the new name or path is saved also.

❑ If you select a *different* document, you are notified that the internal ID does not match. Then you can either accept the selected document anyway, or keep looking.

If you accept the selected document, the model may have rebuild errors, especially if you have added features in the derived part document. If the geometry of the document you selected is not the same as the geometry of the original referenced document, the additional features may have rebuild errors.

Working with Part Configurations

Part configurations allow you to create versions of a part with certain features suppressed, or with different feature parameter values. You can create several configurations of a part in a single document. When you use the part in an assembly, you can choose which configuration to use.

Creating a Configuration

You can create a part configuration that has suppressed features, and different dimensions or other parameters, and give the configuration a unique name. You can then view the part variations by selecting the named configuration.

To create a part configuration:

1   Click the Configuration tab at the bottom of the FeatureManager design tree to change to the Configuration Manager.

2   In the Configuration Manager tree, right-mouse click the part name and select **Add Configuration**. The **Add Configuration** dialog box appears.

3   Enter a **Configuration Name**. Some special characters are not allowed in a configuration name: slashes (\ or /), colon (:), asterisk (*), question mark (?), less than (<), greater than (>), pipe (|). The name must not exceed 30 characters in length.

4   Add **Comments** if desired.

5   Under **Properties for newly inserted items**, you can click **Suppress Features**. New features are always included (not suppressed) in the current, active configuration. This option controls what happens to newly created features when this configuration is *inactive*.

   • When checked, new features added to other configurations are automatically suppressed in this configuration.

   • When unchecked, new features added in other configurations are included (not suppressed) in this configuration.

6   Click **Advanced** to specify how a part configuration is referenced when it is used in a Bill of Materials. Under **Part No. displayed when used in a Bill of Materials**, select one of the following:

   • **Use document name.** Specifies that the part name is used.

   • **Use name below.** When selected, a text box opens, containing a name to be used in the Bill of Materials. The configuration name is filled in by default. Alternatively, you may enter a name (such as a part number) of your own choosing.

   Click **OK** to return to the **Add Configuration** dialog box.
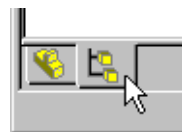
7 Click **OK** to create the new configuration.

8 Return to the FeatureManager design tree view by clicking the FeatureManager tab.

9 Suppress or unsuppress features in the active configuration.

10 Modify the dimensions as needed in the active configuration.

When a part has multiple configurations, the **Modify** dialog box lets you specify where to apply the new dimension: to **This configuration** only, or to **All configurations**.
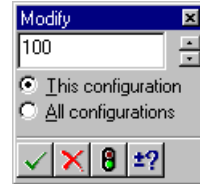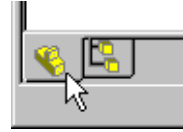
Viewing a Configuration

You can switch between the various part configurations that you have created.

To switch to a different configuration:

1 Click the Configuration tab to change to the Configuration Manager.

2 Right-mouse click the name of the configuration you want to view and select **Show Configuration**,

– or –

Double-click the configuration name.
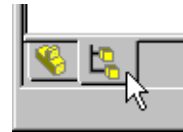
A confirmation message box appears, notifying you that some configurations may take time to display. Click **OK** to continue; click **Cancel** to abort the action. If you do not want the message to display again during this session, click **Don't ask for confirmation again in this session**.

The named configuration becomes the active configuration, and the display updates to reflect any differences in suppressed features or modified dimensions.

Editing a Configuration

To edit a part configuration:

1 View the desired configuration.

2 Change to the FeatureManager view, and change the suppression or dimensions as needed.

To edit the configuration properties:

1 Right-click the configuration name, and select **Properties**. The **Configuration Properties** dialog box appears.

2 Edit the name, comments, or properties as desired.

3 To add or modify custom properties for the configuration, click **Custom**. For more information, see **Custom Properties** on page 5-25.

4 Click **OK**.

To change only the name of the configuration, click-pause-click the name in the Configuration Manager tree, type the new name, and press **Enter**.

Opening a Configuration

When you open a part file with more than one configuration, you can specify which configuration to display. Otherwise, the part is opened in the configuration in use when it was last saved.

To open a configuration of a part:

1 Click **File**, **Open**.

2 Navigate to the desired part, select the **Configure** check box, and click **OK**.

The **Configure Document** dialog box appears, with a list of available configurations.

3 Select the desired configuration.

4 Click **OK**. The part opens in the selected configuration.

Deleting a Configuration

To delete a part configuration:

1 In the Configuration Manager, click the name of the configuration you want to delete.

2 Press the **Delete** key (or click **Edit, Delete**), and click **Yes** to confirm.

> **NOTE:** Deleting a configuration *does not* delete any features.
>
> You cannot delete the configuration that is currently active.

Design Table

A design table allows you to build multiple configurations of parts by specifying parameters in an *embedded* Microsoft Excel spreadsheet. The design table is saved in the part file and is *not linked* back to the original Excel file. Changes you make in the part are not reflected in the original Excel file.

To use design tables, you must have Microsoft Excel 97 installed on your computer.

You can control the following items in a design table:

- dimensions and suppression of features
- configuration properties, including part number in a bill of materials, comments, and custom properties

See **Specifying Parameters** on page 5-19 for detailed information.

To create a design table:

1   Open Microsoft Excel and create a spreadsheet.

> **NOTE:** This procedure describes entering the data in Excel before inserting the table. You can also insert a partially empty spreadsheet and complete editing it in SolidWorks.

2   In the first *column* (row header cells **A2, A3,** etc.), enter the names of the design configurations that you want to create. Configuration names must not be longer than 30 characters, or contain these special characters: slashes (\ or /), colon (:), asterisk (*), question mark (?), less than (<), greater than (>), pipe (|). Leave cell **A1** blank.

3   In the first *row*, (column header cells **B1, C1,** etc.), enter the parameters that you want to control.

	A	B	C
1		parameter1	parameter2
2	**configuration1**	*value1*	*value3*
3	**configuration2**	*value2*	*value4*

4   Enter the parameter values in the spreadsheet cells.

5   Save the spreadsheet.

6   Open the part where you want to use the design table.

7   Click **Insert, Design Table**. In the **Open** dialog box, navigate to the spreadsheet file and click **Open**.

The spreadsheet appears in the part window, the Excel menus and toolbars are displayed, and a message is displayed, indicating the names of any new configurations that were generated by the design table.

8   Edit the design table, if necessary. When you are finished editing, click anywhere outside the table to close it. The SolidWorks menus and toolbars reappear.

To display the configurations added by the design table, click the Configuration tab at the bottom of the window. Double-click the name of the configuration, or right-click the name and select **Show Configuration**.

To edit a design table:

1 Click **Edit, Design Table**. The table appears, and the Excel menus and toolbars are displayed.

2 Edit the table as needed. You can change the parameter values in the cells, add rows for additional configurations, or add columns to control additional parameters.

3 Click anywhere outside of the design table to close it. The configurations update to reflect your changes.

NOTE: Changes you make in the part (not in the table) are not reflected in the original Excel file, or in the embedded file. The next time that the design table is evaluated, the configurations are updated to reflect the data in the table.

For example, assume that **D1@Sketch1** is driven by the design table. If you modify **D1@Sketch1** outside of the table (in the sketch itself, when one of the configurations from the table is active), *the table data does not change*. Furthermore, the value contained in the corresponding table cell is restored to the configuration the next time that the table is evaluated, and your change is undone.

To delete a design table:

1 Click **Edit, Delete Design Table**.

2 Confirm the deletion and click **OK**.

Deleting the design table *does not* delete the configurations created by it.

Specifying Parameters

When used in a design table, the names of dimensions and features must match the name in the model. To ensure an exact match, it is recommended that you copy and paste the name from the **Properties** dialog box for the selected item.

To copy and paste a feature or dimension name:

1 Right-mouse click the item (dimension or feature), and select **Properties**.

2 In the **Properties** dialog box, copy (**Ctrl+C**) the item name (for dimensions, be sure to copy the **Full name**).

3 Paste (**Ctrl+V**) the name into the appropriate column header cell.

Some parameter specifications that appear in the column header cells include a *keyword*, such as **$PARTNUMBER**, or **$COMMENT**. The parameter keywords are *not* case-sensitive.

Values in the body cells are *not* case-sensitive.

This is an example of a design table that creates three configurations, adds a comment for each one, suppresses a feature, and controls a dimension.

	A	B	C	D
1		$COMMENT	$STATE@Cut-Extrude1	D3@Sketch2
2	rev1	large, no cuts	suppressed	125
3	rev2	medium	unsuppressed	100
4	rev3	small	unsuppressed	75

> **NOTE:** You may not include any rows or columns with blank headers in the middle of the design table. The software stops evaluating the data when it reaches a row or column that has an empty header cell. Data that appears in or after a row or column with an empty header is ignored.

Dimensions

A dimension name is in the form:

Dimension@Feature or **Dimension@Sketch<n>**

For example, the full name for the depth of an extruded base feature is **D1@BaseExtrude**. You can also use the **Properties** command to assign more meaningful names to dimensions. See **Dimension Properties** on page 9-11.

In the table body cells, enter the value for the dimension. If a cell is left blank, the default value is zero. This usually results in an error, depending on the type of dimension.

Suppression of features

There are two ways to specify the suppression of features.

❑ **Method 1 (recommended).** The column header for controlling feature suppression uses this syntax:

$STATE@feature

where **feature** is the name of the feature. For example, the column labelled **$STATE@Hole1** controls the suppression of the first hole feature.

In the table body cells, enter the value for the desired suppression: **Suppressed** (or **S**), **Unsuppressed** (or **U**). If a cell is left blank, the default is **Unsuppressed**.

❑ **Method 2.** Enter only the feature name in column header cell. To suppress the feature, leave the table body cell *blank*. To include the feature, enter any string in the body cell. This is the syntax that was used in design tables in versions prior to SolidWorks 98Plus, and is included for backward compatibility.

Part number

You can control the value that is displayed in the **Part No.** column in a Bill of Materials. The column header uses this syntax:

> **$PARTNUMBER**

This corresponds to the **Use name below** option in the **Configuration Properties, Advanced** dialog box.

In the table body cells, enter the part number for each configuration. If a cell is left blank, the default is to use the configuration name (the same as checking **Use name below**, and leaving the default configuration name in the text box).

Comment

The **Configuration Properties** dialog box has a **Comment** box, where you can enter a description, or other additional information about the configuration. The column header for filling in this box for each assembly configuration uses this syntax:

> **$COMMENT**

In the table body cells, enter the comments for the configuration. Comments are optional; if a cell is left blank, the **Comment** box is empty.

Custom properties

The column header for specifying a custom property uses this syntax:

> **$PRP@*property***

where ***property*** is the name of a custom property. You can use one of the custom properties listed in the **Summary Info** dialog box (click **File, Properties**, **Configuration specific**), or you can add a new custom property.

For example, use the header **$prp@Client** to control the custom property **Client** for each configuration. In the table body cells, enter the property value for each configuration. If a cell is left blank, the property is undefined for the configuration.

NOTE: If you plan to use the custom properties as columns in the bill of materials, do not use property names that contain spaces. See **Custom properties in a bill of materials** on page 9-32.

User notes

The design table can include additional columns or rows for information only (notes, intermediate calculations, etc.). Use the keyword **$USER_NOTES** in the header of any column or row that you do not want evaluated. You may use as many of these rows or columns as you need. However, remember that you may not include rows or columns with empty headers in the middle of the table.

Summary

parameter syntax (header cell)	legal values (body cell)	default if value left blank
$comment	any text string	empty
$partnumber	any text string	configuration name
feature_name	blank (suppressed) any string (unsuppressed)	Suppressed
$state@ feature_name	Suppressed, S Unsuppressed, U	Unsuppressed
dimension@feature	any legal decimal value for the dimension	zero (error if not a legal value)
$prp@property	any text string	property is undefined
$user_notes	any text string	not evaluated

Checking Model Geometry

The SolidWorks software automatically checks the model geometry as you create each feature, to prevent invalid geometry. It also provides additional tools for checking the model, which you can use for the following purposes:

- To examine imported geometry (when you import the body as a whole, from another system, such as an IGES or STEP file)
- To make sure that the model exports properly to other formats
- To determine the cause when the model looks wrong

Check Entity

You can check a part model to identify invalid or undesirable geometry.

To check a model:

1 With a part document active, click **Tools, Check**.

2 In the **Check Entity** dialog box, select the type of check that you want to perform:
   - **Check body**. This is a fast check of the whole body that reports any errors on faces or edges. You can use this option first, to determine if the model geometry is valid. If you design very complex models, it is a good idea to use this regularly, after creating several features.
   - **Check features**. If any errors are found when you use **Check body**, you can use this option to identify which features contain errors. This is slower, especially for complex models, because it examines the model feature by feature, to locate the source of the problem.
   - **Check selected item**. This checks a selected face or edge. The name of the selected item appears in the **Item to check** box.

3 Select the type(s) of problems that you want to check for: **Invalid face(s), Invalid edge(s),** and/or **Short edge(s).**

   When you check for **Short edge(s)**, you specify an minimum edge length in the spin box. The software reports any edges that are less than the specified length.

4 To find the smallest radius of curvature on the model, select **Min. radius of curvature**. The face with the smallest radius of curvature is reported. If you chose the **Check selected item** option in Step 2, the smallest radius of curvature on the selected item is reported.

5 Click **Check**. The dialog box remains open so you can check again.

   The **Found** column displays the number of errors of each type.

   The **Result list** box lists the errors by name. To highlight the geometry, and to display a description of the error, select the item in the **Result list**.

6 Click **Close** to close the dialog.

Force Regeneration with Verification on Rebuild

By default, each time you add or modify a feature, the feature is checked against any adjacent faces and edges. For most situations, this default level of error checking is adequate, and it results in fast rebuilding of the model.

However, under certain circumstances, it is recommended that you perform more rigorous checks on the model:

- When **Check Entity** reveals problems in the model
- When the model looks wrong (and you have determined that it is not the result of a display problem)
- When you have problems importing or exporting the model in different formats

To increase the level of error checking, enable **Tools, Options, Verification on rebuild**. When enabled, the software checks every new or changed feature against *all* existing faces and edges, not just adjacent faces and edges. Features that cause invalid geometry fail when this option is enabled.

> **NOTE:** This option has a serious effect on performance. Rebuilding the model is considerably slower and more CPU-intensive. This is a *system option*; it applies to *all* documents, not just the active document. Use this option only when needed, then be sure to disable it when you are done.

To verify the model with the highest level of error checking:

1. Click **Tools, Options, Performance**, and click the **Verification on rebuild** option.
2. Press **Ctrl+Q** to perform a *force regeneration* of the entire model.

> **NOTE: Ctrl+Q** causes a complete rebuild of all the features in the model, from the beginning. A normal **Edit, Rebuild** operation **(Ctrl+B)** only rebuilds new or changed features and their children.

3. Examine and correct any features that fail, beginning with the *first* failed feature in the FeatureManager design tree. Often fixing one failed feature allows subsequent failed features to rebuild successfully.
4. When you are satisfied that all the errors are corrected, click **Tools, Options, Performance**, and *deselect* the **Verification on rebuild** option.

Custom Properties

You can use *custom properties* to add more information to a part or sub-assembly document, or to a configuration of the model in the document. Custom properties can be used in several ways:

- In a bill of materials (see **Bill of Materials** on page 9-29).

- As advanced selection criteria for managing assembly configurations (see **Component Selection by Properties** on page 6-49).

- As text in notes (see **Links to Document Properties** on page 9-19).

- In drawing templates (see **Customizing a Drawing Template** on page 8-5 and **Linking Notes to Document Properties** on page 8-6).

To add custom properties:

1 Click **File, Properties**.

2 To add properties *to the file*, click the **Custom** tab.

   To add properties *to the active configuration*, click the **Configuration specific** tab.

3 Enter a **Name** for the property, or choose one from the list.

   > **NOTE:** If you plan to use the custom properties as columns in the bill of materials, do not use property names that contain spaces. See **Custom properties in a bill of materials** on page 9-32.

4 Select the **Type** of value (text, number, etc.) the custom property will use.

5 Enter a **Value** for the custom property that is compatible with the selection in the **Type** box.

6 Click **Add.** The **Properties** box displays the name, value and type of the custom properties.

To modify custom properties:

1 In the **Name** box, click the property that you want to change.

2 Edit the **Type** or **Value** as needed.

3 Click **Modify**.

To delete custom properties:

In the **Properties** box, click the property that you want to delete, then click **Delete**.

Displaying Annotations in Parts

You can add annotations to your model to further clarify the information the part document provides. You can add much of the necessary detailing of your models in the part document itself. This includes dimensions, notes, symbols, etc. Then you can import the dimensions and annotations from the model into a drawing.

For more information about dimensions and annotations, see Chapter 9, "Detailing."

You can control the display of annotations in the part window using right-mouse menu selections on the **Annotations** folder in the FeatureManager design tree.

To select the types of annotations to display:

1   In the FeatureManager design tree, right-mouse click **Annotations** and select **Details**. The **Annotation Properties** dialog box appears.

2   Under **Display filter**, select the annotation types to display (**Notes**, **Datums**, etc.), or click **Display All Types** to show all types of annotations available for the part.

3   Change the values in the **Text Scale** boxes to modify the scale of the text used in the annotation.

4   Select from the following options:

   • **Always display text at the same size.** When checked, annotations and dimensions are displayed at the same size regardless of zoom.

   • **Display items only in the view in which they are created.** When checked, any annotation is only displayed when the part is in the same view orientation as when the annotation was inserted. Rotating the part or selecting a different view orientation removes the annotation from the display.

   • **Display Annotations.** When checked, all annotation types that are selected in the display filter are displayed.

5   Click **OK** to accept your changes; click **Cancel** to exit the dialog without saving the changes.

To toggle the display of all annotations:

Right-mouse click **Annotations** and select **Display Annotations**. All annotation types that are selected in the **Annotation Properties** dialog are displayed. This has the same effect as checking the **Display Annotations** box in the **Annotation Properties** dialog box.

To toggle the display of feature dimensions:

Right-mouse click **Annotations** and select **Show Feature Dimensions.** This has the same effect as checking the **Feature Dimensions** box in the **Display filter** of the **Annotation Properties** dialog box.

To toggle the display of reference dimensions:

Right-mouse click **Annotations** and select **Show Reference Dimensions.** This has the same effect as checking the **Reference Dimensions** box in the **Display filter** of the **Annotation Properties** dialog box.

Section View of a Model

You can display a view of the model cut through the part or assembly. You can use any combination of standard planes, reference planes, and planar model faces as section planes.

You may find it useful to view both the section view and the complete view of the model at the same time by splitting your screen into two windows. (Drag the split controls on the bottom or right side of the window frame.)

To display a section view of the model:

1 Click **View, Display, Section View,** or click 🔄 on the View toolbar.

2 Click on one or more planes in the FeatureManager design tree, or click one or more planar faces on the model.

Alternatively, you can pre-select planes or planar faces before clicking **View, Display, Section View**. If nothing is selected, **Plane1** is used by default. To pre-select multiple planes or faces, hold the **Ctrl** key as you select.

The selected planes and faces are listed in the **Section Plane(s)/Face(s)** box.

3 Click a plane or face in the **Section Plane(s)/Face(s)** box, then enter an offset value (or use the spinbox arrows to change the value) in the **Section Position** box to move the section plane to the desired position for the cut.

4 Click **Flip the Side to View** to reverse the viewing direction of the section cut.

5 Select a **Preview/Display** mode:

• When **Preview** is checked, the section view is updated whenever any item in the dialog box is changed. If you need to adjust more than one setting, especially with a large part or assembly, this option may not be efficient.

• When **Preview** is not checked, click the **Display** button to update the section view. You can make multiple changes to the settings, then click **Display** again to update the view with all the changes at once.

6 Click **OK**.
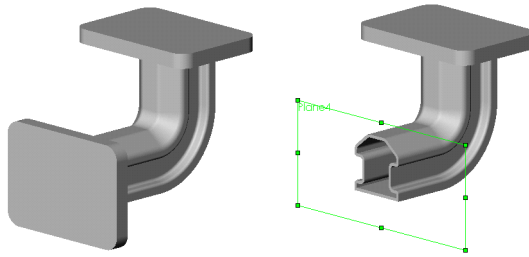
To use the viewing plane to cut a section view:

Instead of model faces and planes, you can use a *viewing plane* to cut the section view. The viewing plane is an invisible plane that is parallel to the screen, and normal to your line of sight. The initial position for this plane is the center of the part body bounding box. As you rotate the part, the angle of the viewing plane relative to the part body changes, cutting the part at a different angle.

To use the viewing plane to make a section cut, click **Use Viewing Plane** in the **Section View** dialog box.

You can offset the **Section Position** of the viewing plane. The **Flip the Side to View** option is not available when you use the viewing plane.

To change the section view:

Click **View, Modify, Section View** and change the settings in the **Edit Section View** dialog box and/or select a different plane.

To return to a normal view of the model:

Click **View, Display, Section View** to remove the checkmark beside the option in the menu, or click 🔲 again.

Mass Properties and Section Properties

You can display the density, mass, volume, surface area, center of mass, inertia tensor, and principal axes of inertia of a part or assembly model.

To display mass properties:

1 Click **Tools, Mass Properties.**

To display the mass properties of an assembly component, select the component first.

If no assembly component is selected, the mass properties for the entire assembly are calculated.

The results are displayed in the **Mass Properties** dialog box, and the principal axes and center of mass are displayed graphically on the model.
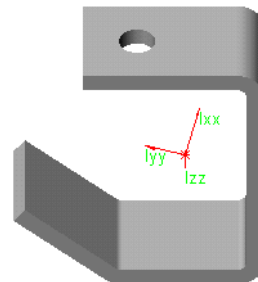
2 You can evaluate a different component without closing the **Mass Properties** dialog box. Select the component, then click **Recalculate**.

3 Click **Options** if you want to display the results using different units than the units specified for the active document.

In the **Measurement Options** dialog box, select one of these systems of units:

- **View measurement settings**. This is a *temporary* system of units. Changing the units here does not affect the units specified for the document, or the system default units (for new documents). You can specify the **Length unit**, **Angular unit**, and other options as needed (decimal places, display of fractions, etc.).

- **View global settings**. Displays the units specified for the active document.

- **View system defaults**. Displays the system default units for new documents.

  You can view the settings for the active document and the system defaults, but you cannot change them here. (To change them, click **Tools, Options, Grid/Units**.)

Click **OK**, then click **Recalculate** to display the results in the selected units.

4 If you defined a coordinate system, you can display the results based on that system, instead of using the document's default coordinate system. Select the name from the **Output coordinate system** list, then click **Recalculate** to update the results.

See **Creating a Coordinate System** on page 3-6 for information about creating a user-defined coordinate system.

5 Click **Print** if you want to print the results directly from this dialog box.

6 Click **Copy** if you want to copy the information to the clipboard, then paste it into another document.

7 Click **Close** to close the dialog box.

Section Properties

You can display certain mass properties for a planar face, sketch, or section face in a drawing view. The properties reported depend on the entity selected.

To display section properties:

1   Select a planar model face in any document, or the crosshatch section face in a section view in a drawing, or select a sketch (either click the sketch in the FeatureManager design tree, or right-click a feature, and select **Edit Sketch**).

> **NOTE:** Section properties of sketches can only be calculated for *closed* profiles.

2   Click **Tools, Section Properties.** The results are displayed in the **Section Properties** dialog box, and the principal axes and center of mass are displayed graphically on the model.

The options and controls in the **Section Properties** dialog box are the same as those found in the **Mass Properties** dialog box. See the previous section for information about using them.

Part Color and Lighting

You can change the color of a part and the properties that define the way the material reacts to light. This section describes changing the color of the *whole part*; for information about changing the color of an individual feature, see **Feature Properties** on page 5-2.

To change the color of a part:

1 Click **Tools, Options,** and select the **Color** tab.

2 Under **System**, select **Shading**, then click the **Edit** button.

3 Choose a color from the palette, or click **Define Custom Colors** to specify a custom color.

4 Click **OK** to close the **Color** dialog.

5 Make sure that **Apply To** is set to **Active Document**, then click **OK** to close the **Options** dialog.

To change the way a part reacts to light:

1 Click **Tools, Options,** and select the **Color** tab.

2 Under **System**, select **Shading**, then click the **Advanced** button.

3 Use the slider controls to change the display properties.
   - **Ambient**. Light reflected and scattered by other objects
   - **Diffuse**. Light scattered equally in all directions on the surface
   - **Specularity**. Ability of a surface to exhibit highlights
   - **Shininess**. A glossy, reflective surface or a dull, matte surface
   - **Transparency**. Ability to pass light through the surface
   - **Emission**. Ability to project light from the surface

4 Click **OK** when done.

Light Source Types

You can adjust the direction, intensity, and color of light in the shaded view of a model. There are three types of light source:

❑ **Ambient Light.** Light that illuminates the part evenly from all directions. For example, the light in a room with white walls has a high degree of ambient light because the light reflects off the walls and other objects; a spot-lighted area has little ambient light because the light is focused in a narrow beam.

❑ **Directional Light.** Light that comes from a source that is infinitely far away from the surface. This is a columnated light source consisting of parallel rays arriving from a single direction. (The sun is a real-world example of directional light.) The direction of the light source is apparent to the viewer.

❑ **Spot Light.** A restricted, focused, point-source light that emits a cone-shaped beam that is brightest at its center.

To adjust the lighting in a shaded view:

1 Click **View**, **Lighting**.

The **Light Sources** dialog box appears, with a tab for each light source. By default, there is an **Ambient Light** source and one **Directional Light** source.
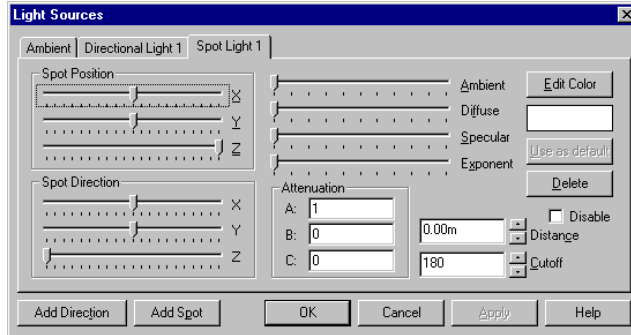
2 On the tab for each light source, select the color, direction, intensity, and other characteristics of the light source, as described below. Experiment until you get the results that you want.

3 Choose from these options as needed:

- **Edit Color.** Defines the color of the light striking the model surface. Select a color from the palette or click **Define Custom Color** to select a custom color.

- **Use as default.** Preserves the **Direction**, **Ambient**, **Diffuse**, and **Specular** values as the settings for the default **Ambient** and **Directional Light** in new models. (Not available for **Spot Light**.)

- **Delete.** Removes the current light source page from the dialog box. (Not available for **Ambient**.)

- **Disable.** Turns off the light source defined on the current page. To turn the light source back on, deselect this checkbox.

4 To add more light sources, click the **Add Direction** or **Add Spot** buttons at the bottom of the dialog box. You may use a maximum of eight light sources in addition to the ambient light source. For each new light source, click the tab and specify the characteristics of that light source.

5 Click **OK** to save the changes; click **Cancel** to end the session without saving the changes.

Light Source Characteristics

The light source characteristics vary depending on the type of light source. Move the slider to the right to increase the effect of each light source characteristic.

Ambient Light Characteristics

- **Ambient.** Light that has been scattered by other objects in the environment.

Directional Light Characteristics

- **Position.** Adjust the sliders to specify the position of the directional light source relative to the center of the scene.
- **Ambient.** Light that has been scattered by other objects in the environment.
- **Diffuse**. Light that comes from one direction, but once it strikes the surface the light scatters equally in all directions so it appears equally bright no matter where the observing eye is located.
- **Specular**. Light that comes from one direction and bounces off the surface. Specularity indicates the ability of a surface to exhibit highlights. A matte surface has no specularity; a glossy surface has high specularity.

Spot Light Characteristics

These additional characteristics are unique to the **Spot Light**:

- **Spot Position.** Specifies the position from which the light arrives. Adjust the sliders to specify the position of the spot light relative to the center of the scene.
- **Spot Direction**. Specifies where the light is pointing in the scene. Adjust the sliders to specify where the spot light strikes the scene. Think of these controls in the same terms as a spotlight in a theater. The position of the light is usually above and behind the audience; the direction the light points is toward a specific part of the stage.
- **Exponent.** The property that results in a more or less concentrated beam of light. A higher spot exponent results in a more focused light source. Move the slider to the right to increase the light concentration.
- **Attenuation.** The property that decreases the intensity of light as the distance from the light source increases. The three boxes (**a**, **b**, and **c**) hold values that are used in an equation to arrive at an attenuation factor.

$$\text{attenuation factor} = \frac{1}{a + b * d + c * d^2}$$

- **Distance.** The distance from the light source to the surface of the model. Use the spinbox arrows to modify the distance. (This value is represented by **d** in the attenuation factor equation.)
- **Cutoff**. The property that describes the spread of the beam, restricting the shape of the light cone. The maximum value is 180; reduce the value to reduce the spread of the light cone.

Picture

You can display a TIFF image (**.tif**) as a background in a part or assembly document.

To insert a TIFF image into a document:

1   With a part or assembly document open, click **Insert, Picture**.

2   Browse to the **.tif** file you want to display, and click **Open**. You can only have one TIFF image in a document.

❑ To toggle the display of the TIFF image, click **View, Display, Picture**. When the **Picture** option is checked, the image is displayed; when the **Picture** option is not checked, the image is hidden.

❑ To delete a TIFF image from a document, click **View, Modify, Picture, Delete**.

❑ To replace one TIFF image with another, click **View, Modify, Picture, Replace**. Browse to the new **.tif** file you want to display, and click **Open**.

Curvature

You can display the model with the surfaces rendered in different colors according to their local radius of curvature. Curvature is defined as the reciprocal of the radius (1/radius), in current model units. By default, the greatest curvature value displayed is 1.000, and the smallest value is 0.001.

As the radius of curvature decreases, the curvature value increases, and the corresponding color changes from black (0.001), through blue, green, and red (1.000). You specify the color gradations associated with the curvature values on the **Color** tab of the **Options** dialog box.

As the radius of curvature increases, the curvature value decreases. A planar surface has a curvature value of zero (1/∞).

To render curved surfaces in colors:

Click **View, Display, Curvature.** The curvature of the model is displayed in color as long as the menu item remains checked. When you point to a model surface, the curvature value and the radius of curvature are shown beside the cursor.

To remove the color, click **View, Display, Curvature** again.

To render an individual model face in colors:

Right-mouse click the model face and select **Face Curvature**. The curvature of the selected face is displayed in color as long as the menu item remains checked. When you point to a model surface, the curvature value and the radius of curvature are shown beside the cursor.

To remove the color, right-mouse click the model face and click **Face Curvature** again.

To set values for the color scale for displaying curvature:

1 Click **Tools, Options**, and select the **Color** tab.

2 Click the **Curvature** button.

3 Select the curvature value edit boxes that you want to change and enter new values. You can modify the values at five intervals on the color scale; values are interpolated for the intermediate colors in the range. The following table shows the curvature values for some common radius values.

Radius	Curvature	Radius	Curvature
0.125	8	10	0.1
0.25	4	20	0.05
0.5	2	25	0.04
0.75	1.333	30	0.033
1	1	40	0.025
1.5	0.667	50	0.02
2	0.5	75	0.013
2.5	0.4	100	0.01
3	0.333	125	0.008
4	0.25	200	0.005
5	0.2	250	0.004
7.5	0.133	500	0.002

4 Click the **Apply** button to see how the color scale changes in response to the new values.

5 Click **OK** to save the new settings.

Sending Part Documents

You can send a part document to another system using electronic mail. This function uses the electronic mail application installed on your system.

To mail a part document to another computer:

1 With a part document open, click **File, Send To**.

2 Enter your mail password, the mailing address of the recipient, and any other information requested by your mail application.

The open document is attached to the mail message.

Working with Assemblies

You can build complex assemblies consisting of many parts and sub-assemblies.

This chapter describes the basics of creating an assembly from parts you have already built, and general information about working with assemblies of any kind. The following topics are discussed:

❑ Design methods

❑ Creating an assembly

❑ Assembly mating

❑ Exploding an assembly view

❑ Simplifying large assemblies

❑ Using assembly configurations

❑ Using advanced component selection methods

❑ Customizing the appearance of an assembly

❑ Mailing an assembly file

Design Methods

You can create an assembly using bottom-up design, top-down design, or a combination of both methods. This chapter explains the bottom-up method. For information about the top-down method, see Chapter 7, "Working with Parts within an Assembly.".

Bottom-Up Design

Bottom-up design is the traditional method. In bottom-up design, you create parts, insert them into an assembly, and mate them as required by your design. Bottom-up design is the preferred technique when you are using previously constructed, off-the-shelf parts.

An advantage of bottom-up design is that because components are designed independently, their relationships and regeneration behavior are simpler than in top-down design. Working bottom-up allows you to focus on the individual parts, and it is a good method to use if you do not need to create references between parts.

1. Create parts

2. Create assembly

3. Insert and mate parts
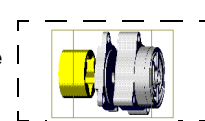
Top-down Design

Top-down design is different because you start your work in the assembly. You can use the geometry of one component to help define other components, or to create cut or hole features that are added only after the components are assembled. Or you can start with a layout sketch, define fixed component locations, planes, etc., then design the parts referencing these definitions.

For example, you can insert a part in an assembly, then build a fixture based on this part. Working top-down, creating the fixture *in context*, allows you to reference model geometry, so you can control the dimensions of the fixture by creating geometric relations to the original part. That way, if you change a dimension of the part, the fixture automatically is updated.

1. Create assembly

2. Create parts in place
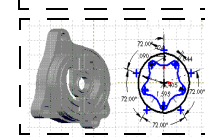
3. Create (or insert) other parts
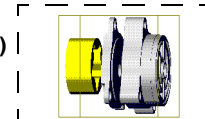
Understanding Assemblies

When a part or assembly is inserted into an assembly, it is referred to as a *component*. An assembly can be a *sub-assembly* of another assembly. Components are *linked* to the assembly file. Assembly files have the **.sldasm** extension.

The Assembly Window

This is a typical assembly window.
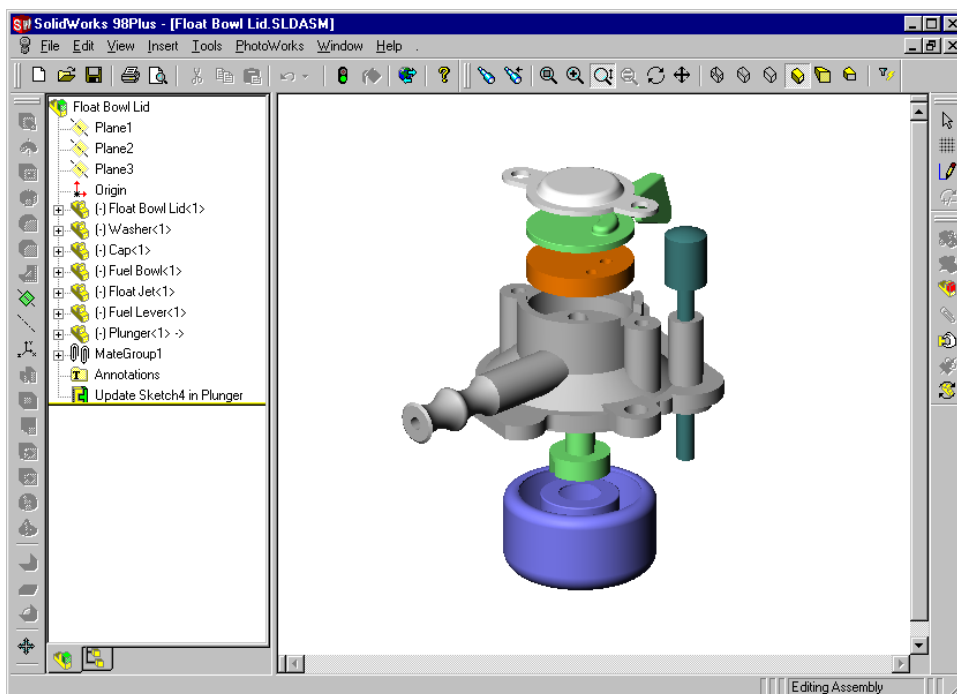
The FeatureManager design tree displays the names of the:

- Assembly, sub-assemblies and parts
- Mategroups and mating relations
- Assembly planes, origin, and annotations
- Assembly features (cuts or holes), and component patterns
- Part features built in the context of the assembly

The first item in the FeatureManager design tree is the name of the assembly. You can expand or collapse each sub-assembly or component to view its detail by clicking the ⊞ beside the component or sub-assembly name.

To collapse all the items in the tree, right-mouse click the assembly icon at the top of the tree, and select **Collapse Items**.

You can use the same part multiple times within an assembly. Each component or sub-assembly has the suffix **<n>**. For each occurrence of the component in the assembly, the number **n** is incremented.

In the FeatureManager design tree, a part or sub-assembly name may have a prefix, that provides information about the state of its relationships to other components. The prefixes are:

    (–)   underdefined

    (+)   overdefined

    (f)   fixed

    (?)   not solved

The absence of a prefix indicates that the component's position is fully defined.

The Assembly Toolbar

The Assembly toolbar gives you quick access to these frequently used assembly tools.

    Move Component

    Rotate Component Around Centerpoint

    Rotate Component Around Axis

    Mate

    Edit Part

    Hide/Show Component

    Change Suppression State

For information about moving and rotating components in an assembly, see **Positioning Components in an Assembly** on page 6-15.

For information about mating components, see **Assembly Mating** on page 6-17.

For information about editing a part in an assembly, see **Editing Parts in the Assembly Context** on page 7-5.

For information about hiding and showing components, or changing the suppression state, see **Simplifying Large Assemblies** on page 6-30.

The FeatureManager Design Tree in an Assembly

There are times when you want to focus on the structure or hierarchy of the design rather than the details of the sketches and features. You can view an assembly's hierarchy in the FeatureManager design tree.

You may want to focus on the design of the assembly without all of the features of the components. By changing the FeatureManager design tree's display mode, you can view the assembly reference planes, components, sub-assemblies, assembly features, and mating relationships. The image of the assembly, including all its components (other than any suppressed or hidden ones), still appears in the SolidWorks window.

Each of these ways of viewing the assembly affects only the level of detail displayed in the FeatureManager design tree. The assembly itself is not affected.

To display the hierarchy or feature detail of an assembly:

❑ To display the hierarchy, right-mouse click the assembly name in the FeatureManager design tree, and select **Show Hierarchy Only**.

The FeatureManager design tree displays only the parts and sub-assemblies, but no lower level detail.

❑ To display the detail again, repeat the procedure, selecting **Show Feature Detail**. To display the detail for a single component, double-click the component in the FeatureManager design tree.

To view an assembly by dependencies or by features:

❑ To display the dependencies, right-mouse click the assembly name in the FeatureManager design tree, and select **View Dependencies**, or click **View, FeatureManager Tree, By Dependencies**.

Dependent items (instead of features) are listed under each component. Dependent items include mates and component patterns.

❑ To display the features again, right-mouse click the assembly name in the FeatureManager design tree, and select **View Features**, or click **View, FeatureManager Tree, By Features**.

Creating an Assembly

An assembly can consist of parts you have previously designed as well as components that you create within the assembly. This section describes how to add, replace, copy, delete, verify, and save components in an assembly.

Adding Components to an Assembly

When you place a component in an assembly, the part file is linked to the assembly file. The component appears in the assembly; however, the data for the component remains in the source part file. Any changes you make to the part file will update the assembly.

There are several ways to add components to a new or existing assembly.

To add components by using the menu:

1   With an assembly open, click **Insert, Component, From File**. The **Open** dialog box appears.

2   Browse to the directory that contains the part or sub-assembly document that you want to insert into the assembly, and select the document. (You can also double-click the document to select it and close the dialog box in one step.)

   To see the selected component before you insert it, click **Preview**.

   To select which configuration of the document to use, click **Configure**. For information about creating configurations, see **Working with Part Configurations** on page 5-15, and **Using Assembly Configurations** on page 6-35.

3   Click **Open**. If you clicked **Configure**, select a configuration from the list and click **OK**.

4   Click the cursor in the area of the assembly window where you want to place the component.

To add components by dragging and dropping from an open document window:

1   Open an assembly document, and open the component document (either a part or a sub-assembly) that you want to insert into the assembly.

2   Click **Window, Tile Horizontally** or **Tile Vertically**.

3   Drag the icon from the top of the FeatureManager design tree area in the component window, and drop it in the assembly window.

   If you are adding a part (not a sub-assembly), you can also drag the part from the graphics area. Depending on the entity that you use to drag the part, you can create some mating relationships automatically when you drop the part. See **Automatic Mating** on page 6-21.

If the component has multiple configurations, you can select which configuration to insert.

- To insert the configuration that is currently active, drag and drop the icon from the top of the FeatureManager design tree.
- To insert a configuration that is not currently active, click the Configuration Manager tab, then drag and drop the name of the desired configuration.

To add components by dragging and dropping from Windows Explorer:

1 Open an assembly.

2 Open **Windows Explorer**. Browse to the directory that contains the desired component (either a part or a sub-assembly document).

3 Drag the document icon from the **Explorer** window, and drop it in the assembly window.

If the component has multiple configurations, the **Select a configuration** dialog box appears. Select the configuration you want to insert, and click **OK**.

To add a component from a hyperlink in Internet Explorer:

1 In Internet Explorer (version 4.0 or higher), navigate to a location that contains hyperlinks to SolidWorks part files.

For example, on the SolidWorks website, in the Support section, navigate to the Solid Modeling Library page. You may also be able to obtain SolidWorks parts from certain third-party vendors.

2 Drag a hyperlink from the Internet Explorer window, and drop it in the assembly window. The **Save As** dialog box appears.

3 Navigate to the directory where you want to save the part, enter a new name if desired, and click **Save**.

The part document is saved locally. If the part has multiple configurations, the **Select a configuration** dialog box appears. Select the configuration you want to insert, and click **OK**. The part is added to the assembly.

To add another component instance by dragging and dropping:

1 With an assembly open, press the **Ctrl** key, then click a component in the FeatureManager design tree.

2 Drag the component into the graphics area, then release the mouse button.

A copy of the component is added to the assembly, and the instance number suffix is incremented.

Adding Patterns of Components

You can add a pattern of components in an assembly based on a feature pattern of an existing component. You can also define a pattern for placing components in an assembly in much the same way as you define a feature pattern in a part. Component patterns are *time-dependent*; they depend on an existing mating relation. See **Time-Dependent Features** on page 7-10 and **Adding a Component Pattern** on page 7-12 for more information.

Inferencing to the Assembly Origin

When you insert a component by dragging it from an open part window, or by clicking **Insert, Component, From File**, you can inference the assembly origin. Look for the cursor shown here.

When you inference the assembly origin:

- the component is fixed **(f)**
- the component origin is coincident with the assembly origin
- the planes of the component and the assembly are aligned

You can also drag a part from an open part window and drop it in the FeatureManager design tree to inference the origin.

Deleting a Component from an Assembly

To delete a component from an assembly:

1   Click the component in the graphics area or in the FeatureManager design tree.

2   Press the **Delete** key, or click **Edit**, **Delete**.

3   Click **Yes** to confirm the deletion. The component and all its dependent items (mates, component patterns, explode steps, etc.) are removed.

Managing the Part Files of an Assembly

Saving an Assembly and Its Parts

Under the **File** menu, there are several ways to save an assembly document. Whichever method you use, the current locations of all referenced files (parts and sub-assemblies) are saved with the assembly document.

❑ **Save**. When you **Save** an assembly, you save the assembly document, and any referenced part or sub-assembly documents that have changed.

❑ **Save As**. When you use **Save As**, you save a copy of the assembly document with a new name or path. You can create a new directory 📁 in the **Save As** dialog box if needed. After you use **Save As**, you are working in the new assembly document. The original assembly document is closed without saving.

❑ **Save as Copy**. Click the **Save As Copy** checkbox in the **Save As** dialog box to save a copy of the assembly document, with a new name or path, *without* replacing the active document. You continue to work in the original assembly document.

❑ With both **Save As** and **Save as Copy**, if any of the components or sub-assemblies have changed since the last **Save** operation, you are notified that they need to be saved also. If you agree to save them, they are saved in their original referenced locations. See **Editing Referenced File Locations** on page 6-12 for information about saving referenced files in new locations.

Verifying Which Part Files Are Used in an Assembly

You can check which part and sub-assembly files an assembly uses. This is useful if you have several versions of a part file.

To list the part files that are used in an assembly, click **File**, **Find References**. The **Search Results** dialog box displays a list of the parts used in the assembly, including the full path names.

Backing Up an Assembly

You can copy all part and sub-assembly files used in an assembly to a new directory. This is an easy way to back up the assembly and all the referenced files.

To back up an assembly:

1 Click **File**, **Find References**.

2 In the **Search Results** dialog box, click **Copy Files**.

3 In the **Choose Directory** dialog box, browse to the destination directory.

4 If you want the files to be stored with same directory structure (in subdirectories under the selected directory), click **Preserve directory structure**.

5 Click **OK**. The assembly file and all the referenced files are copied to the new directory. This operation does not change the open assembly or any of its components; it simply makes a copy of all the files to the selected directory. The active assembly continues to reference the files in their original locations.

The following table lists the effects of saving an assembly in various ways. For this example, assume that these are the active documents:

assembly document	**c:\test\Assem1**
referenced part documents	**c:\test\PartA**
	c:\test\PartB (changed since the last save)

	Saved files	Active assembly after saving	Referenced File Locations
Save	c:\test\Assem1 c:\test\PartB	c:\test\Assem1	c:\test\PartA c:\test\PartB
Save As c:\final\Assem2	c:\final\Assem2 c:\test\PartB	c:\final\Assem2	c:\test\PartA c:\test\PartB
Save As Copy c:\final\Assem2	c:\final\Assem2 c:\test\PartB	c:\test\Assem1	c:\test\PartA c:\test\PartB
Find References, then Copy Files to c:\final	c:\final\Assem1 c:\final\PartA c:\final\PartB	c:\test\Assem1	c:\test\PartA c:\test\PartB

Replacing a Component in an Assembly

An assembly and its components may go through many revisions during the course of a design cycle. Managing the referenced files becomes very important when the referenced documents change.

This is especially true in a multi-user environment. Several users can work on the individual parts and sub-assemblies. Then all the components are brought together and managed in the top-level assembly. As components are added and modified, the references in the top-level assembly need to be updated so that the latest revisions are used.

You could delete the out-of-date components, then insert the new versions. However, deleting a component also deletes its mates, and breaks its external references for in-context features. To restore the assembly to its previous condition, you have to add the mates again, and fix any failed in-context features.

A safer, more efficient way to update the assembly is to *replace* the components as needed. Depending on the differences between the original part and the replacement part, the mates and in-context features may not be affected at all.

There are several ways to replace assembly components.

❑ **Use Reload/Replace.** This is an easy way to replace all instances of a single component, either with an updated version of the same file, or with a new file.

❑ **Edit the component properties.** In addition to replacing a component, editing the component properties allows you to specify a different name for identifying the component in the FeatureManager design tree. This affects only the selected instance.

❑ **Open the assembly with new referenced file locations.** This is an efficient way to replace several components at once, especially in these situations:

• When multiple replacement parts are stored together, in a different directory

• When the files have a common text string in their names that you can search and replace (revision names, for example)

❑ **Save the assembly with new referenced file locations.** Use this method to save the assembly and its components in a new location, and update the referenced file locations at the same time.

This is different from **Save as Copy** (see page 6-9). After this operation, the assembly references the files in the new locations you specify (not the files it referenced when you opened it). This is especially useful when you create in-context features; you can save and reference a copy of the part that includes the in-context features, and leave the original part file unchanged for future use.

To reload or replace an assembly component:

1 Right-mouse click the component you want to reload or replace, and select **Reload**.

2 Click **Open as read-only** if you only need read access, while another user needs write access to the part.

3 Select either **Reload** or **Replace**.

• **Reload** refreshes shared documents. If you open a part file for read-only access and use that part while another user is making changes to the same part, you can reload the part to get the updated version.

• **Replace** lets you replace each open reference to the current file with a reference to the new file instead. Use the **Browse** button to find the new file.

4 Click **OK** to accept the change, or click **Cancel**.

To replace a single instance of a component by editing its properties:

1 In the FeatureManager design tree or the graphics area, right-click the part you want to replace, and select **Component Properties**.

2 Enter a **Component Name** if you want the FeatureManager design tree to list the component by a name that is different from the name of the part file.

If the **Component Name** box is grayed out, click **Tools, Options, External References**, and *deselect* **Update component names when documents are replaced**.

- When checked (the default), the component name in the FeatureManager design tree is the name of the part file. If you replace a component, the name in the tree updates to use the name of the new part file.

- When unchecked, you can specify the component name to be used in the FeatureManager design tree. If you replace a component, the name in the tree does not change.

If your naming convention includes information about the revision, date, etc., this allows you to use a generic name (**housing**), instead of the part file name (**housing_revA3**).

The **Component Name** is used *in the FeatureManager design tree only*; the component's file name does not change. Component names may not include the forward slash (/) or at (@) characters.

3   To locate the replacement file, click **Browse**. Navigate to the new file. If desired, click the **Preview** checkbox to view the part in the **Preview** window.

4   Click **Open**. The **Model Document Path** displays the selected file name.

5   Click **OK** to replace the file.

Editing Referenced File Locations

When you need to replace all the instances of several components, editing the referenced file locations is an efficient method. You can change the references so that the assembly document looks in a different directory to locate the part files. You can also search and replace text strings contained in the file names (for example, a suffix that indicates the revision number).

For the following examples, assume that the assembly document currently references these parts:
c:\in_progress\partA_rev1
c:\in_progress\partB_rev1

You want to update the assembly to reference these parts:
c:\completed\partA_rev2
c:\completed\partB_rev2

To open an assembly with new references:

This operation changes the referenced file locations *as you open* the assembly. Instead of using the referenced file locations that were saved with the assembly, you specify new references now.

1   Click **File, Open**, select the assembly, and click the **References** button.

The **Edit Referenced File Locations** dialog box lists the **Current pathname** (and the same **New pathname**) for the components.

2 Click the referenced file that you want to change, or **Ctrl**-click multiple files, or click **Select All**.

3 To change the directory where the files are found, click **Browse**, then locate the new directory (**completed**, in this example). When you click **OK** in the **Choose Directory** dialog box, the **New pathname** is updated for the selected files.

4 To change the file names, click **Replace**. In the **Find what** and **Replace with** boxes, enter the text strings to find and replace (**rev1** and **rev2**, in this example).

To find occurrences of the string one by one, click **Find Next**, then **Replace** where needed. Otherwise, click **Replace All**.

5 Click **OK**. If any of the files does not exist in the new directory, a red exclamation mark indicates which files were not found, and you can try again.

6 Click **Open**. The assembly opens, using the updated reference locations.

NOTE: You do not have to update the path and the name at the same time.

---

7 Save the assembly. This is necessary to save the new reference locations in the assembly document.

To save an assembly with new references:

This method saves the assembly and its parts in a new location, and updates the referenced file locations at the same time. After this operation, you are working in the newly saved assembly document, with the new references. The original assembly document is closed without saving.

1 Click **File, Save As**, specify the directory and filename for saving the assembly document, then click the **References** button.

2 Select the referenced files that you want to save with new names or paths. To change the path for all the entries on the list, click **Select All**.

3 To edit the path of the selected files, click **Browse,** then navigate to the target directory (**completed**, in the example).

4 To edit the file names of the selected files, click **Replace**, enter the text strings to find and replace (**rev1** to **rev2** in this example), and replace as needed.

NOTE: You can also click-pause-click the file in the **New pathname** list, then type the path to an existing directory, and the new name.

---

5 Click **OK**. If the specified files already exist, you can choose to replace them; if the files do not exist, the assembly and the parts are saved with the new paths and names.

You are now working in the newly saved assembly, which references the newly specified file locations.

When an assembly has in-context features, there are some special considerations for saving referenced file locations. See **Saving Assemblies with In-context Features** on page 7-5.

Preserving Mates with Component Replacement

If the original part and the replacement part are similar, mates used in the original part are applied to the replacement part wherever possible. Preservation of the mates depends on the entities used in the mates being the same. (See **Assembly Mating** on page 6-17 for more information about mates.)

If you replace one part with another part created by deriving or copying, the entities used in the mates are likely to match (depending on the differences between the parts), and the mates will solve as expected.

In this example, the replacement part is an edited copy of the original part. The assembly is able to maintain the concentric mate between the cylindrical faces, and the distance mate between the flat faces.

If the entities used in the mates are not the same, or do not exist, in the replacement part, the mate will have an error. You can either delete the failed mate, or edit the mate to use a different entity.

You can rename the corresponding edges and/or faces on a replacement part to match the edge/face names on the original part to ensure that the mates are preserved.
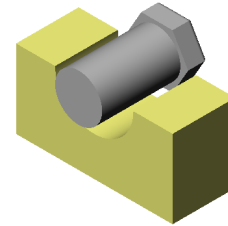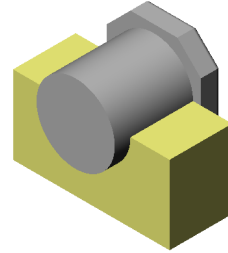
To name the entities used in a mate:

   1   Click **Tools, Options, External References**, and click **Automatically generate names for referenced geometry**. You should enable this option before you add mates if you intend to perform component replacement using the same surface identifiers.

   • When checked, you automatically create surface identifiers (**Face1, Edge2**) in each part when you add a mate; therefore, you need write access to the parts, so that the identifiers can be saved in the part documents.

   • When unchecked (the default), you can mate to parts for which you have read-only access because you use the internal face IDs of the parts. If you do not plan to replace components, leave this option off, especially in a multi-user environment.

   2   If you need to replace components in an assembly whose mates were added when the option was not checked, you can still rename the entities individually. In the original part document, right-click the entity used in the mate, and select **Face Properties** (or **Edge Properties**).

   3   Under **Entity Information**, enter a new **Name**, and click **OK**.

   4   In the replacement part document, right-click the corresponding edge or face, and edit the entity properties, using the same **Name**. Now when you replace the component, the mate can be preserved.

   If the replacement component is derived from the original one, you only need to enter names for the entities that differ between the two parts.

Positioning Components in an Assembly

Once a component is placed in an assembly, you can move it, rotate it, or fix its location. This is useful for rough placement of the components in the assembly. You can then position the components precisely using mating relationships.

As you add mating relationships, you can move the components within the unconstrained degrees of freedom, visualizing the mechanism's behavior.

Fixing the Position of a Component

When working with an assembly, you may want to fix the position of a component so that it cannot move with respect to the assembly origin. By default, the first part in an assembly is fixed; however, you can unfix it at any time.

It is recommended that at least one assembly component is either fixed, or mated to the assembly planes or origin. This gives a frame of reference for all other mates, and may prevent unexpected movement of components when mates are added.

To fix a component:

In the FeatureManager design tree, right-mouse click the component (part or sub-assembly) icon, and select **Fix**.

In the FeatureManager design tree, the prefix **(f)** appears next to the name of the fixed component.

To float (unfix) a component:

In the FeatureManager design tree, right-mouse click the component (part or sub-assembly) icon, and select **Float**.

In the FeatureManager design tree, the prefix changes to reflect the status of the component before its position was fixed.

Moving a Component

When you move a component, it retains any mating relationships (within its own mategroup) with other components. You cannot move a component whose position is fixed or fully defined.

To move a component:

1 Click **Move Component** 🔧 on the Assembly toolbar, or **Tools**, **Component**, **Move**.

2 Drag a component to move it. You can move one component after another; the tool remains active until you click it again, or select another tool.

You can select a component to move in the FeatureManager design tree also. Dragging the mouse anywhere in the graphics area moves the selected component.

NOTE: Clicking **Pan** ⊕ on the View toolbar scrolls the view of the *entire assembly.*

Rotating a Component

There are two ways you can rotate a component: freely around its centerpoint, or around an axis. Only components that have not yet been mated can be freely rotated. A mated component can be rotated around an axis if its mates allow it. When rotating a component around an axis, all mating relationships within the component's own mategroup are maintained.

To rotate a component freely:

1 Click the desired component in the graphics area or the FeatureManager design tree.

2 Click **Tools**, **Component**, **Rotate** or **Rotate Component Around Centerpoint** 🧭 on the Assembly toolbar. You can only use this rotate tool on components that have not been mated.

3 Drag to rotate the component. You can rotate one component after another; the tool remains active until you click it again, or select another tool.

NOTE: Clicking **Rotate View** 🔄 on the View toolbar rotates the view of the *entire assembly.*

To rotate a component around an axis:

1 Click an axis, linear edge, or sketch line around which to rotate the component.

2 Hold the **Ctrl** key and click the component to rotate.

3 Click **Rotate Component Around Axis** 🧭 on the Assembly toolbar.

4 Click and drag to rotate the component around the axis. Dragging the mouse left-to-right rotates about the axis in one direction; right-to-left rotates in the opposite direction.

Measuring Distances Between Components

You can measure the size of items, and the distance and angle between lines, points, surfaces and planes in sketches, part or assembly models, or drawings. See **Measuring Sizes and Distances** on page 5-8 for more information.

Assembly Mating

Mating Relationships

Mating relationships let you precisely position the components with respect to each other in an assembly. They let you define how the components move/rotate with respect to other parts. By adding mating relationships successively, you can move the components into the desired positions.

Mating creates geometric relationships, such as coincident, perpendicular, tangent, and so on. Each mating relationship is valid for specific combinations of geometry. The following table shows the mating relationships that are supported between the various types of geometry.

	Plane	Cylinder	Line	Cone	Point
Plane (planar face or plane)	coincident distance parallel perpendicular angle				
Cylinder (cylindrical face)	tangent	tangent concentric			
Line (linear edge, axis, or sketch line)	coincident parallel perpendicular distance	coincident tangent concentric	coincident parallel perpendicular distance angle		
Cone	---	concentric	concentric	concentric	
Point (vertex or sketchpoint)	coincident distance	coincident concentric	coincident distance	concentric	coincident distance

Mategroups

When you create a new assembly, an empty mategroup is automatically created. This mategroup (which appears with a default name of **MateGroup1**) is shown in the FeatureManager design tree with a double paperclip icon.

When you create a mating relationship, the mating relationship icon (a single paperclip) and name appears in the FeatureManager design tree under the **MateGroup<n>** mategroup. Each assembly has at least one mategroup. Each mategroup consists of mates that are solved together.

For example, if you have two parts in an
assembly, and add a concentric and a
coincident mate between the parts,
MateGroup1 will contain mates called
Concentric1 (*part1, part2*) and
Coincident1 (*part1, part2*).

```
└─ MateGroup1
      └─ Concentric1 (bracket<1>,Yoke_male<1>)
      └─ Coincident1 (bracket<1>,Yoke_male<1>)
```
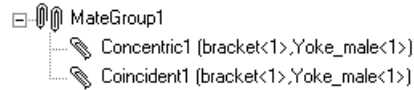
Assemblies can have multiple mategroups. Additional mategroups are automatically
added when needed. This occurs when performing certain operations, such as adding a
mate to a time-dependent feature in the assembly. See **Time-Dependent Features** on page
7-10 for more information.

You can rename mategroups and mates. Click-pause-click on the name in the
FeatureManager design tree, type the new name, and press **Enter**.

Creating a Mating Relationship

To mate components in an assembly:

1   Click **Mate** on the Assembly toolbar or select
    Insert, **Mate**.

2   Select the desired faces, edges, vertices, or reference
    planes on the components. The entities for each
    component are listed in the **Items Selected** box.

    To make selection of the desired entity type easier,
    use the **Selection Filter**.

    > **NOTE:** For components with drafted or curved
    > faces, it is sometimes easiest to create
    > mating relationships between the reference
    > planes of the components and/or the
    > assembly.
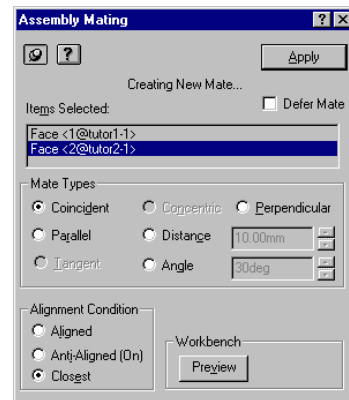
In the **Assembly Mating** dialog box, you can click the pushpin to keep the dialog
box open. This allows you to apply multiple mates before closing the dialog box.

3   Select the desired **Mate Type**. (Only the mating types that are valid for the selected
    entities are available.) If you select **Distance** or **Angle**, you must also specify the
    value.

4   Select an **Alignment Condition**.

    • **Anti-aligned**. The components are on *opposite* sides of a plane parallel to the
      selected faces.

    • **Aligned**. The components are on the *same* side of a plane parallel to the selected
      faces.

    • **Closest**. The components are either aligned or anti-aligned, depending on which
      condition can be satisfied with the least movement.

5 In the **Workbench** area of the dialog box, click **Preview** to see what the assembly will look like with the mate performed.

6 Click **Defer Mate** if you want to define several mates, then solve them all at once. Mates defined while this option is checked are solved when you either close the dialog box or uncheck the option.

7 If you are satisfied with the mate, click **Apply**. If you are not satisfied with the mate, click **Undo** or click a different alignment option and **Preview** again.

8 Click **Apply** when you are satisfied with the mate. When you close the **Assembly Mating** dialog box, any deferred mates are solved, and the components move into the specified positions.

Modifying a Mating Relationship

You can change these characteristics of a mating relationship:
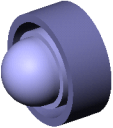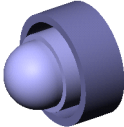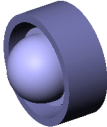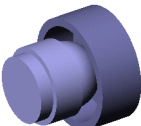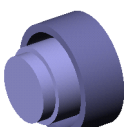
- **Alignment (aligned** or **anti-aligned)** on mates which imply a direction
- **Dimension direction** to flip (reverse) the direction in which a distance is measured
- **Angle** between the mated components for angle mates
- **Distance** between the mated components for distance mates

---

NOTE: A **Distance** mate with a value of zero is more flexible than a **Coincident** mate. In some cases (if the assembly does not explode as expected, or if other mates do not solve), this may be a better choice.

You can link the value of a **Distance** or **Angle** mate to a sketch dimension.

The examples in the following table show the effect of the changing the alignment and flipping the direction for a distance mate. All the examples use the same face selections.

Selected faces
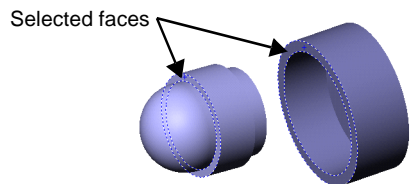
Offset Distance	Component Alignment	Flip Dimension To Other Side	Example
0 (coincident)	aligned	off	
20mm	aligned	off	
20mm	aligned	on	
20mm	anti-aligned	on	
20mm	anti-aligned	off	

To modify a mating relationship:

1 Right-click the desired mating relationship in the FeatureManager design tree to select it. The related geometry in the graphics display is highlighted.

2 Select **Edit Definition** from the right mouse menu.

3 In the **Assembly Mating** dialog box, change the desired options.

4 To replace one component with another, click the entity to replace in the **Items Selected** box, press the **Delete** key, then select the new entity.

5 Click **Apply** to make the change.

> **NOTE:** To change only the dimension value of a distance or angle mate, double-click the mate in the FeatureManager design tree, then double-click the dimension to edit. Enter the new value, then click **Rebuild** ⬛.

Automatic Mating

For some types of mates, you can create mating relationships automatically. You can inference the geometry of existing components as you drag and drop new components into the assembly.

Geometry-Based Mates

You can automatically create some types of mates by dragging parts into assemblies using certain model geometry.

To create mates automatically, based on model geometry:

1   Drag a model into the graphics area of the assembly window, using one of the following entities: a linear or circular edge, a temporary axis, a vertex, or a planar or conical face.

When the cursor is over an entity of another component that is a potential mate partner, the cursor changes to indicate the mate(s) that will result if the component is dropped at this location. A preview of the part snaps into place.

If the preview indicates that you need to flip the alignment condition, press the **Tab** key to toggle the alignment (aligned/anti-aligned).

2   Drop the component. The mate(s) are added automatically.

The following types of automatic mates are supported:

mating entities	type of mate	cursor
linear edge to linear edge	coincident	
planar face to planar face	coincident	
vertex to vertex	coincident	
axis to axis conical face to axis axis to conical face conical face to conical face	concentric	
circular edge to circular edge	coincident (planar faces) and concentric (conical faces)	

Feature-Based Mates

You can also add mates automatically between conical features that have a "peg-in-hole" relationship. One must be a base or boss, and the other must be a hole or a cut. The features may be extruded or revolved, and a cross-section of the feature must be a complete circle.

The features must have a matching set of faces. The conical faces must both be of the same type (either a cone or a cylinder, you may not use one of each type). There must also be a planar face adjacent to the conical face.

A **Concentric** mate is added between the conical faces. The way the part is designed, and the feature that you select, determine the entities used for the **Coincident** mate between the planar faces. For example:

- If you mate the cylindrical base of a screw to the cylindrical face of a countersunk hole, the planar face at the end of the screw mates to the planar face at the bottom of the hole.

- If you mate the conical boss of the screw to the conical face of the hole, the planar face on the head of the screw mates to the planar face at the top of the hole.

- If you create the whole screw as a single revolved base, then mate it to either face of the hole, the top planar faces are mated. When the set of faces matches this way, the coincident mate is applied where the faces meet at the largest circular edge.
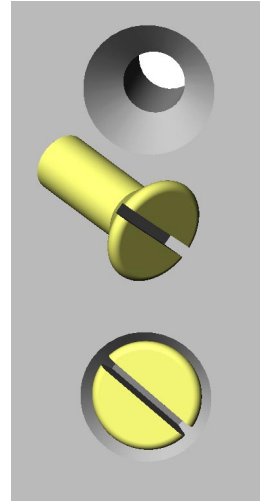
To create mates automatically, based on features:

1 In the FeatureManager design tree of a part document, select a feature with a cylindrical or conical face. Drag the feature name into an assembly graphics window.

When the cursor is over another cylindrical or conical face, it changes to indicate the mates that will result if the component is dropped at this location.

A preview of the part snaps into place. If the preview indicates that you need to flip the alignment condition of the planar faces, press the **Tab** key to toggle the alignment (aligned/anti-aligned).

2 Drop the component. Two mates are applied: a **Concentric** mate between the cylindrical or conical faces, and a **Coincident** mate between the planar faces that are adjacent to the conical faces.

Pattern-Based Mates

In some special cases, you can create as many as three automatic mates at once. Each component must have a circular pattern of cylindrical holes (or bosses) on a planar face with a circular edge. Drag the component into the assembly using the circular edge.

A **Concentric** mate and a **Coincident** mate are added, consistent with the geometry-based mates described earlier. Then, if possible, a third **Concentric** mate is added, between a pattern instance on the part that you are dragging and one on the part that is already in the assembly.

Press the **Tab** key to rotate the part that you are dragging, to align it with the desired pattern instance.

Components to mate   Mate circular edges of flanges   Use the **Tab** key to rotate
                                                                    through pattern instances
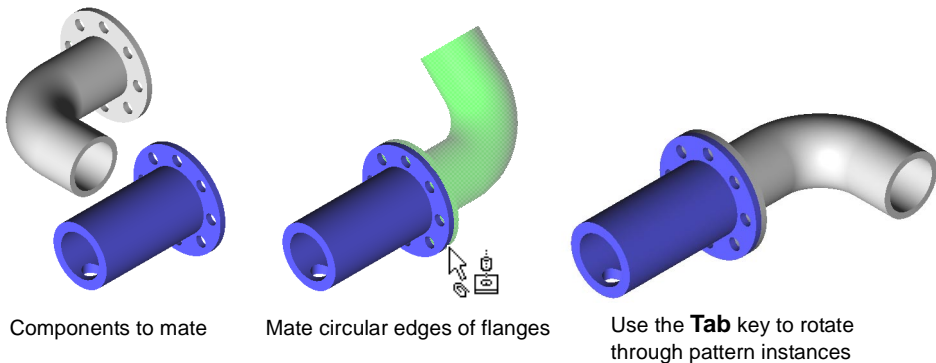
Defining a Mate Reference

A mate reference identifies which entity of a part is to be used for automatic mating.

When you insert a part with a mate reference by dragging its part icon from the top of the FeatureManager design tree, the software can identify potential mate partners for the specified entity.

As you move the cursor in the assembly window, the cursor changes and the preview snaps into place when a potential mate partner is found.

To define a mate reference:

1 In a part document, click **Tools, Mate Reference**.

2 Select an entity to use for automatic mating, and click **OK**. You can use a linear or circular edge, an axis, a vertex, or planar or conical face.

   A part can have only one mate reference.

   To select a different entity, right-mouse click the **MateReference** feature in the FeatureManager design tree, and select **Edit Definition**. Click a new entity, and click **OK**.

Deleting a Mating Relationship

You can delete mating relationships when necessary.

To delete a mating relationship:

1 Click the desired mating relationship in the FeatureManager design tree.

2 Press the **Delete** key, or click **Edit**, **Delete**.

3 Click **Yes** at the prompt to confirm the deletion.

Suppressing a Mating Relationship

You can suppress mating relationships to prevent them from being solved. This allows you to experiment with different types of mates without over defining the assembly. When you suppress a mate, it is suppressed only in the *active* configuration of the assembly. See **Using Assembly Configurations** on page 6-35 for more information.

To suppress a mating relationship:

1 Right-click the mating relationship in the FeatureManager design tree, and select **Properties**.

2 Click **Suppressed**, and click **OK**.

To unsuppress the mate again, repeat the process, and *deselect* **Suppressed**.

You can also select one or more mates (hold the **Ctrl** key to select more than one), and click **Edit, Suppress** (or **Unsuppress**).

Errors in Solving Mates

If changes in a component remove an entity (face, edge, etc.) that is involved in a mate, the mate cannot be solved. The mate is not deleted, however. The mate is suppressed, and is displayed in the FeatureManager design tree with an error symbol. Suppressed mates are not considered when the remaining mates are solved.

Exploding an Assembly View

For manufacturing purposes, it is often useful to separate the components of an assembly to visually analyze their relationships. Exploding the view of an assembly allows you to look at it with the components separated.

An *exploded view* consists of one or more *explode steps*. An exploded view is stored with the assembly configuration in which it is created.

Using AutoExplode

The AutoExplode function is useful when you need to create an exploded view of an assembly that has few components.

To create an exploded view using AutoExplode:

1 Click **Insert, Exploded View**.

2 In the **Assembly Explode** dialog box, click **AutoExplode**.

The assembly explodes, and an **ExplView** feature is added to the active configuration.

> **NOTE:** If the assembly does not explode as expected, try changing **Coincident** mates between faces to **Distance** mates with a value of zero. AutoExplode can always explode **Distance** mates; **Coincident** mates do not always explode automatically.

Creating an Exploded View

To explode an assembly:

1 Click **Insert, Exploded View.**

The **Assembly Exploder** dialog box appears. Examine the **Step Editing Tools**:

New - create a new explode step

Previous Step - edit previous explode step

Next Step - edit next explode step

Undo - undo all changes since the last **Apply**

Delete - delete current explode step

Apply - apply/update current explode step

2   Click **New** ![icon]. The **Assembly Exploder** dialog box expands.

The **Assembly Exploder** dialog box has automatic focus. Each area of the dialog box is automatically activated in the correct order needed to create an explode step. You do not need to click in any area of the dialog box to activate it unless you want to select and delete an incorrect entry.
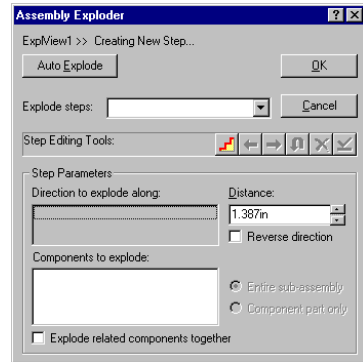
3   Click a component edge or face that is parallel to the direction you want to explode.

A preview arrow appears and a description of what you have selected appears in the **Direction to explode along** box.

If the preview arrow is pointing in the wrong direction, click **Reverse direction**.

4   Click the component you want to explode, either in the FeatureManager design tree or in the graphics area.

The name of the component appears in the **Components to explode** box.

Direction to explode along

Component to explode
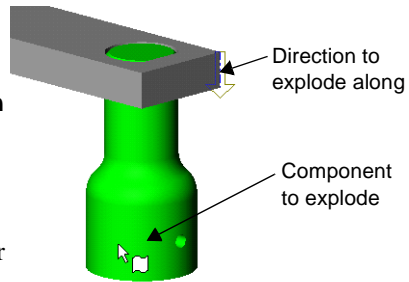
5   Click **Apply** ![icon].

The component explodes from the assembly. Note the drag handles.

6   Drag the arrow-shaped handle to position the component as desired.

7   Click **Apply** ![icon] to confirm this step.

If you prefer to explode a component a precise distance, enter the distance in the **Distance** box, then click **Apply**.

8   If you are satisfied with this exploded view, click **OK**.

drag handle
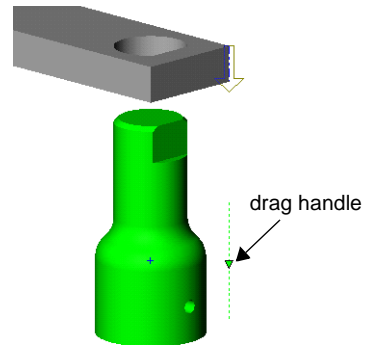
If you want to add more explode steps to the exploded view, click **New**, repeat Steps 3 through 7, then click **OK** to finish the exploded view.

Remember to click **Apply** after defining each step.

9 To collapse the assembly, right-click anywhere and select **Collapse**.
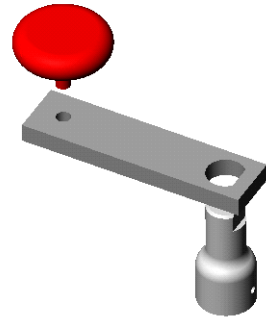
Exploding and Collapsing an Exploded View

To explode and collapse an exploded view:

1 Click the Configuration tab at the bottom of the window.

2 Click the ⊞ beside the desired configuration, and beside the **ExplView** feature to see the **Explode Steps**.

3 Double-click the **ExplView** feature. The assembly explodes.

4 To collapse the exploded view, right-click anywhere and select **Collapse**.
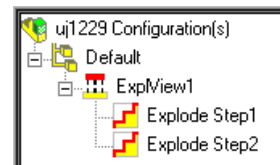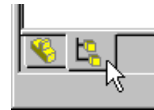
NOTE: To change the name of an exploded view or explode step, click-pause-click the name, type the new name and press **Enter**.

Editing an Exploded View

To edit an exploded view (method 1):

1 In the Configuration Manager, expand the desired configuration, and expand the exploded view to see the steps,

– or –

Right-click the **ExplView** feature for the desired configuration, and select **Edit Definition**.

2 Click the **Explode Step** feature you want to edit, or use the **Next Step** and **Previous Step** buttons to examine and edit each step in turn.

As you select each step, the drag handle appears and the components involved are highlighted (**Shaded** view mode) or enclosed in bounding boxes (**Wireframe, Hidden In Gray,** or **Hidden Lines Removed** view mode).

3 Edit the explode step until the component is positioned as desired.

4 Click **Apply** after editing each step.

5 Repeat for each step as needed, then click **OK** to finish the view.

To edit an exploded view (method 2):

1 Double-click the **ExplView** feature to explode the view.

2 Right-click the component you want to reposition in the graphics area, and select **Show Explode Steps**.

The drag handle(s) for the explode step(s) on the selected component are displayed.

3 Drag the component by the green drag handle to the new position.

4 Repeat as needed for each component, then collapse the assembly.

To edit an exploded view (method 3):

For complex assemblies, you may find it easier to edit the explode steps for a component by selecting the component from the FeatureManager design tree.

1 Double-click the **ExplView** feature to explode the view.

2 Switch from the Configuration Manager to the FeatureManager by clicking the FeatureManager tab at the bottom of the window.

3 Right-click the component to reposition, and select **Show Explode Steps**.

4 Proceed as described above.

Detecting Interference Between Components

In a complex assembly, it may be difficult to visually determine whether components interfere with each other. You can determine the interference between components and visually browse the resulting interference volumes.

To check for interference between components in an assembly:

1 Click **Tools**, **Interference Detection**.

2 Select two or more components in the assembly, or click the assembly icon (top level or sub-assembly) in the FeatureManager design tree.

3 Click **Treat coincidence as interference** if you want coincident entities (overlapping faces, edges, or vertices) to be reported as interferences. Otherwise, overlapping volumes are considered.

4 Click **Check**.

If there is interference, the **Interference Results** box lists the interference occurrences (one occurrence is reported for each pair of interfering components). When you click an item in the list, the related interference volume is highlighted in the graphics display area, and the names of the components involved are listed.

The volume of the interference is reported in the form of length, width, and height of the bounding box around the area of interference. These numbers are displayed on the graphic display of the components.

5 With the dialog box still open, you can select other components to check for interference. Right-click in the graphics area and select **Clear Selections**, then select new components for checking, and click **Check**.

6 Click **OK** to dismiss the dialog box. When the dialog box is dismissed, the interference volumes are dismissed also.
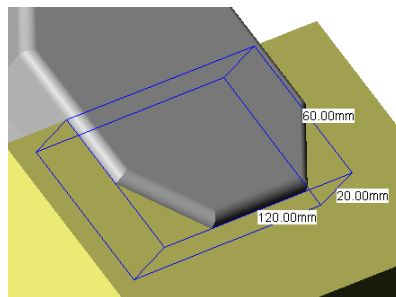
> **NOTE:** If detecting interference is important in your design work, check for interference each time you move or rotate a component.

Simplifying Large Assemblies

Large assemblies can be complex, consisting of hundreds of components. You can simplify a complex assembly by toggling the *visibility* of the components, and by changing the *suppression state* of components.

There are several reasons to simplify an assembly:

❑ To improve system performance, and to reduce rebuild times, especially with very large assemblies.

❑ To create simplified views of the assembly that include certain components, and exclude others.

❑ To create design variations of the assembly, with different combinations of components, and different configurations of the components themselves.

Toggling the Visibility

You can toggle the display of assembly components. Turning off the display of a component temporarily removes it from view, allowing you to work with underlying components more easily.

Hiding or showing a component affects *only the visibility* of the component in the active assembly configuration. Hidden components have the same accessibility and behaviors as shown components in the same suppression state (see the next section for information about suppression states).

Toggling the visibility does not affect the rebuild or evaluation speed. Display performance improves however.

To toggle the visibility of a component (part or sub-assembly):

1 In the FeatureManager design tree or in the graphics area, right-click the component, and select **Component Properties**.

To change several components at once, hold the **Ctrl** key as you select the components, then right-click and select **Component Properties**. If you select the components from the FeatureManager design tree, you must right-click in the graphics area; otherwise, only the last item you clicked remains selected.

2 In the **Component Properties** dialog box, under **Visibility**, select **Hide Model**, and click **OK**. The selected item(s) are removed from the display.

To redisplay the component, select the component(s) in the FeatureManager design tree, then edit the **Component Properties**.

Other ways to toggle the visibility:

❑ Right-mouse click the component, and select **Hide Component** or **Show Component**.

❑ Use the **Hide/Show Component** ⬚ icon on the Assembly toolbar to toggle the hidden and shown states of a component.

❑ Select the component, and click **Edit, Hide Component** (or **Show Component**).

Suppression States

Depending on the scope of the work you plan to do at any given time, you can specify an appropriate suppression state for the components. This can reduce the amount of data that is loaded and evaluated as you work. The assembly displays and rebuilds faster, and you make more efficient use of your system resources.

There are two suppression states for assembly components: *resolved* and *suppressed*.

Resolved

Resolved is the normal state for assembly components. A fully resolved component is fully loaded in memory, fully functional, and fully accessible. All its model data is available, so its entities can be selected, referenced, edited, used in mates, etc.

Suppressed

You can use the suppressed state to remove a component temporarily from the assembly, without deleting it. It is not loaded in memory, and it is no longer a functional member of the assembly. You cannot see a suppressed component, or select any of its entities.

You can use this state to create multiple design variations (configurations) within a single assembly document. See **Using Assembly Configurations** on page 6-35 for more information.

A suppressed component is removed from memory, so rebuild speed and display performance are improved. Because of the reduced complexity, evaluation of remaining components occurs faster.

Suppressing components also helps assemblies load quickly. However, mates that involve the suppressed components are suppressed also. As a result, the positions of components in the assembly can become under defined. In-context features that reference suppressed components may also be affected. When you restore a suppressed component to the fully resolved state, conflicts can occur. Therefore, use the suppressed state carefully when modeling.

Lightweight Parts

You can load an assembly with its active parts *fully resolved* or *lightweight*. When a part is fully resolved, all its model data is loaded in memory. When a part is lightweight, only a subset of its model data is loaded in memory. The remaining model data is loaded on an as-needed basis.

Advantages of Lightweight Parts

You can improve performance of large assemblies significantly by using lightweight parts. Loading an assembly with lightweight parts is faster than loading the same assembly with fully resolved parts.

Lightweight parts are efficient because the full model data for the parts is loaded only as it is needed. Only parts that you select, and parts that are affected by changes that you make in the current editing session, become fully resolved.

Assemblies with lightweight parts rebuild faster because less data is evaluated. Mates on a lightweight part are solved, and you can edit existing mates. In-context features that reference lightweight parts are solved.

To enable lightweight loading of parts:

1 Click **Tools, Options, Performance**.

2 Under **Assemblies**, click **Automatically load parts lightweight**.

NOTE: You can only change this option when no assemblies, or drawings of assemblies, are open.

When this option is checked, all parts are loaded lightweight when you open an assembly. Sub-assemblies themselves are not loaded lightweight, but the individual parts that they contain are lightweight.

When the cursor passes over a lightweight part in the graphics area, a feather cursor is displayed, and the body is surrounded by a bounding box.

Loading Additional Model Data

When an operation requires model data that is not loaded for a lightweight part, the software fully resolves the part for you, to gain access to the additional data. This happens automatically when you do any of the following things:

- Click a part in the graphics area. When you click, the part becomes fully resolved, and the entity (face, edge, etc.) where you clicked is selected and highlighted.

- Right-mouse click a part or feature, and select **Edit Part**, **Edit Sketch** or **Edit Definition**. The part becomes fully resolved, and you are in **Edit Part** mode.

- Rebuild an assembly that has in-context features, equations, or assembly features (cuts or holes) that affect a lightweight part.

- Create a drawing view of an assembly that contains lightweight parts. All lightweight parts become fully resolved in the assembly when the view is added to the drawing sheet.

Certain global operations require that many parts be fully resolved at once. This includes **Mass Properties, Interference Detection**, displaying non-shaded view modes or section views, and exporting to other file types (IGES, SAT, etc.).

To specify what you want to happen when you request one of these operations, click
Tools, Options, Performance, then choose one of these options under **Resolve lightweight parts**:

- **Always**. All lightweight parts are fully resolved for you, and they are included in the operation.

- **Prompt** (default). A dialog box appears, asking if you want the software to resolve the lightweight parts for you. Click **Yes** to fully resolve the parts and continue, or click **Cancel**. If you also click **Always resolve** (before you click **Yes** or **Cancel**), the option is set to **Always**.

NOTE: When a lightweight part becomes fully resolved, all instances of the part, in any configuration, in any open assembly documents, become fully resolved.

Any part that is resolved during the session stays resolved until you close the assembly. All the parts are loaded lightweight again the next time you open the assembly, as long as the option is still enabled.

Changing the Suppression State

You can change the suppression state of any component (part or sub-assembly) to the **Resolved** or **Suppressed** state at any time.

To change the suppression state of components:

1 In the FeatureManager design tree or the graphics area, right-click the desired component, and select **Component Properties**.

   To change several components at once, hold the **Ctrl** key as you select the components, then right-click and select **Component Properties**. If you select the components from the FeatureManager design tree, you must right-click in the graphics area; otherwise, only the last item you clicked remains selected.

2 In the **Component Properties** dialog box, under **Suppression State**, select the desired state, and click **OK**.

Other ways to change the suppression state:

❑ Click a component, then either click **Change Suppression State** 🖼 on the Assembly toolbar, or click **Edit, Suppress** (or **Unsuppress**).

❑ To fully resolve (unsuppress) a suppressed component, right-click the suppressed component, and select **Set to Resolved**.

❑ To fully resolve all lightweight parts, right-click the assembly icon at the top of the FeatureManager design tree, and select **Set Lightweight to Resolved**.

Comparison of Performance and Behavior

	Resolved	Lightweight	Suppressed	Hidden
loaded in memory	yes	partially	no	yes
visible	yes	yes [2]	no	no
features available in FeatureManager design tree	yes	yes [3]	no	no
faces and edges accessible for adding mates	yes	yes [3]	no	no
mates solved	yes	yes	no	yes
in-context features solved	yes [1]	yes [4]	no	yes
assembly features solved	yes	yes [4]	no	yes
considered in global operations	yes	on demand [2]	no	yes
may be edited in context	yes	yes [3]	no	no
load and rebuild speed	normal	faster	faster	normal
display speed	normal	normal	faster	faster

[1] In-context features on *resolved* parts that *reference* lightweight parts update automatically.

[2] Global operations include Mass Properties, Interference Detection, non-shaded views, section views, and exporting to other file formats. When lightweight parts are present, the system prompts you to either resolve them, or cancel the operation.

[3] Lightweight parts are resolved automatically when selected for this operation.

[4] When in-context features, assembly features, or equations affect lightweight parts, the lightweight parts are resolved when the assembly is rebuilt.

Part1<1>   **Resolved**. This is the only suppression state where the component can be expanded (double-click the name, or click the ⊞).

Part1<1>   **Lightweight**. A feather overlay appears on the icon of a lightweight part.

Part1<1>   **Suppressed**. A gray icon indicates that the component is not in use in the active configuration.

Part1<1>   **Hidden**. A white icon indicates that the component is active, but invisible.

Using Assembly Configurations

Configurations allow you to create multiple variations of an assembly within a single assembly document. In an assembly configuration, you can change the following items, in any combination:

❑ Components – suppression state (**Resolved** or **Suppressed**), visibility, referenced configurations of parts and sub-assemblies

❑ Mates – suppression, dimensions for distance or angle mates

❑ Assembly features – suppression, dimensions (see **Working with Assembly Features** on page 7-16)

❑ Component patterns – suppression, dimensions (spacing, instance counts)

❑ Assembly layout sketches – suppression, dimensions

Creating an Assembly Configuration

When you create a new assembly configuration, you specify its properties, then modify the content of the assembly to create the necessary variations in the new, active configuration. Configuration properties include behavior with respect to other configurations, behavior of its sub-assemblies, and display options in a bill of materials.

To create a configuration:

1 Click the Configuration tab at the lower left corner of the window.

2 In the Configuration Manager, right-mouse click the assembly name and select **Add Configuration**. The **Add Configuration** dialog box appears.

3 Enter a **Configuration Name**. Configuration names must not be longer than 30 characters, or contain these special characters: slashes (\ or /), colon (:), asterisk (*), question mark (?), less than (<), greater than (>), pipe (|).

4 Add **Comments** if desired.

5 Select **Properties for newly inserted items**. By default, newly added components are resolved and shown in the active configuration. Newly added mates and other features that belong to the assembly (sketches, reference geometry, etc.) are unsuppressed in the active configuration. These options control what happens when new items are added to *another* configuration, then this configuration is activated again.

Suppress features and mates. When checked, new mates and features added to *other* configurations are suppressed in *this* configuration. Otherwise, new mates and features are unsuppressed in this configuration also.

Suppress component models. When checked, new components added to *other* configurations are suppressed in *this* configuration. Otherwise, new components are resolved in this configuration also.

Hide component models. When checked, new components added to *other* configurations are hidden in *this* configuration. Otherwise, new components are shown in this configuration also.

6   Select the way properties are applied to sub-assemblies:

- Click **Apply properties to sub-assembly root only** to apply the properties selected in Step 5 to only the top sub-assembly and part components. All sub-assembly child components inherit the properties specified in the configuration of the sub-assembly.

- Click **Apply properties to ALL sub-assembly components** to apply the properties selected in Step 5 to all added components. For example, if you want all components of newly added sub-assemblies to be suppressed, regardless of the properties specified in the sub-assembly configuration, choose this option.

7   Click **Advanced** to specify how the assembly configuration is listed in a bill of materials, when it is used as a sub-assembly of a higher level assembly.

Under **Part No. displayed when used in a Bill of Materials**, select one of the following:

- **Use the document name.** Specifies that the assembly name is used.

- **Use name below.** When selected, a text box opens, containing a name to be used in the bill of materials. The configuration name is given by default. Alternatively, you may enter a name (such as a part number) of you own choosing.

Click **Don't show child components in BOM when used as sub-assembly** if you *never* want this configuration to be expanded in the bill of materials, when it is used as a sub-assembly. When unchecked, the child components may be listed in the bill of materials when certain options are selected. See **Bill of Materials** on page 9-29.

8   Click **Custom** to add custom properties to the configuration. You can link custom properties to notes, display them in a bill of materials, or use them as selection criteria. See **Custom Properties** on page 5-25 for more information.

9   Click **OK**. The new configuration name appears in the design tree.

10   Click the FeatureManager tab to return to the FeatureManager view.

11   Make the necessary changes for the variations in the active configuration.
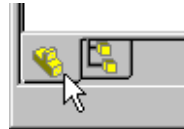
- Change the suppression state, visibility, or referenced configuration of the components.

- Change the dimensions of distance or angle mates, or suppress unwanted mates.

- Modify assembly features. When you change dimensions, the **Modify** dialog box lets you specify whether to apply the change to only **This configuration** or to **All configurations**.

12   **Save** the assembly.

Viewing an Assembly Configuration

To switch to a different configuration:

1 Click the Configuration tab.

2 Either double-click the name of the configuration you want to view, or right-mouse click the name and select **Show Configuration**.

A message appears, notifying you that some configurations may take time to display. Click **OK** to continue; click **Cancel** to abort the action. If you do not want the message to display again, click **Don't ask for confirmation again during this session.**

The named configuration becomes the active configuration, and the display updates to reflect the differences.

Editing an Assembly Configuration

To edit an assembly configuration:

1 View the desired configuration.

2 Change to the FeatureManager view, and change the suppression, visibility, component configurations, etc. as needed.

To edit the configuration properties:

1 Right-click the configuration name, and select **Properties**. The **Configuration Properties** dialog box appears.

2 Edit the name, comments, and properties as desired.

3 Click **OK**.

To change only the name of the configuration, click-pause-click the name in the Configuration Manager, type the new name, and press **Enter**.

Deleting an Assembly Configuration

To delete a configuration:

1 In the Configuration Manager, click the name of the configuration you want to delete.

2 Press the **Delete** key (or click **Edit, Delete**), and click **Yes** to confirm.

> **NOTE:** You cannot delete the active configuration.

Design Tables in Assemblies

A design table allows you to build multiple configurations of assemblies by specifying parameters in an *embedded* Microsoft Excel spreadsheet. The design table is saved in the assembly document and is *not linked* to the original Excel file. Changes you make in the assembly are not reflected in the original Excel file.

To use design tables, you must have Microsoft Excel 97 installed on your computer.

You can control the following parameters in an assembly design table:

- components – suppression state, visibility, referenced configuration
- assembly features – dimensions, suppression
- mates – dimensions of distance and angle mates, suppression
- configuration properties – part number and display in a bill of materials (when used as a sub-assembly), comments, custom properties

To create a design table:

1   Open Microsoft Excel and create a spreadsheet.

> **NOTE:** This procedure describes entering the data in Excel before inserting the table. You can also insert a partially empty spreadsheet and complete editing it in SolidWorks.

2   In the first *column* (row header cells **A2, A3,** etc.), enter the names of the design configurations that you want to create. Configuration names must not be longer than 30 characters, or contain these special characters: slashes (\ or /), colon (:), asterisk (*), question mark (?), less than (<), greater than (>), pipe (|). Leave cell **A1** *blank*.

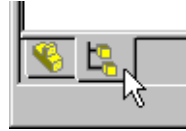3   In the first *row*, (column header cells **B1, C1,** etc.), enter the parameters that you want to control.

	A	B	C
1		parameter1	parameter2
2	**configuration1**	value1	value3
3	**configuration2**	value2	value4

4   Enter the parameter values in the spreadsheet cells. See **Specifying Assembly Parameters** on page 6-39 for detailed information.

5   Save the spreadsheet.

6   Open the assembly where you want to use the design table.

7   Click **Insert, Design Table**. In the **Open** dialog box, navigate to the spreadsheet file and click **Open**.

The spreadsheet appears in the assembly document, the Excel menus and toolbars are displayed, and a message is displayed, indicating the names of any new configurations that were generated by the design table.

8 Edit the design table, if necessary. When you are finished editing, click anywhere outside the table to close it. The SolidWorks menus and toolbars reappear.

To display the configurations added by the design table, click the Configuration tab at the bottom of the window. Double-click the name of the configuration, or right-click the name and select **Show Configuration**.

To edit a design table:

1 Click **Edit, Design Table**. The table appears, and the Excel menus and toolbars are displayed.

2 Edit the table as needed. You can change the parameter values in the cells, add rows for additional configurations, or add columns to control additional parameters.

3 Click anywhere outside of the design table to close it. The configurations update to reflect your changes.

NOTE: Changes you make in the assembly (not in the table) are not reflected in the original Excel file, or in the embedded file. The next time that the design table is evaluated, the configurations are updated to reflect the data in the table.

For example, assume that **D1@Sketch1** is driven by the design table. If you modify **D1@Sketch1** outside of the table (by editing the sketch, in one of the configurations created by the table), *the table data does not change*. Furthermore, the value contained in the corresponding table cell is restored to the configuration the next time that the table is evaluated, and your change is undone.

To delete a design table:

1 Click **Edit, Delete Design Table**.

2 Confirm the deletion and click **OK**.

Deleting the design table *does not* delete the configurations created by it.

Specifying Assembly Parameters

When used in an assembly design table, the names of dimensions, features, components, and configurations must match the name in the model. To ensure an exact match, you can copy and paste the name from the **Properties** dialog box for the selected item.

To copy and paste a feature or dimension name:

1 Right-mouse click the item (dimension, feature, or configuration), and select **Properties**. For a component, select **Component Properties** from the right-mouse menu.

2 In the **Properties** dialog box, copy (**Ctrl+C**) the item name (for dimensions and components, be sure to copy the **Full name**).

3 Paste (**Ctrl+V**) the name into the appropriate column header cell.

Some parameter specifications that appear in the column header cells include a *keyword*, such as **$CONFIGURATION**, or **$COMMENT**. The parameter keywords are *not* case-sensitive.

Some parameters (suppression state, visibility, or referenced configuration of a component) also include a component name and instance number. These parameters use this syntax:

>   **$*KEYWORD*@*component*<*instance*>**

where ***component*** is the name of the component, and ***instance*** is the instance number (an integer), in angle brackets.

For example, the column labelled **$CONFIGURATION@gear<2>** controls the referenced configuration of the second instance of the component named **gear**.

You can specify multiple instances of a component in a single column header cell. Use the following syntax for the instance numbers:

<*>	all instances
<1-4>	range of instances
<1,3,6>	list of non-consecutive instances, separated by commas
<1,3-6,8>	combination, separated by commas

Values that are not item names (**Yes**, **Suppress**, etc.) are *not* case-sensitive. The syntax and legal values for each parameter type are described below.

This is an example of a design table that creates three configurations, adds a comment for each one, suppresses a feature, and controls a dimension.

	A	B	C	D
1		$COMMENT	$STATE@Cut-Extrude1	D3@Sketch2
2	rev1	large, no cuts	suppressed	125
3	rev2	medium	unsuppressed	100
4	rev3	small	unsuppressed	75

NOTE: You may not include any rows or columns with blank headers in the middle of the design table. The software stops evaluating the data when it reaches a row or column that has an empty header cell. Data that appears in or after a row or column with an empty header is ignored.

Dimensions

You can use a design table to control dimensions that belong to assembly features. This includes mates (angle, distance), assembly feature cuts and holes, and component patterns (spacing, instance count). You cannot control the dimensions of a component model contained in the assembly.

A dimension name is in the form:

Dimension@Feature or **Dimension@Sketch<n>**

For example, the full name for the dimension of the first **Distance** mate is **D1@Distance1**. You can also use the **Properties** command to assign more meaningful names to dimensions. See **Dimension Properties** on page 9-11.

In the table body cells, enter the value for the dimension. If a cell is left blank, the default value is zero; if zero is not a legal value for this dimension, an error message is displayed.

Component configuration

The column header for controlling component configuration uses this syntax:

$CONFIGURATION@component<instance>

In the table body cells, enter the name of the desired configuration. Blank cells are not allowed in this column.

Visibility

The column header for controlling component visibility uses this syntax:

$SHOW@component<instance>

In the table body cells, to show the model, enter **YES** (or **Y**). To hide the model, enter **NO** (or **N**). If a cell is left blank, the default is **No** (do not show the model).

Component suppression state

The column header for controlling component suppression state uses this syntax:

$STATE@component<instance>

In the table body cells, enter the value for the desired suppression state: **Suppressed** (or **S**), or **Resolved** (or **R**). You cannot set a component to the lightweight state. If a cell is left blank, the default suppression state is **Resolved**.

Suppression of features that belong to the assembly

The following types of features belong to an assembly: mates, assembly feature holes and cuts, and component patterns. Sketches and reference geometry may also belong to an assembly. You cannot control the suppression of a feature that belongs to an individual assembly component. There are two ways to specify the suppression of assembly features.

❑ **Method 1 (recommended).** The column header for controlling assembly feature suppression uses this syntax:

$STATE@feature

where **feature** is the name of the feature. For example, the column labeled **$STATE@Hole1** controls the suppression of the first assembly feature hole.

In the table body cells, enter the value for the desired suppression: **Suppressed** (or **S**), **Unsuppressed** (or **U**). If a cell is left blank, the default is **Unsuppressed**.

❑ **Method 2.** Enter only the feature name in column header cell. To suppress the feature, leave the table body cell *blank*. To include the feature, enter any string in the body cell. This is the same syntax as that used in *part* design tables, and is included for the convenience of users of earlier versions of SolidWorks, who are already familiar with this method.

Part number

You can control the value that is displayed in the **Part No.** column in a bill of materials, when this configuration is used as a sub-assembly. The column header uses this syntax:

> **$PARTNUMBER**

This corresponds to the **Use name below** option in the **Configuration Properties, Advanced** dialog box.

In the table body cells, enter the part number for each configuration. If a cell is left blank, the default is to use the configuration name (the same as checking **Use name below**, and leaving the default configuration name in the text box).

Comment

The **Configuration Properties** dialog box has a **Comment** box, where you can enter a description, or other additional information about the configuration. The column header for entering information in this box for each assembly configuration uses this syntax:

> **$COMMENT**

In the table body cells, enter the comments for the assembly configuration. Comments are optional; if a cell is left blank, the **Comment** box is empty.

Expand in BOM

You can control how the assembly is listed in a BOM, when this configuration is used as a sub-assembly. The column header uses this syntax:

> **$NEVER_EXPAND_IN_BOM**

This corresponds to the **Don't show child components in BOM when used as a sub-assembly** option in the **Configuration Properties, Advanced** dialog box.

- When *disabled*, the sub-assembly is expanded if you select either **Show parts only** or **Show assemblies and parts in an indented list** when you insert the BOM.

- When *enabled*, the sub-assembly is *never* expanded, regardless of the option you select when you insert the BOM. The effect is the same as selecting **Show top-level subassemblies and parts only**.

In the table body cells, to enable the option (never expand) enter **YES** (or **Y**). To disable the option (allow to expand), enter **NO** (or **N**). If a cell is left blank, the default is to disable the option.

For more information, see **Bill of Materials** on page 9-29.

Custom properties

The column header for specifying a custom property uses this syntax:

> **$PRP@*property***

where ***property*** is the name of a custom property. You can use one of the custom properties listed in the **Summary Info** dialog box (click **File, Properties**, **Configuration specific**), or you can add a new custom property.

NOTE: If you plan to use the custom properties as columns in the bill of materials, do not use property names that contain spaces. See **Custom properties in a bill of materials** on page 9-32.

For example, use the header **$prp@Client** to control the custom property **Client** for each configuration. In the table body cells, enter the property value for each configuration. If a cell is left blank, the property is undefined for the configuration.

User notes

The design table can include additional columns or rows for information only (notes, intermediate calculations, etc.). Use the keyword **$USER_NOTES** in the header of any column or row that you do not want evaluated. You may use as many of these rows or columns as you need. However, remember that you may not include rows or columns with empty headers in the middle of the table.

Summary

parameter syntax (header cell)	legal values (body cell)	default if value left blank
$never_expand_in_BOM	Yes (never expand) No (allow to expand)	No
$comment	any text string	empty
$partnumber	any text string	configuration name
feature_name	blank (suppressed) any string (unsuppressed)	Suppressed
$state@ feature_name	Suppressed, S Unsuppressed, U	Unsuppressed
$show@ component<instance>	Yes, Y No, N	No
$state@ component<instance>	Resolved, R Suppressed, S	Resolved
$configuration@ component<instance>	configuration name	not allowed
dimension@feature	any legal decimal value for the dimension	zero (error if not a legal value)
$prp@property	any text string	property is undefined
$user_notes	any text string	not evaluated

Opening Assemblies

When you open an assembly document, the software accesses the information for all the components that are included in the assembly.

If any of the components has changed since the last time the assembly was opened, it is a good practice to rebuild the assembly before you continue working on it. You can specify whether you want assemblies to be rebuilt when you open them. Under **Tools, Options, Performance**, click **Rebuild assembly on load**, then select one of these options (this is a system option, and applies to all assemblies, not just the active assembly):

- **Never**. Open without rebuilding. This is a fast way to open an assembly when you only want to look at it, but you do not need to edit or save it.

- **Always**. Rebuild any assembly when it is opened, if any components have changed.

- **Prompt**. Ask each time an assembly is opened, if any components have changed. With this option, you can decide when you open each assembly whether or not you want to rebuild. When you open an assembly, click **Yes** or **No**.

   If you also click **Don't ask me again**, the option selected under **Rebuild assembly on load** is updated to reflect your choice (**Yes** changes the option to **Always**, **No** changes the option to **Never**).

By default, when you open an assembly, it is opened in the configuration that was in use the last time it was saved. You can also choose a different named configuration when you open the assembly.

You can use simplified versions (configurations) of parts or sub-assemblies when opening an assembly. See **Component Configurations in an Assembly** on page 6-46.

> **NOTE:** Whenever you open an assembly, if a part or sub-assembly file cannot be found, you are asked to browse for it. You can also substitute a different part file, or the same part file with a new name. See **Replacing a Component in an Assembly** on page 6-10.

Opening a Named Configuration

You can open a named assembly configuration, that is, a variation of an assembly with modified contents (with suppressed components, different dimensions for assembly features, etc.). See **Using Assembly Configurations** on page 6-35 for information about the types of variations that configurations can include.

You can also open an assembly using a specific configuration of its parts, where available. See **Working with Part Configurations** on page 5-15 for information about creating configurations of parts.

To open a named configuration of an assembly:

1 Click **File**, **Open**.

2 In the **Open** dialog box, set **Files of Type** to **Assembly Files(*.asm, *.sldasm)** or **All Files**.

3 Select an assembly and click the **Configure** check box.

4 Click **Open**.

5 In the **Configure Document** dialog box, click **Use Named Configuration** and select the desired configuration.

6 To use a named part configuration, click **Use specified configuration name for all part references when available**, and enter the part configuration name. Any part that has a configuration with the specified name is opened in that configuration.

NOTE: If the referenced configuration of any component specifies **Use Named Configuration**, this option does not override the named configuration.

---

7 Click **OK**. The selected configuration opens.

You can change to another configuration of the assembly while the assembly is open. See **Viewing an Assembly Configuration** on page 6-37 for information.

Suppressing/Resolving All Components

You can quickly create new configurations, by globally suppressing or resolving all the components. The easiest way to do this is to open the assembly in a new configuration, with one of these options:

• structure only shown (all components suppressed)

• all referenced models shown (all components resolved)

To globally suppress or resolve components in a new assembly configuration:

1 Click **File**, **Open**. In the **Open** dialog box, set **Files of Type** to **Assembly Files** or **All Files**.

2 Select an assembly, click **Configure**, and click **Open**.

3 In the **Configure Document** dialog box, select either:

• **New configuration showing assembly structure only.** This opens the assembly structure only. The components are all suppressed, regardless of the state they were in when last saved. No components appear in the assembly model until you change their suppression state to resolved.

• **New configuration showing all referenced models.** This opens the complete assembly. The components are all resolved and shown, regardless of the suppression and visibility state they were in when last saved. All components appear in the assembly model.

4 Enter a new **Configuration Name**, and click **OK**.

Component Configurations in an Assembly

If a component (either an individual part or a sub-assembly) has more than one configuration, you can specify which configuration to use for each instance in the assembly.

To specify the configuration of a component instance:

1   Right-click the component name in the FeatureManager design tree, or right-click on a component face, and select **Component Properties**.

2   Under **Referenced Configuration**, select one of the following:

- **Use component's in-use or last saved configuration**

   Any instances with this option use the configuration that is currently active (if the document is open), or the configuration that was active when it was saved. When you open an assembly, any active components that are not already open in their own windows are loaded into memory. If you change the configuration in use in the component's own window, the newly selected configuration is placed in memory; any instances that use this option update when you return to the assembly.

   To change the active configuration for a component that is not already open, right-click the name in the FeatureManager design tree, and select **Open *part*.sldprt** or ***assembly*.sldasm**. Then make the desired configuration active.

   You cannot change the referenced configuration of a suppressed component, because suppressed components are removed from memory. However, if you open the document of a suppressed component, then change the configuration, the newly selected configuration is used if you restore the component to the resolved state.

   The component name in the FeatureManager design tree indicates whether a component uses the last saved or in-use configuration.

- **Use named configuration**

   To *always* use a specific configuration, regardless of the configuration currently in memory, select this option, and choose a configuration from the pull-down list.

   It is good design practice to use named configurations when components have multiple configurations, to prevent the in-use configuration from affecting the assembly in unexpected ways.

3   If the selected component is a component of a sub-assembly, choose which configuration properties to use:

- **Use visibility properties specified in configuration *<configuration name>* of *<sub-assembly name>*.** The **Configuration specific properties** (**Hide Model, Color**) specified for this component in the selected configuration in the sub-assembly document are used.

- **Override visibility properties specified in configuration *<configuration name>* of *<sub-assembly name>*.** The **Configuration specific properties** specified for this component in this configuration of the assembly are used.

4   Click **OK**.

Assembly Envelopes

An *assembly envelope* is a special type of assembly component. An envelope is not a *functional* component of the assembly. It is a *reference* component, and it is ignored in global assembly operations (Bill of Materials, Mass Properties, etc.).

You can use an envelope by itself to quickly change the visibility of assembly components. You can also use an envelope (either alone or in combination with file properties), to select components for other editing operations, such as suppress, copy, or delete.

This functionality makes it easy to create and manage complex configurations. See **Simplifying Large Assemblies** on page 6-30 for more information about changing the suppression state or visibility of components in assembly configurations.

There are two ways to insert an envelope component:

- Insert an existing part file to use as the envelope.
- Create a new envelope component in the context of the assembly.

In either case, the envelope component must be a *solid* volume (not shelled). This is because selection with envelopes is based on *interference* between the volumes of the functional assembly components and the envelope component. It is *not* based on the *containment* of components within the bounding box of the envelope component.

Envelope components are displayed in a light blue transparent color when you use **Shaded** view mode. An **Envelope** feature is added to the Configuration Manager for each envelope component that you create. Also, envelope components are identified in the FeatureManager design tree.

To create an assembly envelope from a file:

1 Click **Insert, Envelope, From File**.

2 In the **Open Reference** dialog box, select the part file to use as an envelope.

3 Click in the assembly window where you want to place the envelope component. You can mate the envelope component to the other components in the assembly to position it precisely.

To create an assembly envelope in the assembly context:

1 Click **Insert, Envelope, New**.

2 In the **Save As** dialog box, enter a name for the new envelope component. A part document with this name is created.

3 Click a face or plane in the assembly window where you want to begin sketching the envelope component. **Plane1** of the new envelope component is mated to the selected face or plane with an **Inplace** mating relation.

4 Create the base feature and any additional features of the envelope component.

You can reference the geometry of the assembly components as you create the envelope component. If any of the referenced entities changes, the envelope updates accordingly.

5   When you are done defining the new envelope component, right-click anywhere in the graphics area, and select **Edit Assembly:assemblyname**.

To remove an assembly envelope:

1   Select the envelope component in either the FeatureManager design tree or the graphics area. You can also select the **Envelope** feature in the Configuration Manager.

2   Press the **Delete** key. Both the envelope component and the **Envelope** feature are deleted.

To use the envelope for selection:

1   In the Configuration Manager, right-mouse click the **Envelope** feature, and choose **Select using envelope**.

2   Choose one or more criteria for selecting the components:

- **Inside envelope.** Components that are completely inside the envelope are selected.

- **Outside envelope.** Components that are completely outside the envelope are selected.

- **Crossing envelope.** Components that cross the boundary of the envelope are selected.

3   Click **OK**. Components that meet the criteria are selected, and you can perform the desired operation (**Suppress**, **Hide**, **Delete**, etc.).

To use the envelope for component visibility:

1   In the Configuration Manager, right-mouse click the **Envelope** feature, and choose **Show/Hide using envelope**.

2   Specify the envelope criteria for selecting the components (see above).

3   Choose whether to **Show** or **Hide** the components that meet the criteria.

4   Specify what to do with any component that does not meet the criteria:

- If you clicked **Show** in Step 3, you can either **Hide it** (regardless of current display state), or **Leave its show/hide state as is**.

- If you clicked **Hide** in Step 3, you can either **Show it** (regardless of current display state), or **Leave its show/hide state as is**.

5   Click **OK**. The display is updated to hide or show components according to your selections.

Component Selection by Properties

You can select components for editing operations based on properties. Properties include those specified under **File, Properties, Custom** and **Configuration specific**, as well as some properties that are specific to SolidWorks documents, including **Part volume, Configuration name, Document name,** and **Part Mass.**

You can specify a set of properties and values to be used as selection criteria. You can combine properties using the logical operators **And** and **Or**.

For example, you assign the custom property *System* to each component in an assembly. You specify the name of the system the component belongs to as the value. Then you select all components that have the custom property *System* whose value is *Cooling* **Or** *Heating*. This selects all the components that belong to either the cooling system or the heating system of the assembly.

You can save any set of selection criteria as a *query*. Then you can reload it whenever you want to apply the same set of criteria again. The query (**.sqy**) is an ASCII text file that you can read and edit using a text editor such as Notepad.

Advanced Component Selection

You define the selection criteria by specifying the properties, conditions, and property values of components you want to select.

To define advanced selection criteria:

1 Click **Tools, Advanced Select**.

2 Under **Define additional criteria**, select a **Property** from the dropdown list. Properties include those listed under **File, Properties**, on the **Custom** or **Configuration specific** tabs, plus several special SolidWorks properties.

3 Select a **Condition**, and enter a **Value**.

   • For most properties, conditions include standard mathematical operators.

   • For some properties, conditions include **is (exactly)**, **is not,** and **contains**. You use **Contains** to specify a text string which is part of the text of the value; for example, the value "simplified" *contains* the string "sim".

   • For the SolidWorks special property **Part volume**, conditions include **is inside**, **is crossing**, and **is outside**. The values that are available are the names of any envelopes that you have defined. See **Assembly Envelopes** on page 6-47.

4 Click the **Add** button to include the selection criteria in the **Criteria** list.

5 To add another criteria to the list, click **And** or **Or** as a logical operator for combining the criteria, then specify another **Property**, **Condition**, and **Value**. Click **Add** after specifying each new criteria.

6 To delete a criteria from the list, click in the *first column* of the **Criteria** list box, *beside* the criteria to delete. A blue box appears beside the selection. Click the **Delete** button.

7   When you have specified all the necessary criteria, click **Apply**. The selected components are highlighted. If needed, you can modify the selection, and click **Apply** again.

8   If you want, you can **Save Criteria** for future use, or **Load Criteria** that you saved earlier. See **Saving and Reloading Selection Criteria** on page 6-51.

9   Click the pushpin if you want to keep the dialog box open for further selections and operations.

10   Perform the desired editing operation, such as **Suppress, Hide,** or **Copy**.

Advanced Show/Hide Components

For show and hide operations, you can also control the visibility of components that *do not meet* the specified criteria. You use the same methods to select the components as described in the previous section, **Component Selection by Properties** on page 6-49.

To define advanced visibility criteria:

1   In the Configuration Manager, right-mouse click the active configuration, and select **Advanced Show/Hide**.

2   Define the selection criteria by specifying the properties, conditions, and values of components you want to select.

3   Under **Operations**, choose whether to **Show** or **Hide** the components that meet the criteria.

4   Specify what to do with any component that does not meet the criteria:

   • If you clicked **Show** in Step 3, you can either **Hide it** (regardless of current display state), or **Leave its show/hide state as is**.

   • If you clicked **Hide** in Step 3, you can either **Show it** (regardless of current display state), or **Leave its show/hide state as is**.

5   Click **Apply**. If needed, you can modify the selection, and click **Apply** again.

6   If you want, you can **Save Criteria** for future use, or **Load Criteria** that you saved earlier.

7   Click **Close** when you are satisfied with the selections.

Saving and Reloading Selection Criteria

When you use **Advanced Selection** or **Advanced Show/Hide**, you can save the selection criteria to a file. Then you can load the criteria later, to quickly apply the same set of selection criteria whenever you want.

To save selection criteria:

1   In the **Advanced Selection** or **Advanced Show/Hide** dialog box, specify a set of selection criteria.

2   Click **Save Criteria**.

3   In the **Save Criteria** dialog box, browse to the directory where you want the file to be stored, enter a file name, and click **Save**.

The query is saved with the extension **.sqy**

To reload selection criteria:

1   In the **Advanced Selection** or **Advanced Show/Hide** dialog box, click **Load Criteria**.

2   In the **Load Criteria** dialog box, browse to and select the query file (**.sqy**), and click **Open**. The query is loaded, and the criteria are listed in the dialog box.

Displaying Annotations in an Assembly

To specify the display of annotations in an assembly, right-mouse click the **Annotations** folder in the FeatureManager design tree, and select **Details**. In the **Annotation Properties** dialog box, you have these options:

- **Display filter.** Select which annotation types, if any, to display by default. To select specific types, deselect **Display all types**, and select the types you want.

- **Text scale.** Adjust the size of the text as needed.

- **Always display text at the same size.** When checked, all annotations and dimensions are displayed at the same size, regardless of zoom.

- **Display items only in the view in which they are created.** When checked, any annotation is displayed only when the model is viewed in the same orientation as when the annotation was added.

- **Display assembly annotations.** Toggles the display of annotations, including annotations that belong to the assembly, and annotations that are displayed in the individual part documents.

- **Use assembly's setting for all components.** Sets the display of *all* annotations to match the setting for the assembly document, regardless of the setting in the individual part documents. Use this option along with **Display assembly annotations** to display different combinations of annotations.

For complete information about creating and using annotations, see Chapter 9, "Detailing."

Customizing the Appearance of an Assembly

You can select a color for each component in an assembly. By default, the global preferences you set for the application in **Tools**, **Options, Color** are used.

To change the color of a part in an assembly:

1   Right-click the component in the graphics area or in the FeatureManager design tree, and select **Edit Part**.

2   Click **Tools**, **Options**, and click the **Color** tab.

3   Click **Edit,** change color as desired, and click **OK**. The color of *all* instances of the part changes, including the referenced part document.

To change the color of selected part instances in an assembly:

1   Right-click the component instance in the graphics area or in the FeatureManager design tree, and select **Component Properties**.

   To change the color of several components at once, **Ctrl**-click the components, then right-click, and select **Component Properties**. If you select multiple components in the FeatureManager design tree, right-mouse click in the graphics area; otherwise, only the last component you click remains selected.

2   Click the **Color** button.

3   Specify the values for **Red**, **Green**, and **Blue**, and click **OK**,

   – or –

   Click **Color**, click **Change Color**, choose a color from the palette, and click **OK**.

4   To use the default color for the assembly document, click **Use Assembly**.

5   To restore the component to the original color specified in the part document, click **Remove Color**.

6   Click **Advanced** to set other properties for the selected component, such as **Transparency** and **Shininess**.

7   Click **OK**. The color of the selected instance(s) changes, but not the referenced part document.

Assembly Statistics

You can generate a set of statistics for an assembly document. The report includes such items as the total number of components, the number of unique parts and sub-assemblies, the number of components in each suppression state, the number of top-level mates, and the maximum depth of the assembly hierarchy (for nested sub-assemblies).

To generate a list of assembly statistics, click **Tools, Assembly Statistics**. When you are finished reading the report, click **OK** to close the dialog box.

Sending Assembly Documents

You can send assembly files (**.sldasm** files) by electronic mail. You can choose to send only the assembly itself, or you can send the assembly and all referenced documents.

This function uses the electronic mail application installed on your system.

To mail an assembly file:

1 With the assembly open, click **File, Send To**. The **Send Mail** dialog box appears.

2 Click either **<assemblyname>.sldasm only** or **<assemblyname>.sldasm and other documents it references**.

   If you click **<assemblyname>.sldasm and other documents it references**, a list of the referenced documents, the number of documents, and the combined size of the documents is displayed.

   > **NOTE:** The size of the combined documents may be important to you because mailing large size documents may cause transmission problems.

   You can selectively delete files from the mailing.

   • Select the files you do not want to send and click **Remove**.

   • Select and click **Attach** to reattach them.

   • Click **Attach All** to reattach all unselected files.

3 Click **OK** to send your document(s).

4 Enter your mail password, the mailing address, and any other information requested by your mail application.

7

Working with Parts within an Assembly

In this chapter, you will learn about *top-down* assembly design, including several techniques for building and modifying components within the context of an assembly.

These techniques are based on *external references* and *time-dependent features.*

❑ You can use the geometry of one component to help define another. When you create references between components in an assembly, the models are fully associative. Changes you make to a *referenced component* cause the others to update accordingly.

❑ You can create cut or hole features that are added only after the components are assembled. These *assembly features* exist only in the context of the assembly.

❑ You can start with a layout sketch, define fixed component locations, planes, etc., then design the parts referencing these definitions.

❑ You can place component patterns, create molds, and join part volumes.

External References

An *external reference* is created when a part, assembly, or drawing refers to another part, assembly, or drawing. If the referenced components change, the affected features change also.

In an assembly, you can create an in-context feature on one component that references a feature of another component. This in-context feature has an external reference to the other component. If you change the feature on the referenced component, the associated feature changes accordingly. See **Editing Parts in the Assembly Context** on page 7-5 for more information.

An example of an external reference is a mold. The cavity in the mold has an external reference to the design part you use to create the cavity. If you change the design part, the cavity changes to match it. See **Creating Molds** on page 7-18.

You can change or break external references on features by editing the features and changing or deleting the relationship.

A component or feature with an external reference has the suffix **->** if the external reference is in-context (solved). If the external reference is out-of-context, the suffix is **->?**. This can occur if you delete, move, or rename the referenced document.

To list the external references on a part or feature:

Right-mouse click the part or the feature with the **->**, and select **List External References.** The referenced components, features, and entities are listed, as well as the status of each reference.

Setting External Reference Options

To set external reference behavior:

1 Click **Tools, Options, External References**.

2 Choose from these options:

- **Open referenced documents with read-only access.** Specifies that all referenced documents will be opened for read-only access by default.

- **Don't prompt to save read-only referenced documents (discard changes)**. Specifies that when this part is saved or closed, no attempt will be made to save its referenced documents that are opened for read-only access.

- **Allow multiple contexts for parts when editing in assembly.** Allows you to create external references for a single part in more than one assembly context. However, any individual *feature* or *sketch* may only have external references *in one assembly*.

- **Load referenced documents**. Lets you specify whether to load referenced documents when you open a part that has external references (such as a derived part, or a part that has been edited in the context of an assembly).

If you click **Never**, referenced documents are never opened. If you click **Always**, referenced documents are always opened, in a minimized window.

If you click **Prompt**, you are asked about loading externally referenced documents each time you open a derived part. Click **Yes** or **No** in the dialog box that appears when you open the part. If you also click **Don't ask me again**, the option is updated to reflect your choice (**Yes** changes the option to **Always**, **No** changes the option to **Never**).

If you do not open the referenced documents, the external references are considered out-of-context, and are marked with **->?**. To resolve the out-of-context reference, open the referenced document, so that changes can be propagated as needed.

3   Choose from these options under **Assemblies**:

   • **Automatically generate names for referenced geometry**. When this option is *off* (the default), you can mate to parts for which you have read-only access. Unless you plan to replace components, leave this option *off*, especially in a multi-user environment. See **Preserving Mates with Component Replacement** on page 6-14 for more information.

   • **Update component names when documents are replaced**. Deselect this option only if you use the **Component Properties** dialog to assign a component name in the FeatureManager design tree that is different from the filename of the component. See page 6-11 for information about component names.

4   Click **OK**.

Searching for Referenced Documents

When you open any parent document, the other documents that are referenced in the parent document are loaded in memory also. In the case of assemblies, components are loaded in memory according to the suppression state they were in when the assembly was last saved. By default, the software searches for referenced documents *in the following order*:

   1) The directory paths specified in the **Folders** list on the **External References** tab of the **Options** dialog box. This list is only used if the **Search document folder list for external references** option is checked.

   2) The last path you specified to open a document.

   3) The last path the system used to open a document (in the case of the system opening a referenced document last).

   4) The path where the referenced document was located when the parent document was last saved.

   5) The path where the referenced document was located when the parent document was last saved *with the original disk drive designation*.

   For example, assume that you had stored documents on the **C:** drive, then moved them to the **E:** drive. Step 4 searches the proper directories on the **E:** drive, and Step 5 searches the proper directories on the **C:** drive.

6) If a referenced file still is not found, you are given the option to browse for it.

> **NOTE:** All updated reference paths in the parent document are saved when you save the parent document.

You can define the folder list, then enable or disable the search behavior as needed, without affecting the list.

To use a document folder list to search for external references:

1   Click **Tools, Options**, and select the **External References** tab.

2   Click the check box **Search document folder list for external references** to enable the search behavior.

3   Under **Show folders for**, select **Documents**, then choose from these options:

- To *add* a directory path to the search order, click the **Add** button. The **Choose Directory** dialog box appears. Browse to the desired directory, and click **OK** to add the path to the search list for **Documents**.

- To *delete* a directory path from the search list, select the path in the list box, and click the **Delete** button.

- To *change* the search order, select a directory path in the list box, and click the **Move Up** button or **Move Down** button to change the path's position in the search list.

4   Click **OK**.

> **NOTE:** You can also specify the paths for **Palette features**, **Palette parts**, **Palette forming tools,** and **Custom symbols** in this section.
>
> Search paths for these items are *always* used, whether the **Search document folder list for external references** option is enabled or not. See **Organizing Palette Items** on page 10-8 for more information.

Saving Assemblies with In-context Features

There are some special considerations when you save an assembly with in-context features. Parts that have features defined in the context of an assembly must maintain the association with the parent assembly in order to update correctly, both inside and outside of the assembly context.

If you need two versions of the same part (one that includes the in-context features, and one that does not include them), you must edit the referenced file locations when you save the assembly. This prevents the in-context features from affecting the original part document.

Consider this example. In a mold assembly, you edit the mold base in context to create a cavity. If you want to keep the original, unedited mold base part for use in other mold assemblies, you need to do one of the following:

- Save the assembly and the parts to a new directory, and edit the references to reflect the new path

  – or –

- Save the edited mold base part with a new name, and replace the part in the assembly

When you save the assembly, you save its updated reference to the edited mold base (*with* the cavity). This mold base updates correctly, both within the assembly, and on its own. The original mold base is left unchanged.

See **Replacing a Component in an Assembly** on page 6-10 for more information.

Editing Parts in the Assembly Context

While editing an assembly, you can:

❑ Create a part in the assembly window

❑ Edit a part in the assembly window

❑ Open a part file in its own window

Whenever you are creating or editing a component in the context of an assembly, the component and its name in the FeatureManager design tree turn pink. This is to make it obvious that you are editing the *component*, not the *assembly*. You can change this color.

To change the color of a component being edited in context:

1 Click **Tools, Options, Color**, and click **View System Defaults**.

2 In the **System** list, click **Edit Part in Assembly**.

3 Click the **Edit** button, choose a new color from the palette, and click **OK**.

Creating a Part in an Assembly

You can create a new part (or edit an existing part) in the context of an assembly. That way you can use the geometry of other assembly components while designing the part. The new part has its own part file so you can modify it independently from the assembly.

To create a part within an assembly:

1   With the assembly window active, click **Insert, Component, New**. The **Save As** dialog box appears.

2   Enter a name for the new part and click **Save**. This saves the part in a part file so you can edit it separately.

   The name of the new part appears in the FeatureManager design tree, and the component cursor ⬚◇ appears.

3   Select a plane or face on which to position the new part. An **InPlace** (coincident) mate is added between **Plane1** of the new part and the selected plane or face.

   If the assembly is empty, select a plane from the FeatureManager design tree. A sketch is automatically opened in the new part.

   The new part is fully positioned by the **Inplace** mate. No additional mates are required to position it. If you wish to reposition the component, you need to delete the **Inplace** mate first. See **Deleting a Mating Relationship** on page 6-24.

4   Construct the part features, using the same techniques as you use to build a part on its own.

   Reference the geometry of other components in the assembly as needed. If you extrude a feature using the **Up To Next** option, the *next* geometry must be on the same part. You cannot use the **Up To Next** option to extrude to a surface on another component in the assembly.

5   Click **File, Save,** then select the part name in the **Resolve Ambiguity** dialog box, or select the assembly name to save the entire assembly and its components.

6   To return to editing the assembly, right-mouse click the assembly name in the FeatureManager design tree, or right-mouse click anywhere in the graphics area, and select **Edit Assembly:*assemblyname***, or click the **Edit Part** tool 🌎.

Editing a Part in an Assembly

Editing a part while in an assembly allows you to modify a component without leaving the assembly. You can also reference surrounding geometry while creating new features. The sketches that you create while editing a part in the assembly may use any edge or can be dimensioned to any edge of any part. Features may use other end conditions, such as **Up to Surface**, to another component. See **Specifying End Conditions** on page 4-4.

To edit a part while in an assembly:

1 Right-click the component in the FeatureManager design tree or the graphics area and select **Edit Part**, or click **Edit Part** on the Assembly toolbar.

The title bar shows the name of the part in the assembly that is open for editing as **partname in assembly.sldasm**. Note that the message in the status bar indicates that you are now editing the *part* document even though the entire *assembly* is visible.

2 Make necessary changes to the part. You can make most of the same types of changes in the context of an assembly as you can make in the part's own document.

3 When you finish editing the part, right-mouse click the assembly name in the FeatureManager design tree, or right-mouse click anywhere in the graphics area, and select **Edit Assembly:assemblyname**, or click the **Edit Part** icon again to return to editing the assembly.

Opening a Part in its Own Window

While in an assembly, you can open a part file in its own window and make the desired modifications with the part isolated. The changes automatically update the assembly.

To open a part within an assembly:

1 Right-click the component in the FeatureManager design tree or graphics area, and select **Open <filename>**.

The part or sub-assembly file opens in a separate window.

2 Edit the part or sub-assembly as desired.

When you return to the assembly window, the assembly automatically updates to reflect the edits.

To open the assembly in which a part or feature is referenced:

1 In the part window, right-click the feature that has the external reference. The referenced feature has the suffix **->**.

2 Select **Edit in Context**.

The assembly in which the feature has a reference opens in a separate window.

Sub-assembly Editing

You can edit a sub-assembly, at any level of the assembly hierarchy, in context. Most of the operations available for editing a sub-assembly on its own are available when editing a sub-assembly in context.

While you are editing a sub-assembly or one of its components, you can reference the geometry of surrounding components and features at any level of the hierarchy.

You can also reference entities in a sub-assembly while you are editing the top-level assembly.

When you save the top-level assembly document, the edits that you have made to the sub-assembly in context are saved in the sub-assembly document.

Automatic Change of Editing Focus

For some types of editing operations, the software recognizes that the changes you are making belong to the sub-assembly, and not to the top-level assembly. In those cases, the editing focus automatically changes from the top-level assembly to the sub-assembly.

If you edit any of the following items *that belongs to a sub-assembly*, the editing focus changes automatically: sketch, component pattern, mate, assembly feature (cut or hole), reference geometry.

Similarly, if you edit a feature (the definition or the sketch) of a sub-assembly component, the software automatically changes to **Edit Part** mode. References that you make while editing a sub-assembly component are always made in the context of the top-level assembly, not the sub-assembly that contains it.

If you add a mate between a sub-assembly component and some geometry outside that sub-assembly (in the top-level assembly, or in another sub-assembly that is *not* nested under the active sub-assembly), the focus changes automatically, and the mate is added to the lowest level assembly that contains both components. You cannot add mates between components in the same sub-assembly while you are editing the top-level assembly.

Changing the Editing Focus Manually

For some types of editing operations, you need to specify that the changes you are making belong to the sub-assembly, and not to the top-level assembly. This includes any case where the edits could be successfully applied to either one. For example, you can add an assembly feature hole either to a sub-assembly, or to the top-level assembly, with different results.

You must activate the sub-assembly to perform these editing operations:

- Add a component from a file, or drag an existing part document into the sub-assembly. You cannot add a *new* component in the context of a sub-assembly.

- Delete a component from a sub-assembly.

- Add a component pattern, an assembly feature cut or hole, a sketch, or reference geometry.

- Add a mate between components of the active sub-assembly (both at the same level in the hierarchy, or one at a lower level, in a nested sub-assembly of the active sub-assembly).

  It is important to note that you cannot add a mate between a component *within* the active sub-assembly and a component *outside* the active sub-assembly.

- Use dynamic assembly motion on sub-assembly components. You can fix, float, and move sub-assembly components when the sub-assembly is active, to visualize the movement within the sub-assembly.

NOTE: While a sub-assembly is active, the mates in the top-level assembly that position the sub-assembly with respect to the top-level are temporarily ignored.

Top-level mates are solved again when you return to editing the top-level assembly and **Rebuild**.

To activate a sub-assembly for editing in context:

Right-mouse click the sub-assembly icon in the FeatureManager design tree, and select **Edit Subassembly.**

When you activate a sub-assembly, it turns pink, and the rest of the assembly turns gray, just as it does when you edit a part. When you edit a component of a sub-assembly, only the component being edited turns pink.

Time-Dependent Features

SolidWorks gives you the ability to work with time-dependent features in an assembly. Time-dependent features are the basis of top-down design.

Time-dependent features include:

❑ Assembly features (assembly cuts, holes, and weld beads)

❑ Assembly planes, axes, and sketches

❑ Assembly component patterns

When a time-dependent feature references components that have already been positioned by mates, the features become dependent on both the components and the mategroup(s) that position the components. You can view these dependencies by right-mouse clicking the feature and selecting **Parent/Child**.

Mates are solved together as a system. This system is called a *mategroup*. Each assembly has a default mategroup, **MateGroup1**. Within a mategroup, the order in which mates are added does not matter; all the mates are solved at the same time. In a bottom-up design, only one mate group is required. All mates added between components are placed in **MateGroup1**.

In this example, coincident mates are used to align the faces of the components.
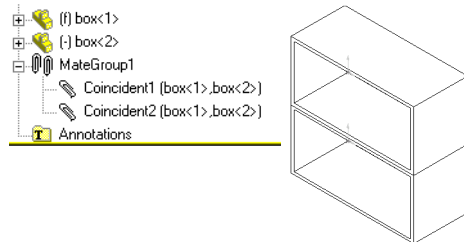
Then a cut is extruded through both components. This is an *assembly feature*, not a cut feature in the part document.

Now add a component (a pin, for example) to the assembly.
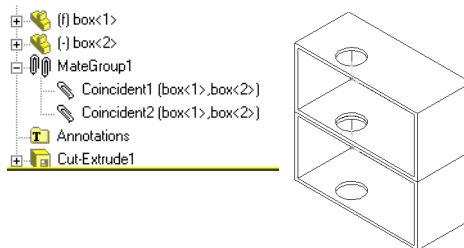
If you mate the pin to the assembly feature cut (which is time-dependent), an additional mate group is required. This is because the new component can only be positioned after the time-dependent feature has been updated.

The position of a component in an assembly is fully defined in the first mate group that contains mates to it. If it is referenced in later mate groups, it is treated as a fixed component.

Additional mate groups are added automatically when required and removed automatically when the last mate they contain is deleted. When a new mate is added, it is put in the earliest mate group possible, based on parent/child relationships.
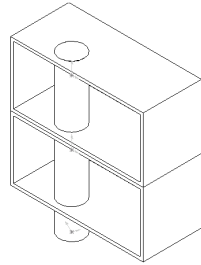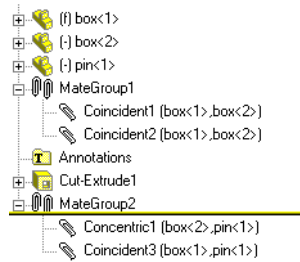
> **NOTE:** It is good design practice to only reference time-dependent features for mating when that is the only way you can achieve the design intent of the assembly. You can edit the positions of the components with more flexibility when time-dependent features are not involved because the order of evaluation does not matter.

It is possible to have a situation where two components have been partially positioned in **MateGroup1,** and in order to complete the positioning, you need to add a mate which references a time-dependent feature that is dependent on **MateGroup1**.

You could do this by mating the flat face of the pin coincident to the top face of the upper part first. Then add the assembly hole, and add the concentric mate between the pin and the assembly hole. The coincident mate is moved to the second mate group for a successful solution. This is because it must be solved simultaneously with the concentric mate to position the pin, and the concentric mate can only be solved after the assembly hole is updated.

In this example, the software detects that the coincident mate can move forward, and moves the mate when the concentric mate is performed. However, in more complicated cases you may need to determine which mates to move forward. To do this, delete the mates that position the component you wish to move, then add the mate to the time-dependent feature (to create the additional mate group), then add mates as needed to fully position the component.

When positioning a component to geometry that depends on a time-dependent feature, it is good design practice to mate the time-dependent feature *first*, then add additional mates as needed.

Adding a Component Pattern

You can define a pattern for placing components in an assembly in much the same way as you define a feature pattern in a part.

You can also place a pattern of components in an assembly based on a feature pattern of an existing component. For example, you can insert a set of bolts in a pattern of bolt holes on an assembly component.

To define a new pattern for placing components in an assembly:

1   Insert and mate components in an assembly.

2   Click **Insert, Component Pattern**.

3   In the **Pattern Type** dialog box, click **Define your own pattern**, choose **Linear** or **Circular**, and click **Next**.

4   Click the **Seed component** (the component to copy).

5   Click the **Along Edge/Dim** box, select an entity to indicate the pattern direction, and click **Reverse Direction** if necessary.

   • For a linear pattern, select a linear edge or a linear dimension.

   • For a circular pattern, select an axis, a linear edge, or an angular dimension.

6   Specify the **Spacing** and **Instances**, and click **OK**.

To use an existing pattern to place a pattern of components in an assembly:

1   Insert into the assembly a component that has a pattern feature, and a component that you can mate with the pattern.

2   Mate the component to be copied to the *original* feature used to create the pattern on the first component (not one of the copied instances).

3   Click **Insert, Component Pattern**.

4   In the **Pattern Type** dialog box, click **Use an existing feature pattern** and click **Next**.

5   In the **Derived Component Pattern** dialog box, click the **Seed Component(s)** box and select the component to copy from the FeatureManager design tree.

6   Click the **Pattern Feature** box, then click the pattern feature in the FeatureManager design tree.

7   Click **Finish**.
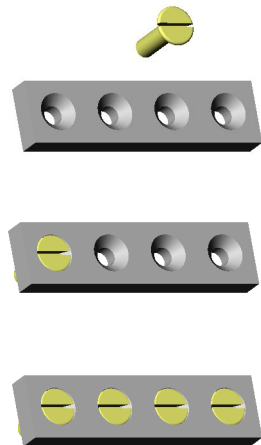
To remove an instance in a component pattern:

    1   Expand the pattern feature in the FeatureManager design tree, and select the instance you want to delete, or click a face on the instance you want to delete in graphics area.

    2   Press the **Delete** key.

To restore the instance:

    1   Right-mouse click the pattern feature in the FeatureManager design tree, and select **Edit Definition**.

    2   In the **Positions to Skip** list, select the instance and press the **Delete** key (to remove it from the list).

See **Linear Pattern** on page 4-42 and **Circular Pattern** on page 4-43 for more information about patterns.

Reorder and Rollback

Reorder

Reordering of assembly items in the FeatureManager design tree is dictated by parent/child relationships, just as it is with features in parts.

You can reorder mates within a mate group, assembly planes, axes, or sketches.

You cannot reorder the first mate group, assembly patterns, or in-context part features. You cannot reorder items in the section of the FeatureManager design tree where the components or the system default features are listed.

Rollback

You can rollback assembly planes, axes, sketches, patterns, assembly cuts and holes, mate groups and in-context part features. In general, rollback of assembly items behaves the same as it does with features in parts.

The following behaviors are specific to assemblies:

❑ If you rollback before a mate group, all components positioned by that mate group are also rolled back. They are removed from the display until you roll forward again.

❑ If you rollback an assembly pattern, the components generated by that pattern are rolled back.

❑ You cannot rollback into the section of the FeatureManager design tree that lists the components (that is, after some of the components are added, or before others). This is because the components are not time-dependent. Use hide or suppress for this purpose.

❑ If you rollback to layout features before the components, all the components are rolled back as well.

Using an Assembly Layout Sketch

You can design an assembly from the top-down using layout sketches. You can construct a sketch showing where each assembly component belongs. Then, you can create and modify the design before you create any parts. In addition, you can use the layout sketch to make changes in the assembly at any time.

The major advantage of designing an assembly using a layout sketch is that if you change the layout sketch, the assembly and its parts are automatically updated. You can make changes quickly, and in just one place.

To use an assembly layout sketch, do the following:

❑ Create a layout sketch in which various sketch entities represent parts in the assembly. Indicate a tentative location for each component, capturing the overall design intent.

❑ Reference the geometry in the layout sketch when you create each component. Use the layout sketch to define the component size, shape, and location within the assembly; make sure that each part references the layout sketch.

To design an assembly using the layout sketch technique:

The following example is typical of an automobile engine accessory drive belt system. The system has several pulleys and idlers, connected by a serpentine belt.

1 Create an assembly sketch. Use circles to represent the outer rims of the pulleys. Use tangent lines between the circles to represent the serpentine belt. Dimension the size and location of each pulley.

2 For *each* pulley, insert a new part, sketch a circle, add a **Coradial** relation with the corresponding circle in the layout sketch, then extrude the sketch. Add other features to the pulley parts as needed.

3 Using the layout sketch as a guide, create the belt around the pulleys as another new part.
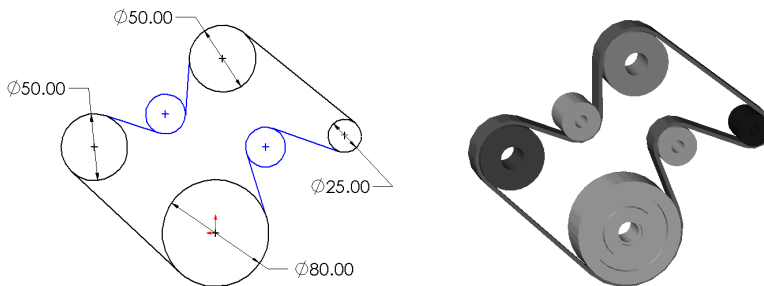
You can change the location or the size of the pulleys by editing the layout sketch.

To make changes to your design:

The assembly now consists of a belt and a set of pulleys whose locations and sizes are dictated by the layout sketch. The advantage of this type of design is clear when you have to make changes to the design.

For example, working *in the layout sketch,* you can:

- Rearrange the pulleys so that a slipping pulley has more of its surface covered by the belt
- Make a driven pulley larger so less torque is required to turn it
- Drag the pulleys and belt to dynamically experiment with different solutions to your problem
- Establish relationships within the sketch that would be difficult or impossible to create within the assembly
- Set an angle dimension to specify how much of each pulley must be covered by the belt

When you exit the layout sketch, the assembly and the parts are updated.
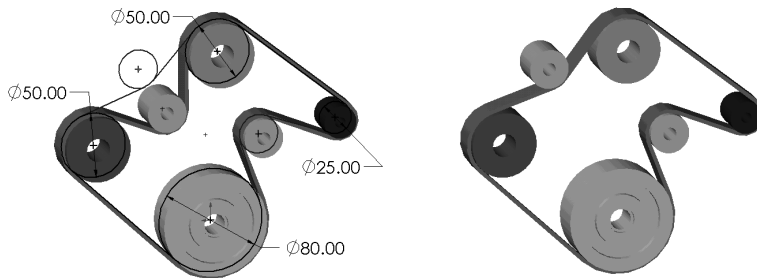
If you designed this assembly without a layout sketch, you would need to change many dimensions, or move components in the assembly, then rebuild each time.

The layout sketch does not have to be the master plan for a design. If you had a model of an engine with some fixed pulley locations, you could relate the layout sketch to the model. Then, to design the belt system, you could make the circles in the layout sketch coincident with the known locations of the pulleys in the model. You would still have the freedom to drag the other pulleys to different locations, and the assembly would update automatically when you made changes.

Working with Assembly Features

While in an assembly, you can create cut or hole features that exist *in the assembly only*. You determine which parts you want the feature to affect by setting the scope.

This is useful for creating cuts or holes that are added *after* the parts are actually assembled, and that affect more than one component. When you want to add a cut or hole to a single component in an assembly, it is better to edit the part in context than to use an assembly feature.

The **Assembly Feature** menu has three selections: **Cut**, **Hole,** and **Weld Bead**. Weld beads are described in Chapter 12, "Welding."

Setting the Scope

The scope of an assembly feature is the list of which parts you want to be affected by the feature. By setting the scope, you can selectively choose the parts you want the feature to impact. For example, a hole cut through the assembly only cuts the parts you select.

The default setting is that an assembly feature affects *all* parts that it passes through in the assembly.

To set the scope:

1   In an assembly, select **Edit**, **Feature Scope**.

2   Select the components that assembly features will affect by clicking the components in the graphics area, or the component names in the FeatureManager design tree.

The selected component names appear in the **Edit Assembly Feature Scope** dialog box.

To remove a component from the list, click the component again in the graphics area or in the FeatureManager design tree.

3   Click **OK**.

Creating an Assembly Feature

Assembly feature cuts and holes affect the assembly only; the part files are not affected. You can set the scope before you create an assembly feature, or you can edit the scope afterwards.

To create an extruded cut assembly feature:

1   Open a sketch on a face or plane, and sketch a profile of the cut.

2   Click **Insert**, **Assembly Feature**, **Cut**, **Extrude**.

3   Set the **Depth**, **Type**, and other options as needed in the **Extrude Feature** dialog box.

4   Click **OK**.

To create an assembly feature revolved cut:

1 Open a sketch on a face or plane, and sketch a profile of the cut and a centerline.

2 Click **Insert**, **Assembly Feature**, **Cut**, **Revolve**.

3 Set the **Type** and **Angle** in the **Revolve Feature** dialog box.

4 Click **OK**.

To create an assembly feature hole:

1 Click the planar face where you want to create the hole.

2 Click **Insert**, **Assembly Feature**, **Hole**, **Simple** and set the **Diameter** and **Type**. If you choose the **Blind** type, specify the **Depth**.

3 Click **OK**.

To edit an assembly feature:

Right-click the assembly feature in the FeatureManager design tree, and select either **Edit Sketch**, **Edit Definition**, or **Feature Scope**.

Creating Molds

A mold consists of a base containing a cavity with the shape of the desired part. When you create a mold you must account for the *scaling factor*, that is the amount the material in the mold shrinks or expands as it solidifies. The scaling factor depends on the type of material used and the shape of the mold, and is expressed as a percentage (+/- 20%) of the linear size (not volume) of the design part.

SolidWorks sizes the cavity by the specified scaling factor, using the following formula:

$$\text{cavity size} = \text{part size} * \left(1 + \frac{\text{scaling factor}}{100}\right)$$

For example:

Desired adjustment	Scaling factor in % (enter this number)	Resulting cavity size
2% material shrinkage (cavity larger than part)	2.0	part size $*$ 1.02
2% material expansion (cavity smaller than part)	-2.0	part size $*$ 0.98

Creating Drafts for Molds

You may need to add draft angles to molded design parts, to make it possible to remove the finished part from the mold. Refer to **Split Line** on page 3-11 and **Draft** on page 4-31 for information about splitting faces and adding draft angles.

Simple Mold

To create a mold cavity:

1 Create the design part you want to mold.

2 Create the mold base. Make sure that the mold base is large enough to contain the part you want to mold.

3 Create an assembly containing the mold base and design part(s). You may insert multiple design parts in a single mold base to create multiple cavities.

4 Position the design part *inside* the mold base, using mating relationships as needed.
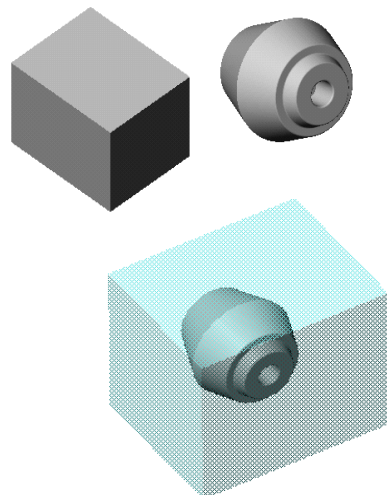
5 In the assembly window, right-click the mold base in the graphics area or in the FeatureManager design tree and select **Edit Part**,

– or –

Click the mold base and click **Edit Part** .

> **NOTE:** It is important to be aware that you are editing the *part*, not the *assembly*, because the changes you are about to make will be reflected in the original part file of the mold base.
>
> If you use standard mold base sizes, and do not want the originals to be affected by this operation, use the **Save As** command in the mold base part document to save it with a different name for use in each new mold assembly. Otherwise, the original mold base will include the cavity you are about to insert.
>
> Also see **Editing Referenced File Locations** on page 6-12 for information about ways to replace the references to the mold base component.

6 In the FeatureManager design tree, select the design part(s).

7 Select **Insert**, **Features, Cavity**.

8 In the **Cavity** dialog box, enter the **Scaling Factor in %** (to a maximum of 20%).

A positive value expands the cavity, a negative value shrinks the cavity. See the examples at the beginning of this section.

9 Select the point about which scaling occurs in the **Scaling Type** field.

- **About Component Centroids.** Scales the cavity for each part about its own centroid.
- **About Component Origins.** Scales the cavity for each part about its own origin.
- **About Mold Base Origin.** Scales the cavity for each part about the origin of the mold base part.

10 Click **OK**.

> **NOTE:** If there are multiple solutions for creating the cavity (if creating the cavity creates more than one separate solid body), you must choose which solution to keep. In the **Resolve Result Ambiguity** dialog box, click the **Next** and **Previous** buttons to see the possible solutions. The highlighted geometry will be *kept* (not *removed*) when the cavity is created. Click **OK** when the desired solution is highlighted.

A cavity in the shape of the design part is created. The cavity size reflects the scaling factor you specified. Any changes you make to the design part will update the cavity in the mold base.

mold base

cavity

design part
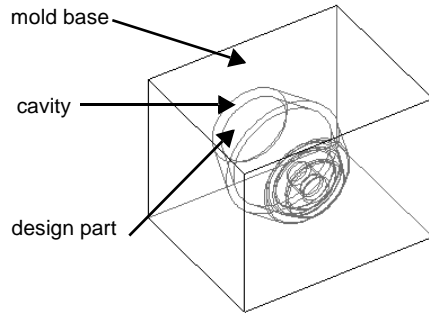
To cut the mold base to make two mold halves:

1 In the FeatureManager design tree, click the mold base and select **File**, **Derive Component Part**.

A part file is created for the derived part (a part that has another part as its first feature).

2 Click to select a face on the mold base.

3 Open a sketch and sketch a line that, when projected, divides the mold base as desired (the parting plane). You can also select a model edge, and click **Convert Entities** to project the edge as a parting line.

4 Click **Insert**, **Cut**, **Extrude** . Leave the **Flip Side to Cut** check box unchecked in the **Extrude Cut Feature** dialog box.

5 Click **OK**. The mold base is cut, exposing the cavity within the mold base. This is one half of the mold.

6 Save this part.

7 To create the other half of the mold, save the part again with another name, then edit the cut. Reverse the direction of the cut by checking the **Flip Side to Cut** check box in the **Extrude Cut Feature** dialog box.
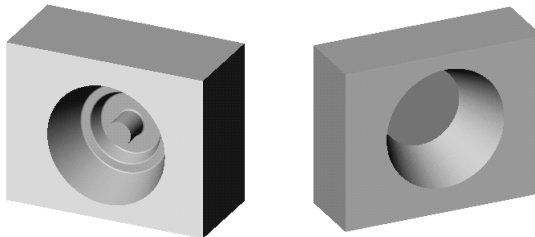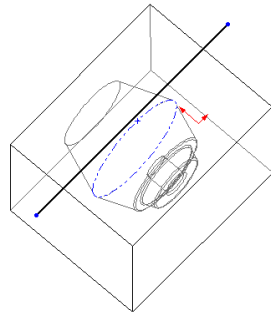
Mold with Complex Parting Lines

Some types of design parts require complex parting lines. When an extruded cut cannot produce the desired parting line, you can use a radiated surface to cut the mold.

1 Create a design part with split lines (or model edges) on the faces where you want to split the mold. See **Split Line** on page 3-11.

2 Following the procedure for a **Simple Mold** on page 7-18, create an interim assembly, insert a cavity in the mold base, and create a derived part for each half of the mold.
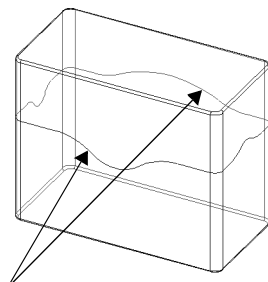
Use a **Split Line** to create edges for cutting the mold.
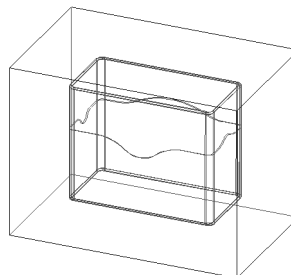
Interim assembly, with a cavity

3 In each derived part, insert a **Radiate Surface**. Select the edges created by the split line (or the model edges), and specify a **Radiate distance** that is large enough to extend the surface beyond the outside faces of the mold base. See **Radiate Surface** on page 3-18.

4 Select the surface, and click **Insert, Cut, With Surface**. In one of the derived mold halves, click **Flip the side to cut away** when you insert the cut. See **To cut a part with a surface:** on page 3-21.
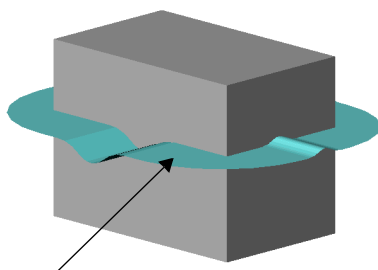
Radiate Surface beyond the faces of the mold base
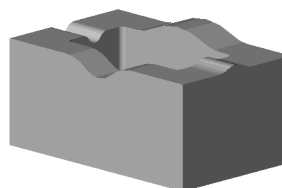
Finished mold half, with surface hidden

Core and Cavity Mold

In the **Simple Mold** on page 7-18, the base of the design part was extruded from a midplane, with an inward draft angle. By parting the mold at the midplane, the design part can easily be removed from the mold.

In this example, the design part is a bowl with drafted sides. To remove the design part from the mold, the parting line needs to be coincident with the top edge of the bowl. A portion of one mold piece fits inside the other piece. You need to cut the mold differently to create the necessary pieces.

To prepare the parts for cutting, follow the instructions beginning on page 7-18:

1 Create an interim assembly containing the mold base and the design part. Mate the components as needed.

2 Create the cavity in the mold base part.

3 Derive two component parts: one for the female portion of the mold and one for the male portion.

Interim assembly

Finished mold

Design component

Mold base

female portion   male portion
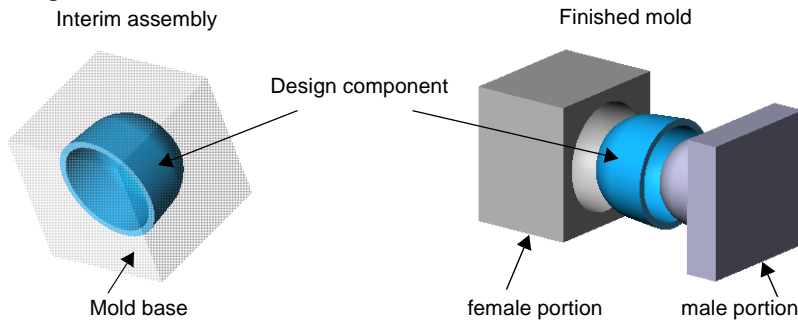
To create the female portion of the mold:

1 In one of the derived parts, open a sketch on a side of the mold base that is perpendicular to the top face of the bowl.

2 Select the edge of the cavity that corresponds to the top edge of the bowl, and click **Convert Entities** ⬜. Drag the ends of the converted line beyond the edges of the mold base.

3 Click **Insert**, **Cut**, **Extrude** 🔳. Select an **End Condition Type** of **Through All**. If the preview arrow is not pointing away from the bowl, click **Flip side to cut**, and click **OK**.
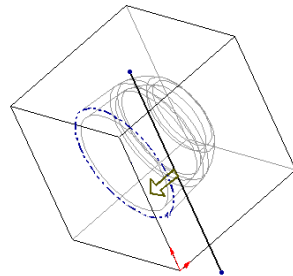
The **Resolve Result Ambiguity** dialog box appears. This lets you choose which piece of the model to keep when there are multiple solutions.

When the cut removes the material above the cavity, there are two possible pieces left: a box with a cup-shaped indention (the material *outside* of the cavity), or a dome-shaped piece (the material *inside* of the cavity).

Select the solution that keeps the material *outside* the cavity.

Section view reveals two possible solutions.

Reject the solution that keeps the material inside the cavity.
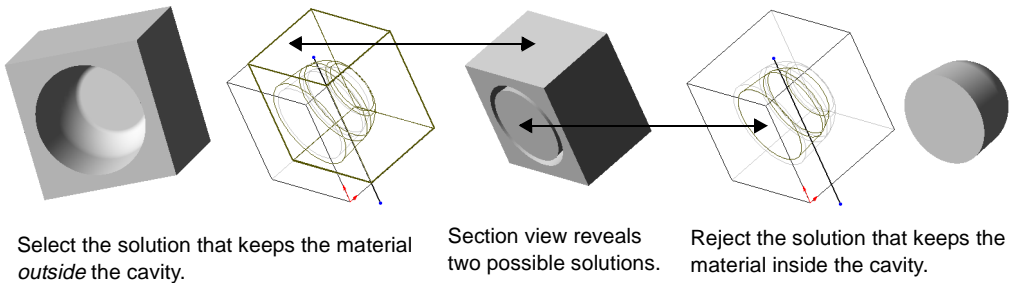
To see each of the solutions, click the **Next** and **Previous** buttons. The highlighted geometry will be *kept* (not *removed*) when the feature is created.

4 Click **OK** when the desired solution is highlighted.

To create the male portion of the mold:

1 In the other derived part, open a sketch on the face of the mold base that is closest to the *bottom* of the bowl. Select the *inside* edge at the *top* of the cavity, and click **Convert Entities** ⬜.

2 Click **Insert**, **Cut**, **Extrude** 🔲. Select an **End Condition Type** of **Up to Next**, and click **OK**. The **Next** faces are the faces of the cavity; this cut exposes those faces. Now you can see the shape on the inside of the mold base.

3 Open another sketch on the same face.

 • Select the model face you are sketching on again, and click **Convert Entities** ⬜ to project the outside edges of the mold base into the sketch.

 • Select the *outside* edge at the *top* of the cavity, and click **Convert Entities** ⬜.

4 Click **Insert**, **Cut**, **Extrude** 🔲. Select an **End Condition Type** of **Up to Surface**, then click the face at the *top* of the cavity. Click **OK**.

The cut removes all the material *between* the sketch loops, up to the selected face.

Convert entities to create two sketch loops

Extrude up to this face
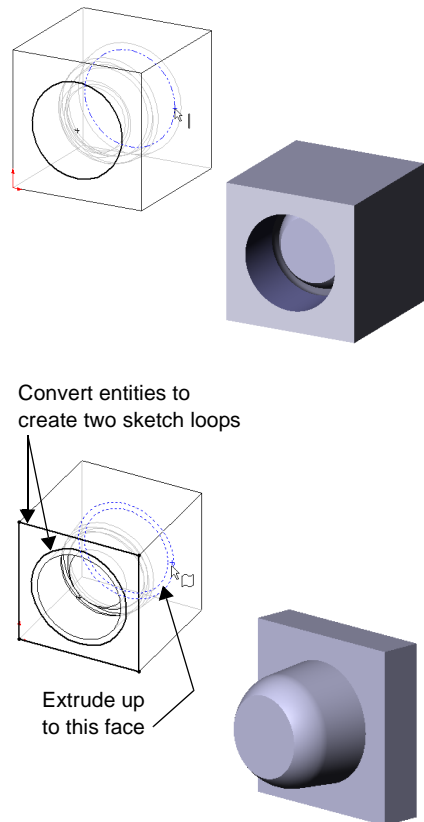
Joining Parts

You can join two or more parts to create a new part. The join operation removes surfaces that intrude into each other's space and creates a single solid volume. Joining makes the model less confusing by removing overlapping solids.

To join parts:

1 Create the parts that you want to join, then create an assembly containing the parts.

2 Position the parts as desired in the assembly. The parts may either touch each other or intrude into one another.
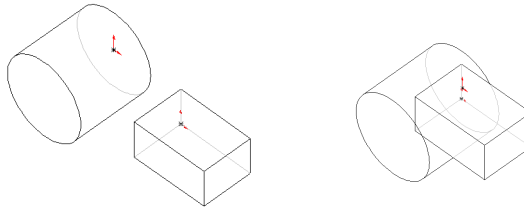
3 **Save** the assembly but do not close the window.

4 Click **Insert, Component, New** to insert a new part into the assembly, and **Save** it.

5 Click a face or plane on one of the components.

6 Click **Insert, Features, Join**. The **Join** dialog box appears.

7 In the FeatureManager design tree, click the components you want to join.

8 Check **Force Surface to Surface Contact** if you want to join parts that just touch each other.

    The system finds all abutting faces that touch but do not intrude into each other's space. It then finds the affected face with the smallest area, extends it into the other component, and fills in the resultant gap.

9 Click **Hide Input Components** to improve graphics rendering.

    This hides the original input components after the join is complete. When the input components are hidden, their icons in the FeatureManager design tree are shown without color and in gray outline.

10 Click **OK**.

11 **Save** the part.

12 Click **Rebuild** to view the newly joined parts.

13 To exit from **Edit Part** mode, right-click anywhere and select **Edit Assembly**.
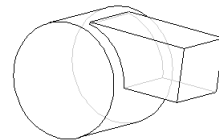
To identify which parts are used to create a joined part, either:

❑ In the FeatureManager design tree, right-click the **Join** feature of the joined component and select **Edit Definition**. You can also right-click the joined component in the model and edit its definition.

❑ Select the joined component, and click **File, Find References.**

❑ Expand the joined component, then expand the **Join** feature to see the absorbed references.

To edit the definition of the joined part to add or remove components:

1   In the FeatureManager design tree, right-click the **Join** feature of the joined part and select **Edit Definition**.

The **Join** dialog box appears. Joined parts are listed in the **Design Component** box.

2   In the FeatureManager design tree, click the part you want to remove from the joined component, or select new components to be joined to the existing component.

3   Click **OK**. The joined component updates to reflect the changes.

> **NOTE:** The joined part is fully associative to the original parts and to the assembly. Any changes made to the original parts are reflected in the joined part. If the parts move relative to each other in the assembly, the joined part changes also.

8

Drawings

You can create 2D drawings of the 3D solid parts and assemblies you design. Parts, assemblies, and drawings are linked documents; any changes that you make to the part or assembly change the drawing document.

> **NOTE:** One-way associativity can be set between drawings and models during the software installation. This permits changes to the drawing that do not change the model. This option can only be reset with a new installation of the software.

Generally, a drawing consists of several views generated from the model. Views can also be created from existing views. For example, a section view is created from an existing drawing view.

This chapter describes:

❑ Setting default drawing and page set-up options

❑ Working with drawing templates and sheets

❑ Creating drawings with standard views

❑ Creating other views, such as detail, section, projected, broken, etc.

❑ Aligning and displaying views

❑ 2D sketching

Dimensions, notes, datums, geometric tolerances, and other notations complete the drawing. Dimensions and other annotations are discussed in Chapter 9, "Detailing."

Drawing and Page Setup Options

You have a variety of options to customize your drawings to your company's standards and to conform to the requirements of your printer or plotter.

To customize your SolidWorks 98Plus drawings, you can:

❑ Select default drawing options by clicking **Tools, Options** and then reviewing the options available on these tabs:

- **Drawings**. Specify the scale, and various display and update options for views. See **Drawings Options** on page A-8.

- **Crosshatch**. Specify the crosshatch patterns used for section views, and for user-defined hatched areas. See **Crosshatch Options** on page A-4.

- **Line Font**. Specify the line weights and styles used for various edge types in the drawing. See **Line Font Options** on page A-20.

- **Detailing**. Specify numerous options related to dimensions and other types of annotations. See **Detailing Options** on page A-4.

❑ Set the page properties to meet your printer or plotter requirements. Click **File**, **Page Setup** to specify:

- Paper margins and orientation

- Line weights

- Scale of printed drawing

- Custom headers and footers

  For more information, see **Page Setup** on page 1-34.

❑ Customize your drawing templates to conform to the standards of your workplace. You can:

- Customize information blocks and text (See **Customizing a Drawing Template** on page 8-5.)

- Change the sheet size and orientation (See **Modifying the Sheet Setup** on page 8-3.)

Creating a Drawing

Drawings consist of one or more views generated from a part or assembly. The part or assembly associated with the drawing must be open and saved before you can create the drawing.

Drawing files have the **.slddrw** extension.

To create a new drawing:

1   Click **File**, **New**, or click ⬚ on the **Standard** toolbar. The **New** dialog box appears.

2   Select **Drawing** and click **OK**. The **Template To Use** dialog box appears.

3   Click one of the following to select a template:

- **Standard Template.** Select a standard sheet size template (for example, **C-Landscape**) from the list.

- **Custom Template.** Click **Browse,** navigate to a custom template on your system or the network, and click **Open**. See **Customizing a Drawing Template** on page 8-5 for information about creating a custom template.

- **No Template**. Select a blank, standard sheet from the **Paper Size** list, or choose the **User Defined** option. If you choose **User Defined**, you must specify the paper size in the **Width** and **Height** boxes.

4   Click **OK**.
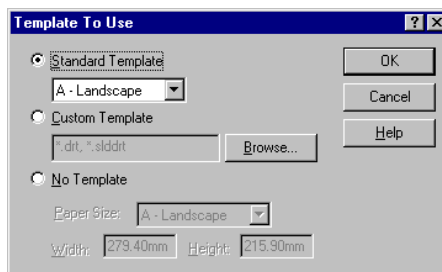
A new drawing document is opened, using the selected template, and the current sheet scale is displayed in the status bar at the bottom of the SolidWorks window.

Modifying the Sheet Setup

You can set up the sheet details when you start a drawing, or later. You can also modify existing sheet details. If you choose **No Template** when you open a new drawing, the default values specified in the **Drawings Options** are used.

To specify sheet details:

1   Right-mouse click the sheet icon in the FeatureManager design tree, any blank area of the drawing sheet, or the sheet tab at the bottom of the drawing window, and select **Properties**. If the drawing is composed of several sheets, select the sheet icon or tab that corresponds to the sheet you want to modify.

2   Make the appropriate entries on the **Sheet Setup** dialog box:

- **Name**. Enter a title in the box.

- **Paper Size**. Select a standard size from the list, or select **User Defined** to specify a custom paper size. If you selected **User Defined**, specify a **Height** and **Width** for the paper.

- **Scale**. Specify the default scale for all views on the sheet.

- **Template**. Select a standard template from the list, or select **Custom** or **None**. If you selected **Custom**, use the **Browse** button to select a custom template.

- **Type of Projection**. Select **First Angle** or the default, **Third Angle**.

- **Next View Label, Next Datum Label.** Specify the letter of the alphabet that will be used.

- **Use custom property values from model shown in**. This option is used only if more than one model document is shown on the sheet. If the drawing has notes that are linked to custom file properties of a model, select which view contains the model whose properties you want to use. If you do not specify otherwise, the properties of the model in the first view added to the sheet are used. See **Linking Notes to Document Properties** on page 8-6.
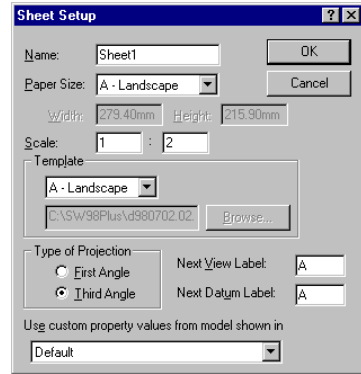
3   Click **OK**.

Multiple Sheets

You can add sheets to the drawing at any time.

To add a sheet:

1   Click **Insert**, **Sheet** or right-mouse click on the sheet tab and select **Add** from the menu. The **Sheet Setup** dialog box appears, with the default name of the next new sheet in sequence, for example, Sheet 2.

2   Set the parameters as described above, and click **OK**.
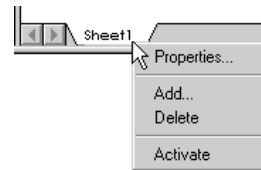
To delete a sheet:

1   Right-mouse click the sheet tab, and select **Delete**.

2   Click **Yes** in the **Confirm Delete** dialog box.

You can also click on the sheet in the FeatureManager design tree or the graphics window, and press the **Delete** key. Click **Yes** in the **Confirm Delete** dialog box.

To view a different sheet:

Click the sheet tab, or right-click the sheet tab and select **Activate**.

Customizing a Drawing Template

You can customize drawing templates to match your company's standard format.

To edit the drawing template:

1 Right-mouse click the sheet or template icon in the FeatureManager design tree, or any blank area on the drawing sheet and select **Edit Template** from the menu.

2 To *edit* existing text in the title blocks, double-click the text.

The **Properties** sheet for **Note** appears.

- Enter new text in the **Note Text** box.

- Modify the **Angle** and **Text Justification** (**Center, Left, Right**) if needed.

- If you want to use a different font, deselect **Use Document's Font**, then click **Font** to select a new font style or size.

- To specify a link to a file or URL, click **Insert Hyperlink** .

- To link the text of the note to a property, click **Link to Property** . See **Linking Notes to Document Properties** on page 8-6.

3 Click **OK** to accept the new font and/or text.

4 To edit the template format by moving, deleting, and adding lines or text in the text blocks:

- To delete, click the line or text and press the **Delete** key.

- To move, click the line or text and drag to a new location.

- To add lines, click the **Tools**, **Sketch Entity**, **Line** or .

- To add text, click on the sheet, then click **Insert**, **Annotations**, **Note** or **A**. Specify the text properties, then drag the text to the desired location.

5 The standard templates supplied with the software include an anchor point for placing the bill of materials in a specific location on the sheet. To set the anchor point, expand the **Template** icon for the sheet. Right-mouse click the **Bill of Materials** icon, select **Set anchor**, then click where you want the anchor point on the sheet.

You can also right-mouse click a sketch point in the template, and select **Set as Anchor**. The anchor point is highlighted in pink, and its visibility toggles when you click the icon.

NOTE: You can add an anchor point to drawings that you created in versions prior to SolidWorks 98Plus.

Right-mouse click the sheet, and select **Edit Template**. Then right-mouse click a sketch point (add one if needed), and select **Set as Anchor**.

If the sheet does not already have a template, you create one when you perform this procedure.

For information about using the anchor point when you insert a BOM, see **Bill of Materials** on page 9-29.

6 To add objects from other applications, such as bitmaps of your company logo, standard note text from a file, etc., click **Insert, Object**.

You may create a new object or insert one from a file. Objects may be *linked* or *embedded* in template files. For more information, see Chapter 14, "Object Linking and Embedding."

To save the template:

Click **File**, **Save Template**.

- To overwrite a **Standard Template**, select the template from the list, and click **OK**.

- To save the template with a new name, click **Custom Template**. Then **Browse** to the desired directory, enter a new template name, and click **Save**. (Template files have the **.slddrt** extension.)

When you save the template, any custom file properties that you have added to the drawing document are saved with the template. The custom properties in the template are added to any new drawing that uses the template.

See **Custom Properties** on page 5-25 for more information.

To end editing the template and return to working with an individual sheet:

Click **Edit, Sheet**, or right-mouse click the sheet or template icon in the FeatureManager design tree, or any blank area of the sheet, and select **Edit Sheet**.

Linking Notes to Document Properties

To automatically insert information in a drawing, you can link note text in the template to document properties.

All SolidWorks documents have the system-defined property **SW-File Name** (the name of the document, without the extension). Additionally, drawings have the following system-defined properties:

Property name	Value
SW-Sheet Scale	scale of the active sheet
SW-Template Size	sheet size of the active sheet
SW-Current Sheet	sheet number of the active sheet
SW-Total Sheets	total number of sheets in the active drawing document
SW-Sheet Name	name of the active sheet

You can link a note to the properties of the model that is shown in the drawing (the **SW-File Name** property, or a user-defined custom property in the model document).

While you are editing the template, a variable for the property name is displayed (**$PRP: "property name"**). When you return to editing the sheet, the value of the property is displayed, if it is found. If the property value cannot be found, the variable for the property name is displayed on the sheet.

To link a note in the template to a property:

1 In the **Note Properties** dialog box, click **Link to Property** ⬛.

2 In the **Link to Property** dialog box, select a **Property Name** from the list. The list contains the names of the system-defined properties, and any custom properties to which you have assigned values in *the active drawing document*.

A linked note may include additional text, and it may include links to more than one property. For example, to display the current sheet number and the total number of sheets, you can add this note:

   SHEET $PRP: "SW-Current Sheet" OF $PRP: "SW-Total Sheets"

On the sheet, the property values are displayed:

   SHEET 1 OF 2 (on the first sheet of a two-sheet drawing)

3 To link the note to the properties of *the model shown in the drawing*, click **External model reference**, then select a **Property Name** from the list. The value of the property for the model shown in *the first view* inserted into the sheet is displayed by default.

If the sheet contains views of more than one model, you can specify which view contains the model whose properties you want to use. Right-mouse click the sheet, select **Properties**, then select the view from the list under **Use custom property values from model shown in.**

If no model is shown in the drawing (for example, when you are creating a new template), you may either select a property from the drawing's list, or add a new custom property. The property is resolved when a model is added to the sheet.

When you save the template, any new properties that you added are saved with the template. The properties are added to any new drawing that uses the template.

4 Click **Rebuild** ⬛ to update the drawing, if necessary.

The Drawing Window

The drawing window includes a FeatureManager design tree that is similar to the design tree in the part and assembly windows. The FeatureManager design tree for drawings consists of a hierarchical list of items pertaining to the drawing. There is an icon for each sheet; under each sheet, there are icons for the template and each view.

A ⊞ symbol beside an item's icon indicates that it contains associated items. Click the ⊞ to expand the item and display its contents.

Standard views contain the feature list for the part or assembly shown in the view. Generated views, such as detail or section views, contain different, view-specific items (detail circles, section lines, etc.).
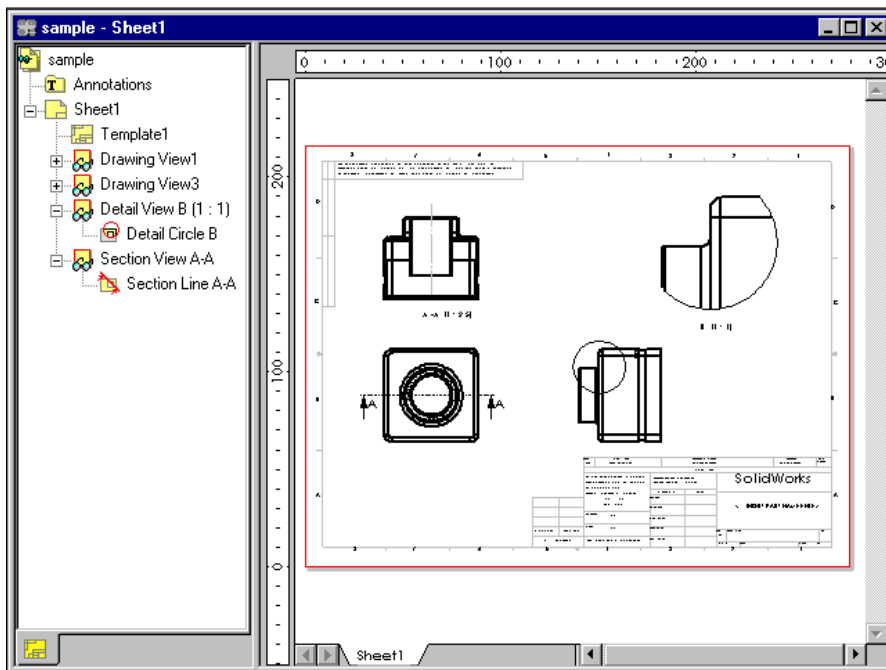
Views of Parts and Assemblies

You can generate drawings of a part or assembly in a variety of view types: **Standard 3 View**, **Projection**, **Named**, and **Relative to Model**. You can also create **Auxiliary**, **Detail**, **Section**, **Aligned Section**, or **Broken** views from existing views. The part or assembly document must be open and saved before you create a drawing of it.

You can also create an **Empty** view for sketching. See **2D Sketching** on page 8-35 for more information.

For some operations, you need to *select* a view; for others, you *activate* a view.

❑ *Selecting* a view allows you to create a projected view, to insert break lines for a broken view, to move the view on the sheet, to resize the view boundary, etc. A selected view has a green border with drag handles. To select a view, click anywhere within the boundary.

❑ *Activating* a view allows you to sketch entities on that view. This is necessary for sketching section lines, defining areas for detail views, etc. To edit the sketch entity, the view to which it belongs must be active. The border of an active view is a red shadowed box.

You can set an option that causes drawing views to activate automatically when the cursor passes over them. Click **Tools, Options, Drawings**, and select **Dynamic Drawing View Activation**.

❑ When checked, the view *closest to the cursor position* is activated automatically. To stop the dynamic activation temporarily, you can lock the focus on a view or on the sheet. Right-mouse click the view or the sheet, and select **Lock View Focus** or **Lock Sheet Focus**.

- **Lock View Focus** allows you to add sketch entities to a view, even when the cursor is close to another view. You can be sure that the items you are adding belong to the view you want.

- **Lock Sheet Focus** allows you to add sketch entities to the *sheet*. Otherwise, the sketch entities belong to the view that is closest to where you begin sketching.

To return to dynamic activation mode, right-mouse click again, and select **Unlock View Focus** (or double-click a different view) or **Unlock Sheet Focus** (or double-click any view).

❑ When unchecked, double-click anywhere within a view's boundary to activate the view, or right-click within the boundary and select **Activate View**. To de-activate a view, either double-click again, or activate a different view, or right-click and select **Activate Sheet**.

View Boundaries

As the cursor passes over the edge of a drawing view, the view boundary is highlighted. The size of the view boundary is calculated automatically, based on the size, shape, and orientation of the model shown in the view. You can adjust the size of the view boundary (enlarging the view boundary may make it easier to select or activate a view). You cannot make the view boundary smaller than the model displayed in the view. View boundaries, and the views they contain, may overlap.

To change the size of the view boundary:

1   Select the view.

2   Move the cursor over a green drag handle at the corner or side of the boundary until it changes to a resize cursor shape.

3   Drag the handle as needed to resize the boundary.
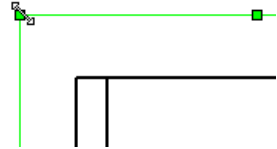
To turn the boundary display on/off:

On the **Drawings** tab of the **Options** dialog box, click **Display drawing view borders.** When checked, the boundary of each view is displayed as a gray box. When unchecked, boundaries are hidden. This option applies to *all* the views in the drawing.

Even if the boundaries are hidden, they still become highlighted when the cursor passes over them.

Standard 3 View

The **Standard 3 View** option under **Insert**, **Drawing View** creates three default orthographic views of a part or assembly displayed at the same time.

There are several ways to create a Standard 3 View drawing.

To create the Standard 3 View:

1   Open the saved part or assembly, and open a new drawing.

2   Click **Insert**, **Drawing View**, **Standard 3 View,** or on the Drawing toolbar. The cursor changes to include a small part symbol.

3   Switch to the part or assembly window (select it by name from the **Window** menu, or tile the windows, then select it), and click anywhere to select.

The drawing window returns to the front (if necessary) and the three views are placed in the drawing.

To create the Standard 3 View (drag-and-drop method):

The default view created when you drag and drop a part or assembly into a drawing is the **Standard 3 View**.

1   Open a new drawing window.

2   Drag a part or assembly document from the Windows Explorer window, and drop it into the drawing window.

– or –

Drag the name from the top of the FeatureManager tree of an open part or assembly document, and drop it into the drawing window

To create the Standard 3 View from a hyperlink in Internet Explorer:

1   In Internet Explorer (version 4.0 or higher), navigate to a location that contains hyperlinks to SolidWorks part files.

For example, on the SolidWorks website, in the Support section, navigate to the Solid Modeling Library page. You may also be able to obtain SolidWorks parts from certain third-party vendors.

2   Drag the hyperlink from the Internet Explorer window, and drop it in an open drawing window. The **Save As** dialog box appears.

3   Navigate to the directory where you want to save the part, enter a new name if desired, and click **Save**.

The part document is saved locally, and the views of the part are added to the drawing.

The lower left view in a third angle projection is the default **Front** view from the part or assembly. For first angle, the front view is at the upper left. See **Modifying the Sheet Setup** on page 8-3 for information about setting the default projection.
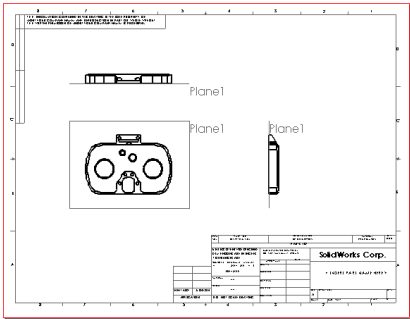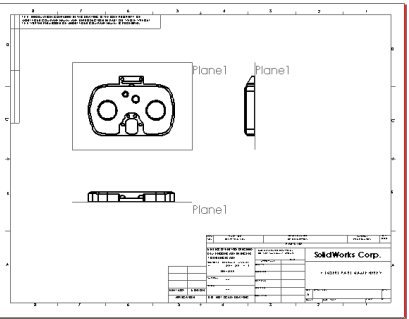
3rd Angle Projection   1st Angle Projection

The alignment of the top and side views is fixed in relation to the front view. The top view can be moved vertically, the side view can be moved horizontally.

For more information about arranging views on a sheet, see **Moving Views** on page 8-29 and **Aligning Views** on page 8-31.

Projection View 🖼

A **Projection** view is created by projecting an orthogonal view, using **First Angle** or **Third Angle** projection, as specified in the **Sheet Setup** dialog.

To create a projection view:

1 Select an existing view by clicking inside its boundary.

2 Click **Insert**, **Drawing View**, **Projection** or 🖼 on the Drawing toolbar. The cursor shape changes to ✛.

As you move the cursor, a preview of the view boundary is displayed, snapped to the nearest projection. To override the snapping behavior as you place the projection view, press the **Ctrl** key as you move the preview around. To resume the snapping behavior while dragging, release the **Ctrl** key.

3 Move the cursor until the view is where you want it to be, then click to place the view.

4 Select a projection direction by clicking beside the selected view in the desired direction. You can project to the left, right, above, or below (in this example, the user clicked to the right of the selected view).

Selected view   Projected view
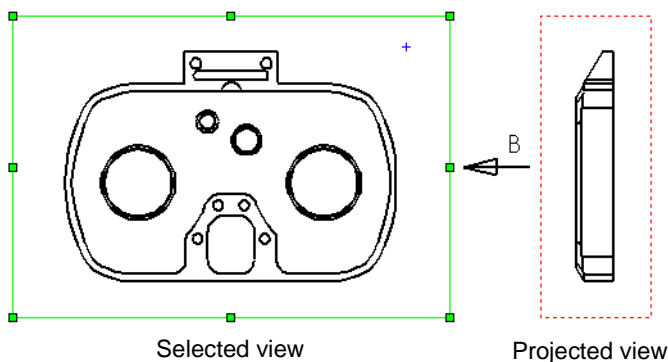
A projection view is placed on the sheet, aligned to the view from which it was created. By default, you can move a projection view only in the direction of the projection. See **View Alignment and Display** on page 8-29 for information about changing the alignment of views.

5 To display an arrow indicating the direction of projection, right-click the projection view on the sheet or in the FeatureManager design tree, and select **Properties**. Click **Display view arrow** and enter a label if desired (maximum of two characters).

Auxiliary View

An **Auxiliary** view is similar to a projection view, but unfolded normal to a reference edge in an existing view.

To create an auxiliary view:

1 Select the reference edge (*not* a horizontal or vertical edge, which would create a standard projection view).

2 Click **Insert**, **Drawing View**, **Auxiliary** or on the Drawing toolbar.

As you move the cursor, a preview of the view boundary is displayed. To override the default alignment as you place the auxiliary view, press the **Ctrl** key as you move the preview around. To resume the snapping behavior while dragging, release the **Ctrl** key.

3 Move the cursor until the view is where you want it to be, then click to place the view.

The auxiliary view is placed on the sheet, aligned to the edge from which it was created. By default, you can move an auxiliary view only in the direction of the projection. See **View Alignment and Display** on page 8-29 for information about changing the alignment of views

A view arrow (or set of arrows, if you use ANSI dimensioning standards) that indicates the direction of projection is added automatically.

Selected edge ——————   Auxiliary view
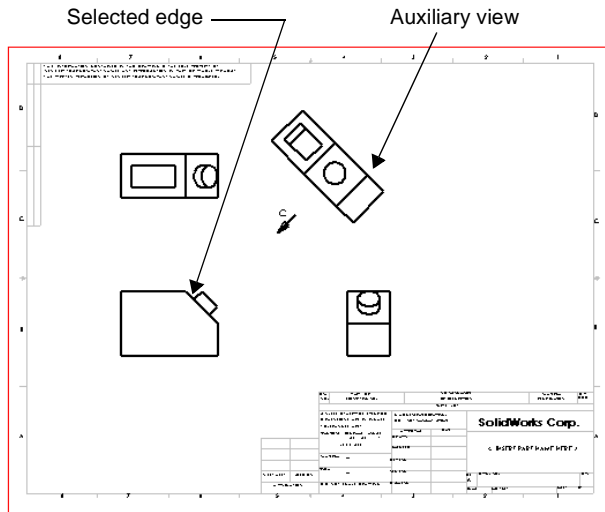
4 To remove the arrow, or to change the label, right-click the auxiliary view on the sheet or in the FeatureManager design tree, and select **Properties**. Click **Display view arrow** and enter a label if desired (maximum of two characters).

Named View ⬚ᴺ

You create a **Named** view by selecting from the view names in the model document. Named views include:

❑ Standard orthographic views (**Front**, **Top**, **Isometric**, and so on)

❑ **Current** model view

❑ Custom views that you created in the model by zooming, rotating the model, enabling **Perspective** 🔲 if desired, then saving the view by name. The entire part or assembly is displayed at the scale of the selected view, even if the selected view orientation in the part or assembly displays a zoomed-in view.

For more information about creating a named view in the **View Orientation** dialog box, refer to **Using Named Views** on page 1-30.

To insert a named view:

1 In the drawing, click **Insert**, **Drawing View**, **Named View** or ⬚ᴺ on the Drawing toolbar. The cursor changes to 🔖.

2 Switch to the part or assembly window (select it by name from the **Window** menu), and click anywhere.

The **Drawing View - Named View** dialog box appears. The list contains the names of all the standard views, the **Current** view, and any custom views you created in the model.
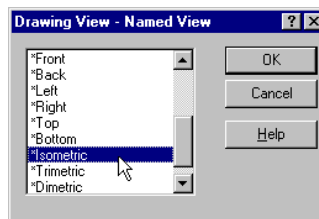
3 Double-click a named view from the list, or select a named view and click **OK**. The cursor changes to ✛.

4 Switch to the drawing window, and click where you want to place the view. Named views are not aligned to other views by default, and may be moved freely to any location on the drawing sheet.

Default Front View

Isometric View

Saved Named View or Current View
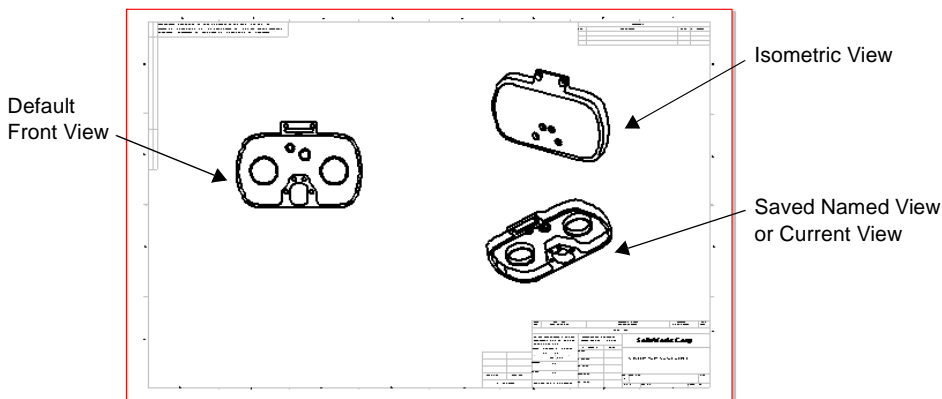
Relative to Model View

A **Relative to Model** view is an orthographic view defined by two orthogonal faces or planes in the model and the specification of their respective orientations. This view type is also used to set the first orthographic view in a drawing if it is to be something other than a default view.

To insert a relative to model view:

1   Click **Insert**, **Drawing View**, **Relative To Model** or on the Drawing toolbar. The cursor changes to the model face selection shape.

2   Click a face in the model window that you want to have a specific orientation.

   The **Drawing View Orientation** dialog box appears.

3   Select an orientation (**Front**, **Top**, **Left**, etc.), and click **OK**.

4   Select another face, *perpendicular* to the first.

5   Select the orientation for this face from the **Drawing View Orientation** list.

6   Click with the cursor on the drawing sheet where you want to place the view. Relative to model views are not aligned to other views by default, and may be moved freely to any location on the drawing sheet.
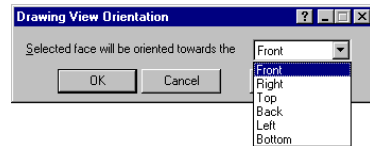
> **NOTE:** Tiling the drawing and model windows may be easier than switching windows when you need to select the model faces and view location.

Example: A true view of an angled face perpendicular to the front face of a model is created in this manner. The angled face is specified as **Front** and the front face as **Left**. If the angle of the face in the model changes, the views update to maintain the orientation as originally specified.
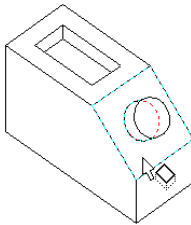
Click the angled face, and select **Front**
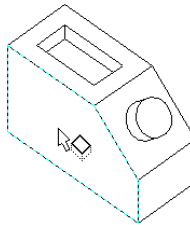
Click a perpendicular face, and select **Left**
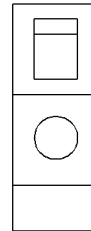
Resulting view, **Relative to Model**

Detail View ⊙

You create a **Detail** view to show a portion of a view, usually at an enlarged scale. This detail may be of an orthographic or 3D view or a section view.

To create a detail view:

1 Activate an existing view (see page 8-9).

2 Using a sketch entity tool, create a closed profile around the area to be detailed. (The profile is usually, but not necessarily a circle.)

3 Select one entity of the profile.

4 Click **Insert**, **Drawing View**, **Detail** or ⊙ on the Drawing toolbar. As you move the mouse, a preview of the view boundary is displayed.

5 Click when the view is where you want to place the labelled detail view.

The note on the detail view includes the letter label and scale.

Sketched detail circle   Detail circle label   Detail view note

Active view   Detail view
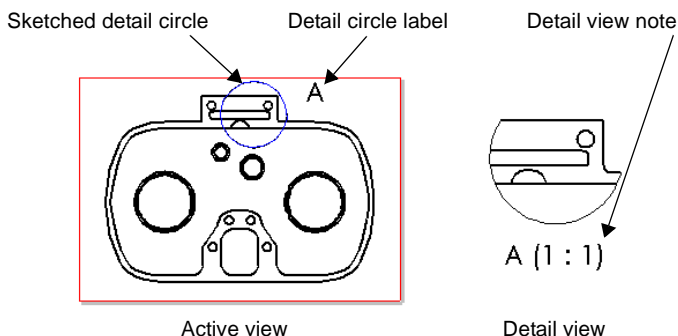
Detail views are not aligned to other views by default, and may be moved freely to any location on the drawing sheet.

To modify a detail view:

❑ To change the scale of the detail view, right-click the view on the sheet or in the FeatureManager design tree, and select **Properties**. Under **Scale**, *deselect* **Use sheet's scale**, enter a new scale ratio, and click **OK**.

❑ To change the detail view note text, double-click the text (or right-click the text and select **Properties**), and modify it as needed in the **Note** dialog box.

• If you change the label in the note, the label on the detail circle updates.

• If you change the scale in the note, the detail view scale updates.

❑ To change only the detail circle label, double-click it, enter the new label, and press **Enter**. The detail view note updates.

❑ To change the detail circle label font and letter, right-click the letter and select **Properties**. Enter a new letter if needed. Deselect **Use Document's Font**, click **Font,** then choose a new size or style.

❑ To move the note or the detail circle label, drag the item with the mouse.
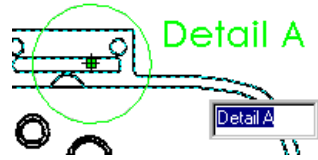
To redefine the cutting edge of the detail circle:

1 Right-mouse click on the sketched detail circle and select **Edit Sketch**.

2 Click and drag the profile to change the shape or size. Note the cursor shape for the detail circle.

3 Click **Edit, Rebuild** or ⬛ on the Standard toolbar to update the detail view.

You can also reshape the detail circle by dragging it. The view automatically updates.

To specify the line font for the detail view border:

1 Click **Tools, Options**.

2 On the **Line Font** tab, under **Type of Edge**, click **Detail Border**, then choose a **Line Style** and **Line Weight**.

3 Click **OK**.

Broken View

You can use a broken view (or interrupted view) on the drawing of a long part that has a uniform cross-section. This makes it possible to display the part in a larger scale on a smaller size drawing sheet.

Reference dimensions and model dimensions associated with the broken area reflect the actual model values. For information about dimensioning a drawing view, see Chapter 9, "Detailing."

To create a broken view:

1 Select the drawing view and click **Insert, Vertical Break** (or **Horizontal Break**). Two break lines appear in the view.

   You can add more than one set of break lines to a view; however, all breaks must be in the same direction. You can only add breaks to an *unbroken* view.

2 Drag the break lines to the places on the part where you want the breaks to occur.

3 Right-mouse click inside the highlighted boundary and select **Break View** from the menu.

   The part is displayed with a gap in the part geometry. In addition to model geometry, broken views also support cosmetic threads, axes, and sketch entities (section lines, detail circles, etc.).
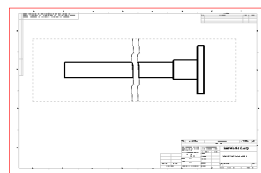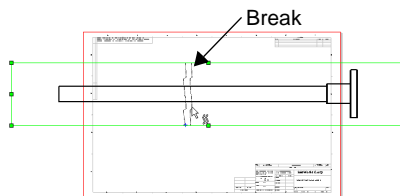
To restore a broken view to its unbroken state:

Right-mouse click the broken view, and select **Un-Break View**.

To modify a broken view:

• To change the position of the break, drag the break lines.

• To change the width of the break gap, click **Tools**, **Options,** and select the **Detailing** tab. Enter a new value in the **Break Gap** box under **Break Lines**.

• To change the shape of the break lines, right-mouse click a break line and select a line style from the menu: **Straight Cut**, **Curve Cut, Zig Zag Cut**, or **Small Zig Zag Cut.**
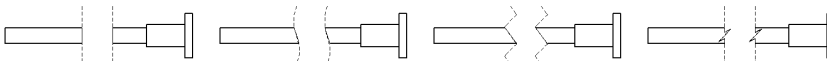
Section View

You create a **Section** view by partitioning the view with a section line. The view may be a straight cut section or an offset section defined by a stepped section line. The section line may also include arcs.

To create a section view:

1 Activate an existing drawing view (double-click the view, or right-click and select **Activate View**).

2 Click **Tools**, **Sketch Entity**, **Centerline** or **Line**, or click ⬚ or ⬚ on the Sketch Tools toolbar.

3 Sketch a single or stepped centerline or line through the model in the active view.

4 Select the sketched centerline or line (if it is not already selected). For a stepped line, you only need to select one segment.

5 Click **Insert**, **Make Section Line**.

Notice the arrows, indicating the direction of the cut. If necessary, double-click on the section line to reverse the direction of the arrows.

Now you can edit other section line properties that control the resulting view, if you wish. See **Modifying Section Views** on page 8-22 for more information.

6 With the section line still selected, click **Insert**, **Drawing View**, **Section** or ⬚ on the Drawing toolbar.
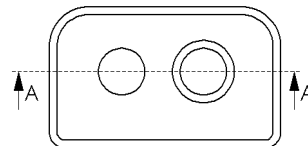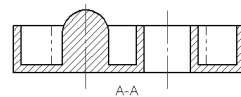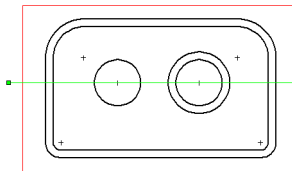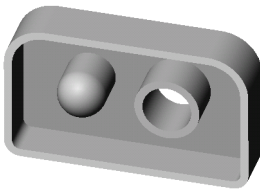
NOTE: If you prefer to create the section line and the default section view in a single step, skip Steps 5 and 6. Instead, after you select the sketched segment, click ⬚ or **Insert, Drawing View, Section**.

As you move the cursor, a preview of the view boundary is displayed. By default, the view is aligned in the direction of the arrows on the section line. To override the default alignment as you place the view, press the **Ctrl** key. To return to the default alignment while dragging, release the **Ctrl** key.

In its default alignment, a section view can only move in the direction of arrows. See **View Alignment and Display** on page 8-29 for information about changing the alignment of views.

Aligned Section View

You can create an aligned section view through a model, or portion of a model, that is aligned with a selected section line segment.

To create an aligned section view:

1 Activate an existing drawing view (double-click the view, or right-click and select **Activate View**).

2 Click **Tools**, **Sketch Entity**, **Centerline** or **Line**, or click ⊡ or ◲ from the **Sketch Tools** toolbar.

3 Sketch the section line. The section line should consist of two connected lines at an angle to each other.

4 Select the segment of the sketched section line to which you want to align the view.

5 Click **Insert**, **Make Section Line**.

Notice the arrows, indicating the direction of the cut. If necessary, double-click on the section line to reverse the direction of the arrows.

Now you can edit other section line properties that control the resulting view, if you wish. See **Modifying Section Views** on page 8-22 for more information.

6 With the section line still selected, click **Insert, Drawing View, Aligned Section** or ⊡ on the Drawing toolbar.

NOTE: If you prefer to create the section line and the default aligned section view in a single step, skip Steps 5 and 6. Instead, after you select the sketched section line segment, click ⊡ or **Insert, Drawing View, Aligned Section**.

You cannot change the angle of the line that you selected to align the view; you can change its position, however.
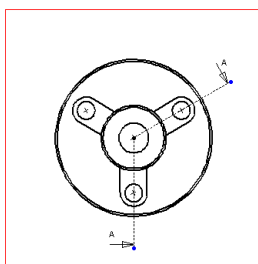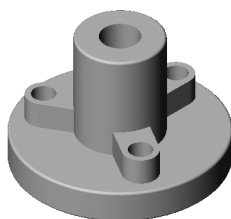
Active view with section line   Aligned section
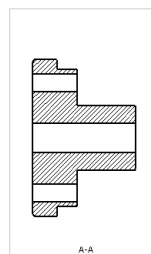
As you move the cursor, a preview of the view boundary is displayed. By default, the view is aligned in the direction of the arrows on the selected segment of the section line. To override the default alignment as you place the view, press the **Ctrl** key. To return to the default alignment while dragging, release the **Ctrl** key.

In its default alignment, an aligned section view can only move in the direction of arrows on the selected segment. See **View Alignment and Display** on page 8-29 for information about changing the alignment of views.

Modifying Section Views

❑ To edit the letter label on the section line, double-click the letter label, enter a new letter, and press **Enter**.

❑ To edit the note on the section view, double-click the text, make any necessary changes (edit the text, add symbols, change the font, etc.) in the **Note** dialog box, and click **OK**.

❑ To edit the location of section lines, drag them as needed. You can also right-mouse click the section line and select **Edit Sketch**, then make changes.

❑ To change the cut direction as indicated by the arrows, double-click anywhere along the section line.

❑ After changing the cut direction or editing the section lines, update the view using one of these methods:

• Click ![icon] or **Edit**, **Rebuild**.

• Right-mouse click the section view and select **Update View**.

• Select the view and click **Update View** ![icon] .

Modifying the crosshatch

If you specify a crosshatch pattern in a *part* document, that pattern is used for section views created for the part (and for instances of the part in assemblies). Otherwise, section views use the default crosshatch pattern, as defined in **Tools**, **Options, Crosshatch**. Changes to the default crosshatch pattern selection take effect in the *next* SolidWorks session. See **Crosshatch Options** on page A-4 for more information.

To modify the crosshatch pattern for a section view:

1 Right-mouse click the face, or **Ctrl**-click multiple faces, and select **Crosshatch Properties**.

2 Specify the **Pattern**, **Scale** and **Angle**, and observe the **Preview**.

3 Under **Apply To**, choose whether to change only the selected **Region**, the section faces for the entire **Component** (in a section view of an assembly), or all the section faces in the **View**.

4 To remove the crosshatch for a layer (**Region**, **Component**, or **View**), select the layer in the **Apply To** list, and click **Remove Crosshatch**. (This option is only available for layers that can logically be removed.)

The crosshatch is removed a layer at a time. For example, if you remove the crosshatch from a region, any crosshatch applied earlier to the whole component is revealed. If you remove crosshatch from a component, the default crosshatch for the view is revealed.

5 Click **Apply** or **OK** to accept the changes, or click **Cancel**.

Modifying the centerline display

Centerlines are automatically added where needed (holes, circular extruded bosses or cuts, and revolved features) in a section view. To specify the distance to extend the centerline beyond the model edge, click **Tools, Options, Detailing,** and under **Dimensioning Standard**, enter a value for **Centerline extension**.

You can modify the length of the centerlines by dragging the endpoints. If you do not want the centerlines, you can either delete them from the section view, or you can turn off the display by clicking **View**, and *deselecting* **Axes**.

Modifying the section line properties

You can also make changes to a **Section View** or **Aligned Section View** by editing the section line properties. You can set these properties either before or after creating the section view.

If you want to set these properties *before* creating the view, sketch the section line, select it, then click **Insert, Make Section Line**. This makes the sketched line into a section line without automatically creating the associated view. Then right-mouse click the section line and select **Properties**.

If you want to modify the section view *after* it is created, right-mouse click the section line (not the view) and select **Properties**. You need to click 🔟 or **Edit**, **Rebuild**, or right-click the section view and select **Update view**, after modifying section line properties in order to update the view.

These two options affect only the labels:

- **Label.** Change the letter that is associated with this section line by selecting the letter and entering a new label.
- **Use Document's Font.** When checked, the section line text uses the same font as the rest of the document. When unchecked, you can click the **Font** button and specify the font style and size to use.

The remaining options in the **Section Line** dialog box control the representation of the model in the resulting view. They may be used individually or in combination, for either section views or aligned section views.
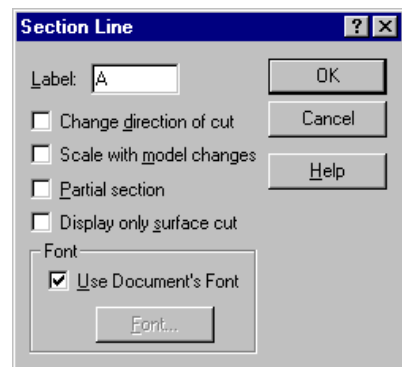
Experiment until you get the results that you want. Remember to update the view after you make changes to the **Section Line Properties**.

The examples that follow are based on this model and the view containing the section line shown here.
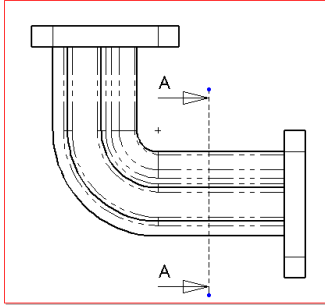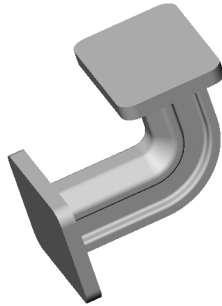
- **Change direction of cut.** Reverse the direction of the section line cut. This has the same effect as double-clicking the section line.

- **Scale with model changes.** The section line changes size parametrically with the model. This could result in a section cut that is not what you intended, however. The default for this option is off.

- **Partial section.** If the section line does not go all the way across the view, selecting this option causes the section view to include only the portion of the model crossed.

Change direction of cut
checked for both views
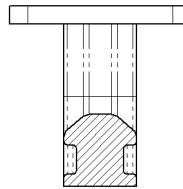
Partial section
unchecked
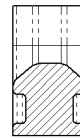
Partial section
checked

- **Display only surface cut.** A simplified view, with surfaces behind the section plane removed. Only the surface(s) actually cut by the section line are displayed in the section view.
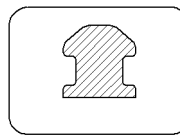
Display only surface cut
unchecked

Display only surface cut
checked

Section Views of Assemblies

When you create a section (or aligned section) view of an assembly, you can exclude (leave uncut) selected components by specifying the **Section Scope**.

To exclude components from a section view of an assembly:

1 Activate a view, and create a section line as described in **Section View** on page 8-20 or **Aligned Section View** on page 8-21.

2 Click **Insert, Drawing View, Section** [icon], or **Aligned Section** [icon]. The **Section Scope** dialog box appears.

3 On the active drawing view, or in the FeatureManager design tree, click the components that you want to exclude from the section view. The selected components are listed in the **Excluded Components** box.

NOTE: When selecting components in the FeatureManager design tree, be sure to select them from the *active view*.

To remove a component from the list, select the component again.

Both components included   Shaft excluded
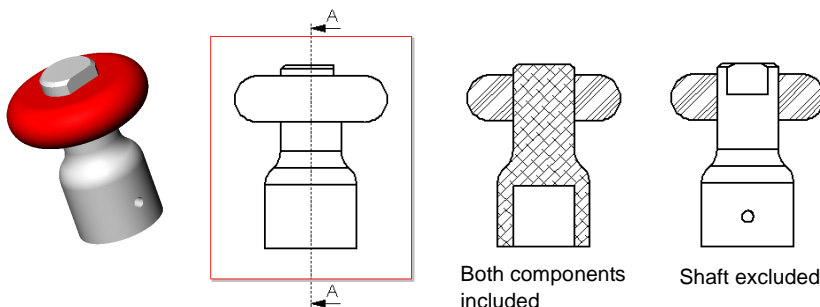
4 Set the **Exclusion Status** if the selected component is used more than once in the assembly (for example, if it is a member of a pattern, or if it is used as a component of more than one subassembly):

- If you want to exclude *all the instances* of the selected component in the assembly, click **Don't Cut All Instances**. In the resulting view, all instances of the selected component will be left uncut.

- If you want to exclude *only the selected instance*, leave this unchecked. In the resulting view, only the selected instance will be uncut; all others will be cut.

5 If you want the software to automatically adjust for neighboring components with the same crosshatch pattern, click **Auto Hatching.** The angle used in the pattern automatically alternates in 90 degree increments. If there are still adjacent section faces with the same pattern, the pattern spacing is adjusted.
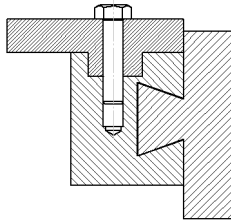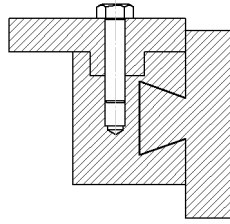
Auto Hatching ON   Auto Hatching OFF

If you do not use auto hatching, you can manually adjust the angle and spacing on adjacent components. See **Modifying the crosshatch** on page 8-22.

NOTE: A detail view created from an assembly section view inherits the crosshatch patterns of its parent view.

---

6   Click **OK**.

In the resulting section view, excluded components are not cut. They may be shown in the view, or you may remove them from the view by setting the section view property **Display only surface cut**. See **Modifying Section Views** on page 8-22 for more information.

To edit the section scope for the view:

Right-mouse click the *section* view, either on the sheet or in the FeatureManager design tree, select **Properties**, and click the **Section Scope** tab of the **Drawing View Properties** dialog box.

- To add components to the **Excluded Components** list, click the components in the section view (*not* the parent view).

- To remove a component from the list, select the component again.

- To change the **Exclusion Status** for components that are used more than once in the assembly, select the component in the **Excluded Components** list, then click **Don't Cut All Instances**.

View Properties

The **Drawing View Properties** sheet provides information about the drawing view and the model or assembly that it represents, the view's scale, dimension type, and configuration.

To examine or modify drawing view properties:

Right-click a drawing view on the sheet or in the FeatureManager design tree, and select **Properties**.

❑ **View Information**. Displays the name and type of the selected view.

❑ **Scale**. To change the scale of the view, deselect **Use sheet's scale**, then change the scale ratio value in the **Scale** boxes. The default scale ratio is 1:2.

❑ **Model Information**. Displays the model name and path to the document. If the selected view is a **Named view**, you can specify a different view orientation. Select a standard or user-defined view from the list.
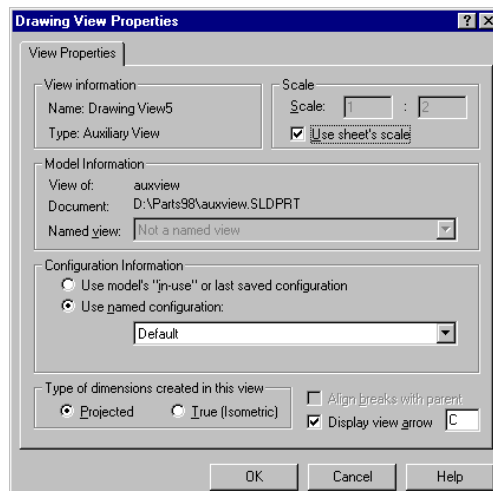
❑ **Configuration Information**. To specify the model configuration to use, click **Use model's "in-use" or last saved configuration**, or click **Use named configuration** then select an existing configuration from the list.

❑ **Show in exploded state**. To use the exploded view of an assembly, select this option. (You must create an exploded view in the assembly document first.)

❑ **Type of dimensions created in this view**. To specify the dimension type, select either **Projected** or **True (Isometric).**

❑ **Align breaks with parent**. To align a broken view with the broken view that it was derived from, select this option. (Click **Edit, Rebuild**, if necessary.)

❑ **Display view arrow**. To display an arrow indicating the direction of projection, select this option and specify the label (two characters maximum).

Updating Views

The update mode for all the views of a new drawing is controlled by the option **Automatic drawing view update when model changes** on the **Drawing** tab of the **Tools, Options** dialog box. When checked (the default), all views are updated automatically when the model changes. When unchecked, you update the views manually. You can change this option for the current drawing at any time.

In manual view regeneration mode, you can update all the views at once when you are ready, or update them one by one. When the model changes, any drawing views that need to be updated are marked with a light gray crosshatch pattern.

> **NOTE:** You cannot activate or edit a drawing view that needs to be updated.

To change the update mode in the current drawing:

Right-mouse click the Drawing icon at the top of the FeatureManager design tree, and click **Auto view regeneration**. If there is no check mark, manual view regeneration is enabled.

To manually update drawing views:

1   Change to manual view regeneration mode as described above.

2   To update all the views at once, use one of these methods:

- Click **Edit, Update All Views.**
- Click **Edit, Rebuild** or ⑧ to update any views that need to be updated.
- Press **Ctrl-Q** to force a regeneration of the entire drawing.
- Right-mouse click the Drawing icon at the top of the FeatureManager design tree, and click **Update All Views**.

3   To update a single drawing view, use one of these methods:

- Select the view, then click **Edit, Update View,** or click ⑧ on the Drawing toolbar. (Use **Tools, Customize** to add the **Update View** icon to the toolbar, if necessary.)
- Right-mouse click the view, and select **Update View**.

You may select the view either in the FeatureManager design tree or on the drawing sheet for either method. You also can **Ctrl**-click several views at once for update in this way.

View Alignment and Display

Moving Views

To move a view:

Move the mouse over the boundary of a view, and drag the view to a new location when the boundary is highlighted. You can also select a view, then move the cursor over the boundary until the move cursor ⬚⬚ appears, then drag. Note the following exceptions:

❑ **Standard 3 View**. The alignment of the front view is fixed in relation to the other two views. When you move it, the other views also move. The other two views can move independently, but only horizontally or vertically, toward or away from the front view.

❑ **Auxiliary, Projection, Section,** and **Aligned Section** views are aligned with the parent views from which they are created, and only move in the direction of the projection.

❑ **Broken** views respect the alignment of the view before it was broken.

❑ **Named**, **Detail, Relative to Model,** and **Empty** views may be moved freely on the sheet; they are not aligned to any other views by default.

To break a view's default alignment:

For views that have a default alignment, you can break the alignment and move the views independently.

Right-click inside the view boundary (not on the model), and select **Alignment, Break Alignment** from the menu, or click **Tools, Align Drawing View, Break Alignment**.

To return a view to its default alignment:

You can return a view whose alignment you have broken to its original alignment.

Right-click inside the view boundary (not on the model), and select **Alignment, Default Alignment**, or click **Tools, Align Drawing View, Default Alignment**.

Copying and Pasting Views

You can cut, copy, and paste drawing views from one sheet to another in the same drawing, and from one drawing document to another.

You can also drag and drop views from sheet to sheet within a drawing (but not to another drawing document).

To perform these operations on multiple views at once, hold the **Ctrl** key as you select the views.

To cut/copy/paste a drawing view:

1 Select the view either on the sheet or in the FeatureManager design tree.

2 Click **Cut** 🔪 or **Copy** 📋, or use the selections on the **Edit** menu.

3 Change to the target sheet, click where you want to paste the view, and click **Paste** 📋 or **Edit, Paste**.

> **NOTE:** If you want to copy a detail or section view from one drawing to another, you must also copy the parent view from which you created it. You can either copy the parent view first, or you can copy both the parent and the detail/section view at the same time.

To drag and drop a drawing view:

1 Select a view on the active drawing sheet, or select the view's icon in the FeatureManager design tree.

2 To *move* the view, hold the **Shift** key, then drag the selected view or its icon, and drop it below the sheet icon for the target sheet.

To *copy* the view, hold the **Ctrl** key, then drag and drop the view or its icon below the sheet icon for the target sheet.

Aligning Views

You can change the alignment of views that are not aligned by default, or views whose alignment you have broken as described in the previous section. You can rotate a view based on a linear model edge, or you can align one view to another.

To rotate a drawing view:

1 Select a linear model edge in a drawing view.

2 Click **Tools**, **Align Drawing View**, **Horizontal Edge** or **Vertical Edge**.

The view rotates until the selected edge is oriented horizontally or vertically.

If there are any views projected from a view you change this way, they update to maintain their projection.

Selected edge
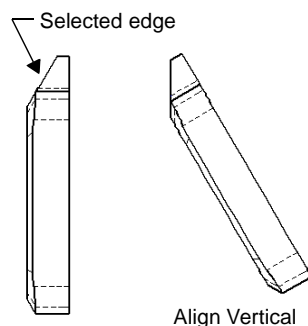
Align Vertical

To align one drawing view to another:

1 Select a drawing view, and click **Tools, Align Drawing View, Horizontal to another view** or **Vertical to another view**.

– or –

Right-click a drawing view, and select **Alignment, Align Horizontal** or **Align Vertical**.

The cursor shape changes to ⬚.

2 Select the reference view to which you want to align.

The centers of the views are aligned along the chosen direction. The alignment is maintained if you move the reference view.

To break an alignment created this way, right-click the view and select **Alignment, Break Alignment** from the menu, or click **Tools, Align Drawing View, Break Alignment**.

Tangent Edges

To specify the default for tangent edge display in drawing views, select the appropriate option under **Tools, Options, Drawings**. You can display tangent edges as visible lines, or with a different line font (as specified on the **Line Font** tab of the **Options** dialog box), or you can remove them from the display.
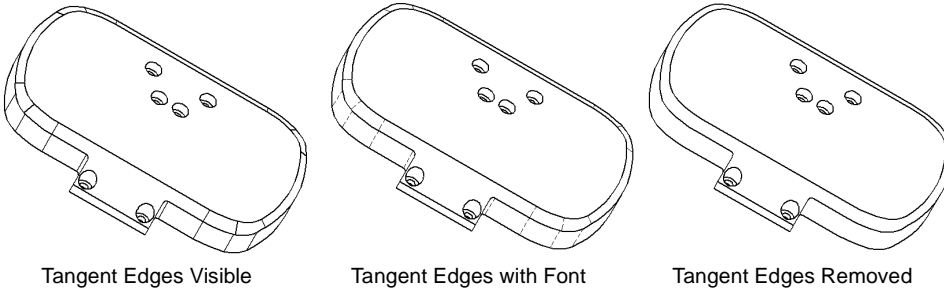
Tangent Edges Visible   Tangent Edges with Font   Tangent Edges Removed

If you need to select hidden (removed) tangent edges, click **Tools, Options, Drawings**, and click **Select hidden entities**.

You can modify the display of tangent edges for a selected view.

To modify the display of tangent edges in a drawing view:

Right-mouse click within the view boundary (not on the model) and select **Tangent Edge,** then **Tangent Edges Visible, Tangent Edges with Font**, or **Tangent Edges Removed**.

Line Format

The tools on the Line Format toolbar let you modify the appearance of individual lines, edges, and sketch entities in a drawing.

To change the format of a line, edge, or sketch entity:

1   Select the line, edge, or sketch entity you want to change. To select more than one line at a time, press the **Ctrl** key as you select.

2   Click a tool on the Line Format toolbar and change the format as necessary:

- **Line Color.** Click 🖉, choose a color from the palette, and click **OK**.

- **Line Thickness.** Click ☰, and choose a line thickness. As you move the mouse over the menu, the thickness name (**Thin**, **Normal**, **Thick**, etc.) is displayed in the Status bar. Corresponding line weights for printing are defined in **File, Page Setup**, on the **Printer** tab, under **Line Weights**.

- **Line Style.** Click 🛒, and choose a style. As you move the mouse over the menu, the corresponding line style name (**Solid**, **Dashed**, **Phantom**, etc.) is displayed in the Status bar.

To return a line to its original format, right-mouse click the line, and select **Reset line font**.

View Visibility

You can use **Hide View** to hide an entire view while working on a drawing. Once hidden, you can use **Show View** to see that view again. If you hide a view that has a dependent detail or section view, you are prompted with the option of hiding the dependent views as well.

You also can selectively hide and show edges in a drawing view.

To hide/show views:

1 Right-mouse click the view on the sheet or in the FeatureManager design tree, and select **Hide View**.

2 To re-display a view, right-mouse click the view, and select **Show View**.

> **NOTE:** The views that you hide are invisible unless the view is active (the standard red outline is visible), or the menu selection **View, Show Hidden Views** is checked (the view boundary is displayed with a gray **X**). The cursor shape changes to and the view boundary highlights when the cursor passes over a hidden view.

To hide/show edges:

Right-mouse click an edge in a drawing view, and select **Hide Edge**. The edge is removed from the display, but it still is highlighted when the cursor passes over it.

To display the edge again, right-mouse click the edge, and select **Show Edge**.

Printing a Drawing

You can print or plot an entire drawing sheet, or just a selected area of the sheet. For basic information about printing and plotting any SolidWorks document, see Chapter 1, "SolidWorks Fundamentals."

To print the whole drawing sheet:

1 Click **File, Print**. In the **Print** dialog box, under **Print range**, click **All** or specify the **Pages** you want to print.

2 Under **Scale**, check **Scale To Fit** to print the entire sheet on the page, or deselect the option and specify a **Scale** value (by percent).

3 Click **OK**.

To print a selected area of a drawing sheet:

1 Click **File, Print**. In the **Print** dialog box, under **Print range**, click **Selection**, and click **OK**.

The **Print Selection** dialog box appears, and a selection frame is displayed on the drawing sheet. The frame reflects the current printer settings (page size, orientation, etc.) as defined under **File, Page Setup, Print Setup**.

2   Select a scale factor to apply to the selected area.

- **Model scale (1:1).** This is the default. This means that the selected area is printed at *actual* size; a model dimension of 100mm is printed at 100mm. The default sheet scale is used to calculate the correct print size; therefore, for views that use a scale different than the default sheet scale, you may need to use a custom scale to achieve the desired result.

- **Sheet scale (n:n).** The selected area is printed as it appears on the full sheet. If the drawing size and the paper size are the same, the entire sheet is printed. Otherwise, only the selected area is printed, as it would appear on the full sheet.

- **Custom scale.** The selected area is printed at a scale factor you define. Enter the values you want in the boxes, then click **Apply scale**.

When you change the scale factor, the selection frame size changes accordingly.

3   Drag the selection frame to position it over the area you want to print. You can pan or zoom the view, or change the sheet while the selection frame is displayed.

4   Click **OK** to print the selected area.

Sending Drawing Documents

You can send a drawing document to another system using electronic mail. You can choose to send only the drawing itself, or you can send the drawing and all referenced documents. This function uses the electronic mail application installed on your system.

To mail a drawing document to another computer:

1   With a drawing document active, click **File, Send To**.

2   In the **Send Mail** dialog box, click a radio button to select one of the following:

- Send the current document only.

- Send the current document and the other document(s) that it references.

A list of the referenced documents, the number of documents, and the combined size of the documents is displayed.

> **NOTE:** The size of the combined documents may be important to you because mailing large size documents may cause transmission problems.

If you choose to send the drawing and referenced documents, you do not have to send all the documents. You can:

- Click a filename you do not want to send, and click **Remove**.

- Click a removed filename and click **Attach** to replace the file.

- To return all the removed files, click **Attach All**.

3   Click **OK** to send the document(s).

4   Enter your mail password, the mailing address, and any other information requested by your mail application.

2D Sketching

You can create drawing geometry using 2D sketched geometry only, without reference to existing models or assemblies. This sketched geometry may be controlled by relations (collinear, parallel, tangent, and so on), as well as parametric dimensions.

Before starting to sketch geometry, it is often desirable to display a grid. Right-mouse click the active drawing sheet and select **Display Grid** from the menu.

To start sketching, click one of the icons on the **Sketch Tools** toolbar or click **Tools**, **Sketch Entity**, and select a tool. Then you can:

- Create geometry by selecting **Line**, **Centerpoint Arc, Tangent Arc, 3 Pt Arc, Circle, Ellipse, Centerpoint Ellipse, Spline, Rectangle, Point**, or **Centerline.**
- Reference existing geometry by selecting **Convert Entities,** which places coincident copies of selected edges to reference from.
- Copy geometry about a centerline using the **Pattern/Mirror** tool.
- Modify the sketch using the **Fillet** and **Trim/Extend** tools.

When you dimension sketched geometry, the geometry automatically updates when you modify dimensions and then click **Edit**, **Rebuild**.

You can move, rotate, or scale sketched geometry on a drawing by using the **Modify** function. Click **Tools, Sketch Tools, Modify** or [icon] on the Sketch toolbar. The behavior of **Modify** in a drawing is the same as its behavior in a sketch. For more information about using **Modify**, see **Modify Sketch** on page 2-19.

To define relations on sketched entities:

1 **Ctrl**-select the entities and click [icon] on the **Sketch Relations** toolbar or click **Tools, Relations, Add**.

2 Select from the relations in the dialog box. Available choices are based on the entities types you picked.

3 Click **OK**.

For more information about adding and deleting geometric relations, see **Geometric Relations** on page 2-28 and **Display/Delete Relations** on page 2-32.

Empty Drawing View

You can create an empty drawing view to contain your sketch geometry. When this view is activated, all sketch geometry added belongs to this view. The sketch geometry can then be scaled, moved, and deleted as a group while still retaining the editability of the individual sketch entity.

To create an empty drawing view:

Click **Insert, Drawing View, Empty**.

Converting Sketched Entities

To convert sketched curves to construction geometry:

   1 Select a sketched line, arc, or circle in a drawing.

   2 Click **Tools, Sketch Entity, Construction Geometry**.

– or –

   1 Right-click a sketched line, arc, or circle on a drawing.

   2 Select **Construction Geometry** from the right-mouse menu.

9

Detailing

You can add much of the necessary detailing of your models in the part and assembly documents. This includes dimensions, notes, symbols, etc. Then you can import the dimensions and annotations from the model into a drawing.

Once in the drawing, you can add other annotations, reference dimensions, and a bill of materials if required.

This chapter discusses the following detailing topics:

❑ Selecting detailing options

❑ Dimensioning in drawings

❑ Annotating parts, assemblies, and drawings

❑ Adding a bill of materials to an assembly drawing

Detailing Options

You have many options for customizing your drawings to meet your company's detailing standards. To select default drawing and detailing options, click **Tools, Options,** then review the options available on these tabs:

❑ **Drawings**. Specify the scale, and various display and update options for views. See **Drawings Options** on page A-8.

❑ **Crosshatch**. Specify the crosshatch patterns used for section views, and for user-defined hatched areas. See **Crosshatch Options** on page A-4.

❑ **Line Font**. Specify the line weights and styles used for various edge types in the drawing. See **Line Font Options** on page A-20.

❑ **Detailing**. Specify numerous options related to dimensions and other types of annotations. See **Detailing Options** on page A-4.

Annotating Drawing Documents

You can add many types of annotations to SolidWorks *part* and *assembly* documents. Additionally, for *drawing* documents, you can:

❑ **Import the annotations, dimensions, and reference geometry from the assembly and part documents into the drawing.**

Changes that you make to imported annotations and dimensions in the part or assembly document are updated in the drawing automatically. Changes you make to imported annotations and driving dimensions in the drawing are updated in the part or assembly document automatically.

> **NOTE:** One-way associativity can be set between drawings and models during the software installation. This permits changes to the drawing that do not change the model. This option can only be reset with a new installation of the software.

The imported items are displayed in the **Imported Annotations** color defined in **Tools, Options, Color** tab. You can drag the attachment point along the edge or face where it is attached, but you cannot re-attach an annotation to a different edge/face/vertex or free space. The attachment of the annotation in the drawing does not affect the attachment in the part or assembly.

For more information about importing reference geometry, see **Reference Geometry in a Drawing** on page 9-15.

❑ **Add annotations and reference dimensions in the drawing.**

Annotations and reference dimensions added in a drawing do not affect the part or assembly document. Reference dimensions and annotations are displayed in the **Reference Annotations** color defined in the **Tools, Options, Color** tab.

For more information about creating reference dimensions in a drawing, see **Adding Reference Dimensions** on page 9-5. For more information about creating annotations in any type of SolidWorks document, see **Creating Annotations** on page 9-17.

Dimensions in a Drawing

Dimensions in a SolidWorks drawing are associated with the model, and changes in the model are reflected in the drawing.

Typically, you create dimensions as you create each part feature, then insert those dimensions into the various drawing views. Changing a dimension in the model updates the drawing, and changing an imported dimension in a drawing changes the model.

You may also add dimensions in the drawing document, but these are *reference* dimensions, and are *driven*; you cannot edit the value of reference dimensions to change the model. However, the values of reference dimensions change when the model dimensions change.

By default, imported dimensions are black. This includes dimensions that are blue in the part or assembly document (such as the extrusion depth). Reference dimensions are gray and appear with parentheses. To display reference dimensions without parentheses, *deselect* **Add Parentheses by Default** on the **Detailing** tab of the **Options** dialog box, or right-click an individual reference dimension and *deselect* **Show Parentheses**.
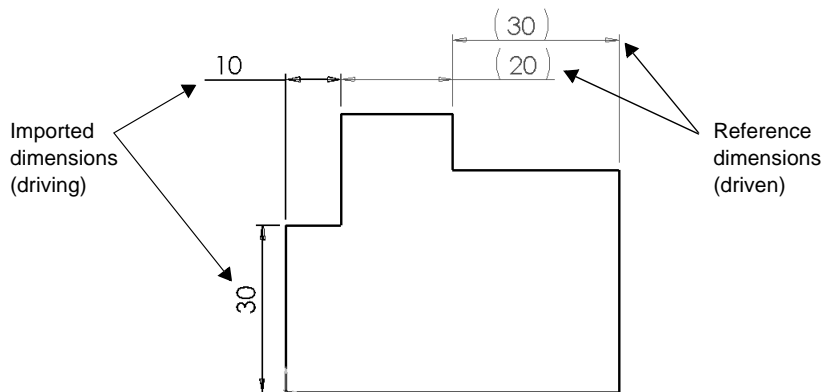
Importing Model Dimensions

You can import the dimensions from the model into all the drawing views at once, or into selected drawing views.

❑ When you import dimensions *to all views at once*, dimensions appear in the most appropriate view. For example, the diameter of a hole is displayed in its **Normal To** view. Features that appear in partial views, such as **Detail** or **Section** views, are dimensioned in those views first. Inserting dimensions this way reduces the effort required to clean up the drawing.

❑ When you import dimensions *to selected views*, you can import the dimensions for the entire model, or you can selectively import the dimensions for one or more components (in an assembly drawing) or features (in a part or assembly drawing).

Dimensions are placed only in the views where they are appropriate. Duplicate dimensions are not imported automatically. Once a dimension has been imported to one view, it is not imported again to another view.

You may delete a dimension from one view, then import it to a different view, or you may move or copy it to another view (see **Moving and Copying Dimensions** on page 9-7).

To insert the model dimensions in a drawing:

1   Select the view(s) to which you want to import dimensions. If you want to import dimensions to all the views at once, do not pre-select a view.

2   Click **Insert, Model Items.**

The **Insert Model Items** dialog box appears.

3   Select **Dimensions**.

To display the instance count in a pattern or the revolution count in a helix, select **Instance/ Revolution Counts.**

4   Change the selection of views if necessary. To import dimensions to selected views, deselect **Import Dimensions into All Views**, then click the desired views on the drawing sheet.

The selected views are listed in the **Import into Views** box. Click the views again to remove them from the list; right-mouse click the drawing sheet, and select **Clear selections** to remove all the views from the list.

5   Click **OK**.
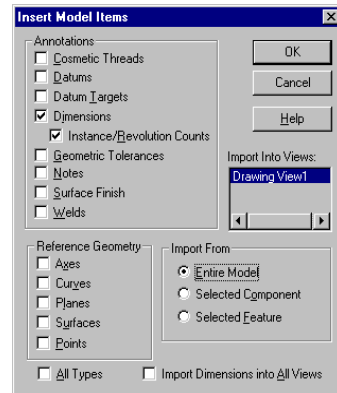
To insert model dimensions for a selected component or feature in a drawing:

1   In a part drawing, click a face of the feature; in an assembly drawing, click a component or the face of a feature (or expand the **Drawing View** in the FeatureManager design tree, and select the component or feature).

2 Click **Insert, Model Items,** and select **Dimensions** (and **Instance/Revolution Counts** if needed).

3 Under **Import From:**

- Click **Selected Feature** to add the dimensions for only the selected feature.

- Click **Selected Component** to add the dimensions for the selected component (available when an assembly component, or a feature of an assembly component, is selected).

- Click **Entire Model** to add the dimensions for the entire part or all the assembly components.

NOTE: When **Selected Feature** or **Selected Component** is checked, dimensions for the item are displayed only in the view where you selected the item. You cannot select a different view or all views.

---

4 Click **OK**.

Adding Reference Dimensions

You can use the same methods to add parallel, horizontal, and vertical reference dimensions to a drawing as you use to dimension sketches. See **Dimensioning Sketches** on page 2-22 for more information. Additionally, you can create *ordinate* or *baseline* dimensions in a drawing.

NOTE: Reference dimensions are not driving dimensions. You cannot edit reference dimensions to change the model. However, when you change the model, the reference dimensions update accordingly.

To add parallel dimensions to a drawing:

1 Click 🖊 on the Sketch Relations toolbar, or click **Tools, Dimensions, Parallel**.

2 Click the geometry to dimension. As you move the cursor around the model, a preview of the dimension type is displayed. The cursor position relative to the attachment points causes the dimension to snap to the appropriate type (horizontal, vertical, linear, radial, etc.).

To override the snap behavior, before selecting the points to dimension, right-click and select either **Horizontal Dimension** or **Vertical Dimension** (or click **Tools, Dimensions, Horizontal** or **Vertical**).

3 When the preview indicates the desired dimension type and location, click to place the dimension.
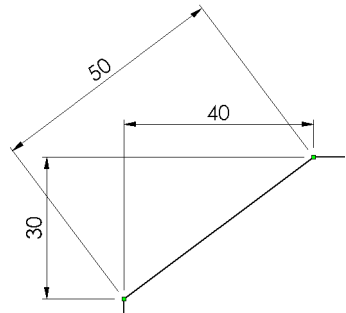
Baseline Dimensions

To add baseline dimensions to a drawing view:

1 Click **Tools**, **Dimension**, **Baseline** (or click
   [image] on the Sketch Relations toolbar, then right-
   click and select **Baseline Dimension**).

2 Click the edge or vertex you want to use as a
   baseline.

3 Click the edges or vertices you want to
   dimension using the same baseline.
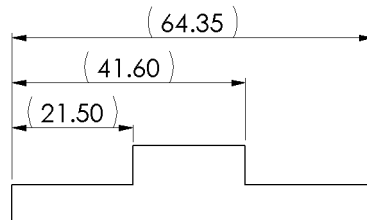
As you click each item, the dimension is placed
in the view.

The dimensions are aligned using the spacing values specified for **Dimension offset
distances** (click **Tools, Options, Detailing, Arrows** to modify the values).

Baseline dimensions are automatically grouped, so that when you drag any member of
the group, all the members move together, maintaining the spacing.

Ordinate Dimensions

Before you add ordinate dimensions, *deselect* the **Single command per pick** option on
the **General** tab of the **Options** dialog box, if necessary.

To insert ordinate dimensions:

1 Click **Tools**, **Dimension**, **Ordinate** or **Horizontal Ordinate** or **Vertical Ordinate** (or
   click [image] on the Sketch Relations toolbar, then right-mouse click, and select **Ordinate
   Dimension, Horizontal Ordinate**, or **Vertical Ordinate**).

2 Click the entity at the zero position, and click
   to place the dimension outside the model.

3 Click the edges or vertices you want to
   dimension using the same ordinate. As you
   click each item, the dimension is placed in the
   view, aligned to the zero ordinate.
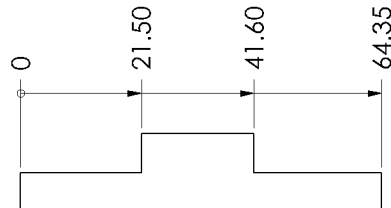
Ordinate dimensions are automatically
grouped, so that when you drag any member
of the group, all the members move together,
maintaining the alignment.

To disconnect a dimension from the alignment group, right-click the dimension, and
select **Break Alignment**.

To remove the arrows:

Right-mouse click any of the ordinates, and select **Properties**. Deselect **Display as chain dimension**.

To add additional ordinate dimensions:

1 Right-mouse click any of the ordinate dimensions, and select **Add to Ordinate**.

2 Select an entity to dimension, and click to place the dimension. The added dimension is aligned and grouped to the others in the same ordinate group.

To line up ordinate dimensions:

Right-mouse click any of the ordinate dimensions, and select **Align Ordinate** from the menu. Any ordinates that have been disconnected from the group, and moved, are re-aligned to the zero dimension. However, they are *not* returned to the alignment group. See **Aligning Dimensions** on page 9-8 for more information.

To jog ordinate dimensions to reduce crowding:

1 Right-mouse click an ordinate dimension to move and select **Jog**. A check mark appears next to the menu item.

2 Pull the dimension in any direction. The leader bends so that you can place the dimension freely.

3 To return the dimension to its original place, right-click the dimension, and deselect **Jog**.
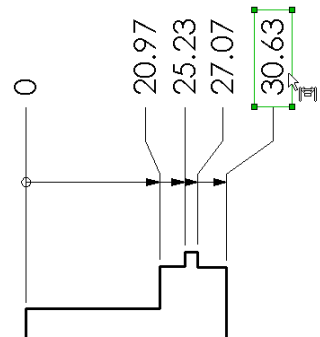
Moving and Copying Dimensions

Once dimensions are displayed, you can move them within a view or to another view. When you drag a dimension from one place to another, the dimension reattaches to the model, as appropriate. You can only move or copy dimensions to a view where the orientation is appropriate for that dimension.

• To *move* a dimension *within the view*, drag the dimension to the new position and release the mouse button.

• To *move* a dimension *from one view to another view*, press the **Shift** key as you drag the dimension to another view, then release the mouse button.

• To *copy* a dimension from one view to another view, press the **Ctrl** key as you drag the dimension to another view, then release the mouse button.

• To move or copy several dimensions at once, hold the **Ctrl** key as you select.
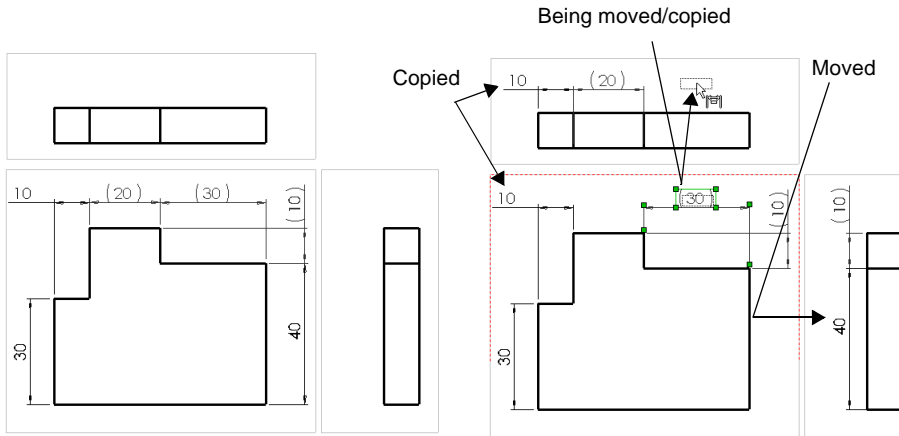
Aligning Dimensions

To distribute parallel dimensions:

1 Select a group of dimensions, either by dragging a rectangle around them, or by holding the **Ctrl** key while selecting them.

2 Click **Tools, Dimensions, Align Parallel,** or ⊞ on the Drawing toolbar.

The dimensions are distributed with the parallel arrows spaced uniformly. To specify the spacing values, click **Tools, Options**, select the **Detailing** tab, and click the **Arrows** button. Under **Dimension offset distances**, enter values for **From model** (used only for baseline dimensions) and **From last dimension**.

The dimensions are grouped so that they move together when you move any one of them, and the spacing is maintained.
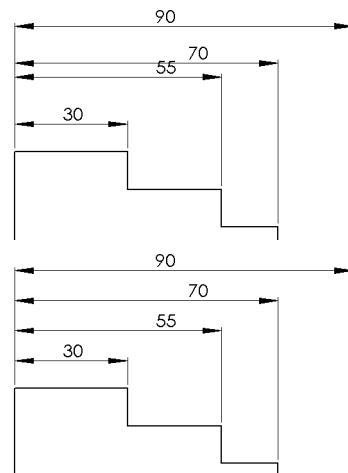
To align dimensions collinear:

1 Select a group of dimensions, either by dragging a rectangle around them, or by holding the **Ctrl** key while selecting them.

2 Click **Tools, Dimensions, Align Collinear,** or ![icon] on the Drawing toolbar.

The dimensions are aligned with the arrows collinear, in line with the last dimension chosen.

The dimensions are grouped so that they move together when you move any one of them.
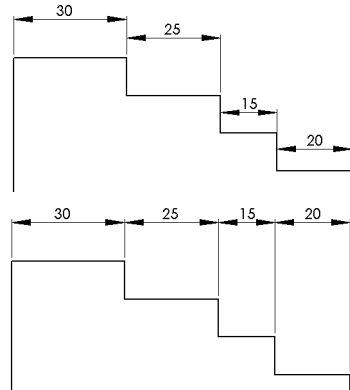
To ungroup dimensions that are aligned (baseline, ordinate, parallel or collinear):

Right-click the dimension to ungroup, and select **Break Alignment**. The selected dimension is disconnected from the group and can move freely; any others that were grouped before remain connected.

To break the alignment of the entire group, select all the dimensions in the group (drag a rectangle around them, or hold the **Ctrl** key while selecting them). Right-click one of them, and select **Break Alignment**.

To see if a dimension is aligned to others:

Right-click the dimension you want to know about. If it is a member of an aligned group, you can select **Show Alignment**. A blue dot appears on each of the other dimensions in the aligned group.

If the selected dimension is not a member of an aligned group, the **Show Alignment** item does not appear on the right-mouse menu.
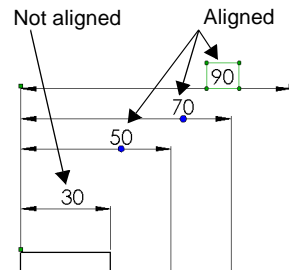
To center dimension text between witness lines:

Right-click a dimension (linear, radial, or angular), and select **Center Text**.

If the text is inside the witness lines (or inside the circle, in the case of a radial or diameter dimension), it snaps to the center, and is locked in position between the witness lines. If the text is outside the witness lines, it remains outside. To unlock the text position, right-click the dimension, and *deselect* **Center Text**.

You can also drag the text toward the center of the witness lines to snap it to the center. When you center dimension text this way, its position is not locked between the witness lines, so you can move it freely.

To offset dimension text from its arrows:

Right-click a dimension (linear, linear radial, or angular), and select **Offset text**.

A leader is attached to the dimension text, and you can drag the text to a new location. When you select an offset dimension, a handle is displayed on the arrows so you can move them away from the model if needed.

To restore the dimension to its position between the arrows, right-click the dimension again, and *deselect* **Offset text**.
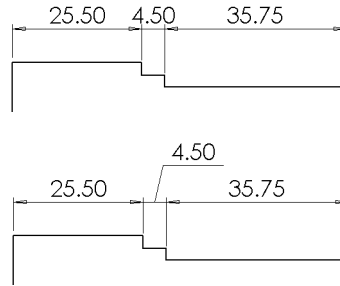
Modifying Witness Lines

You can change the default attachment point of dimension witness lines, and you can slant the witness lines if needed.

To change the model attachment point of dimension witness lines:

1   Click a dimension. Handles are displayed on the witness and extension lines.

2   Drag the handle on the *witness* line (at the attachment point) to the desired position or to a vertex (watch for the inferencing cursor). If you choose a vertex, the default witness line gap is used.
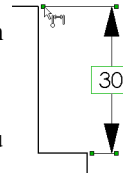
To return the witness line to its original position, right-click the dimension and select **Default Witness Snap**.

To slant witness lines:

1   Click a dimension. Handles are displayed on the witness and extension lines.

2   Drag the handle on the *extension* line to slant the witness lines as needed.

To return the dimension to its original position, right-click the dimension and select **Remove slant**.
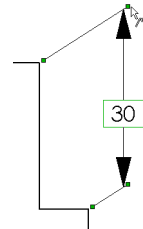
To flip the direction of a broken leader:

If a vertical dimension with a broken leader is outside the witness lines, you can flip the direction of the leader.

Click the dimension, then click the handle at the corner of the broken leader.
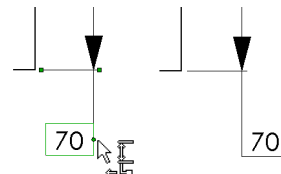
Modifying Dimension Values

To modify a dimension value in a drawing:

1 Double click the text of the dimension. The dimension value is displayed in the **Modify** box.

2 Use the spinbox arrows to change the value, or enter a new value and press **Enter**.

3 Click ✓ to accept the current value and exit.

Click ✗ to restore the original value and exit.

Click ±? to reset the spin box increment.
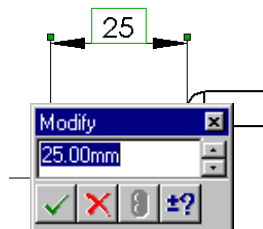
> **NOTE:** If the **Input dimension value** option is checked (under **Tools, Options, General**), the **Modify** dialog box is displayed automatically when you change the attachment of a dimension.

Dimension Properties

You set the default properties for dimensions on the **Detailing** tab of the **Options** dialog box. See **Detailing Options** on page A-4 for more information.

You can modify the properties of a selected dimension at any time during the detailing process.

To modify the properties of a dimension:

1 Right-click the dimension, and select **Properties**.

To select multiple dimensions, hold the **Ctrl** key as you select, then right-click and select **Properties**. You can only edit the properties that can be applied to all the selected dimensions. For example, you cannot edit the **Name**, because names must be unique.

2 Change values as described below.

3 Click **OK**.

General Properties

The following properties are common to all types of dimensions.

- **Value.** The numeric value of a dimension.

- **Name.** The name given to a dimension and used in relations associated with the dimension.

- **Full Name.** The full name of the dimension, which consists of the **Name**, described above, and the name of the sketch or feature in which it is found (for example, **D2@Sketch1**). The **Full Name** is used in equations and design tables. It is updated when you edit the **Name**.

- **Arrow Style.** Either **Use document's style**, as defined on the **Detailing** tab of the **Options** dialog box, or deselect this checkbox, then choose a different style from the list.

- **Arrows.** Defines placement of arrows with respect to witness lines.

- **Font**. Either **Use document's font**, as defined on the **Detailing** tab of the **Options** dialog box, or deselect this checkbox, then click **Font**, and select a new style or size.

- **Display Precision**. Either **Use document's precision**, as defined on the **Detailing** tab of the **Options** dialog box, or deselect this checkbox, then click **Precision**, and specify the precision of the primary and alternate units of measure.

- **Display.** Use the checkboxes to selectively specify which **Witness Lines** and **Leaders** to display.

- **Modify Text**. Allows you to add prefix and suffix text and symbols. See page 9-13.

- **Tolerance**. Allows you to specify and display dimension tolerances. See page 9-14.

- **Use document's units**. To change the units from those specified for the document, deselect this checkbox, then click the **Units** button. In the **Dimension Units** dialog box, select a different **Length unit** or **Angular unit**, and select other options (**Fractions**, etc.), depending on the type of dimension and units selected.

Selection-specific Properties

Depending on the type of dimension (linear, radius, reference, driving, etc.) you choose, some of these additional properties may apply.

- **Driven.** Specifies that the dimension is driven by other dimensions and conditions.

- **Read Only.** Determines whether feature dimensions can be changed or not.

- **Display as Dual Dimension**. Enables dimension display in two different units.

- **Diameter dimension**. Specifies that arc dimensions are diameter, *not radius* dimensions. To convert a diameter dimension to a radius dimension, deselect this option.

- **First / Second Arc Condition**. Choose **Center, Min,** or **Max** to specify the way to dimension distances between an arc or circle and another entity. See **Dimensioning Arcs and Circles** on page 2-24 for more information.

- **Foreshortened radius**. Specifies that the radius dimension line is foreshortened when the centerpoint of a radius is outside of the drawing or interferes with another drawing view.

- **Display with parentheses**. Specifies whether to display reference dimensions with parentheses.

- **Use document's second arrow**. Specifies that a second arrow is used for diameter dimensions, according to whether or not **Display second outside arrow** is checked under **Tools, Options, Detailing, Arrows**.
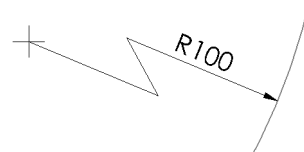
- **Display second outside arrow**. Specifies that a second arrow is used for diameter dimensions, regardless of whether or not the option is checked in **Tools, Options, Detailing, Arrows**.

⌀15

- **Dimension to inside of arc**. Specifies that dimensions to arcs are always to the inside of the arc.

- **Display as chain dimension**. Displays ordinate dimensions with an arrow from the base point to the dimension position.

- **Display with solid leader.** Specifies the display of dimensions with solid leaders.

- **Display as linear dimension.** Allows the display of a radius or diameter dimension as a linear dimension.
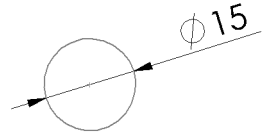
Modify Text

You can add a variety of symbols to the text of a dimension.

To change the text of a dimension:

1 In the **Dimension Properties** dialog box, click **Modify Text.**

2 Place the cursor in the **Dimension Text** box where you want to add a symbol or text. The dimension value is represented by **<DIM>** in the middle box. As you make modifications, the text and symbols are displayed in the **Preview** box.

- Add text or symbols to the left or right of the dimension in the middle box.

- Add text or symbols above or below the dimension in the upper or lower box.

3 To add a symbol:

a) Place the cursor in the text box where you want the symbol to appear and click **Add Symbol**.

b) Select a library from the **Symbol Library** list.

c) Select a symbol name from the list, and click **OK**.

4 To replace the dimension value **<DIM>** if it was deleted accidentally, click **Add Value.**

5 To center the text between the witness lines, click **Center text**. This has the same effect as selecting **Center Text** from the right-mouse menu. See page 9-9.

6 Click **OK**.

Tolerance

To modify the tolerance of a dimension:

1   In the **Dimension Properties** dialog box, click **Tolerance.**

2   Under **Tolerance Display**, select the type of tolerance to display: **None, Basic, Bilateral, Limit, Symmetric, MIN, MAX** or **Fit.**

3   Specify **Maximum / Minimum Variation** values appropriate for the type of tolerance that you selected.

   Linear / Angular Tolerance indicates the type of dimensions to which the current changes apply. This is provided for information only.

4   Specify the **Font**. Either **Use dimension's font**, or deselect the checkbox, then specify the **Font Scale** (a number from 0 to 10.0) or **Font Height**.

Dimension Visibility

You can change the visibility of an individual dimension. Model or reference dimensions may be *hidden* or *shown* to control the appearance of the drawing, temporarily or permanently.

To change the visibility of a dimension:

1   Click **View**, **Hide/Show Dimensions.** The **Show/Hide** cursor appears.

2   Select the dimensions that you wish to hide or show. Select again to toggle between hidden and shown states.

> **NOTE:** Any previously hidden dimensions re-appear in a dim gray color when you choose **Hide/Show Dimensions** again.

Reference Geometry in a Drawing

The reference geometry contained in a model is hidden by default in drawing views. When you import the reference geometry, all entities of the selected types are shown. You can import reference geometry to a selected view, or to all views at once, in the same way that you import dimensions. See **Importing Model Dimensions** on page 9-4 for more information.

To import reference geometry:

1 Click **Insert, Model Items**.

2 In the **Insert Model Items** dialog box, under **Reference Geometry**, select the types to display (**Axes, Curves, Planes, Surfaces, Points**), then click **OK**.

Planes, axes, curves, and points are shown in gray. Surfaces are shown as visible lines (black).

To toggle the display status of an individual entity, right-click the entity in the FeatureManager design tree, and select **Hide** or **Show**.
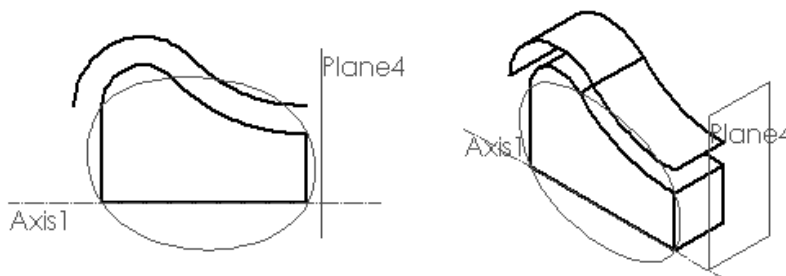
Annotations

Annotations behave like dimensions in each type of SolidWorks document. You can add them in the part or assembly document, then import them to the drawing views, or you can create them in the drawing. See **Annotating Drawing Documents** on page 9-2, and **Moving and Copying Dimensions** on page 9-7 for more information about how dimensions and annotations behave.

Importing Model Annotations

You can import the annotations that belong in a view all at once, or you can selectively import the annotations for a component or feature.

To insert model annotations in a drawing:

1 Select the view(s) you want to annotate. If you want to annotate all the views at once, do not pre-select a view.

2 Click **Insert, Model Items.**

3 In the **Insert Model Items** dialog box, select the annotation types you want to add, or click **All Types** to import all annotations, dimensions, and reference geometry.

4 Click **OK**. The annotations that are appropriate to the selected view are displayed.

To insert model annotations for a selected component or feature in a drawing:

1 In a part drawing, click on a face of the feature, or in an assembly drawing, click on a component or the face of a feature.

Alternatively, you can expand the **Drawing View** in the FeatureManager design tree where you want to add the annotations, and select the feature(s).

2 Click **Insert, Model Items,** and select the annotation types to add.

3 Under **Import From:**

- Click **Selected Feature** to add the annotations for only the selected feature.

- Click **Selected Component** to add the annotations for the selected component (assembly component or feature of an assembly component selected).

- Click **Entire Model** to add the annotations for the entire part or all the assembly components.

> **NOTE:** When **Selected Feature** or **Selected Component** is checked, annotations for the item are displayed only in the view where you selected the item. You cannot select a different view or all views.

4 Click **OK**.

Creating Annotations

You can add *all* types of annotations to a drawing document. You can add *most* types in a part or assembly document, then import them into a drawing document. However, there are some types that you can add *only in a drawing document.* Annotations that are limited to drawing documents are indicated as such in the sections that follow.

To add annotations to any SolidWorks document, click **Insert, Annotations** and select from the available annotation types in the pulldown menu.

You can also click one of the icons in the Annotations toolbar. Only those annotation types that are appropriate for the selected entity are available for selection; the others are displayed in gray.
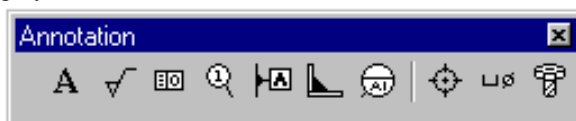
Aligning Annotations

You can control the snap behavior of detail items that you move. Click **Tools, Options**, and select the **Drawing** tab. Under **Detail Item Snapping**, you have these options:

- **Infer when dragging corner**. When checked, you drag the selected item by the *corner handles* to infer to the *corner* of another detail item.

- **Infer when dragging center**. When checked, you drag the *center* of the item to infer to the *center* of another detail item.

- To ignore all inferences, and to prevent the display of inferencing lines, press the **Alt** key while dragging detail items.

To align annotations with neighboring annotations:

1 Click a detail item (note, balloon callout, etc.).

2 Drag the item until inferencing lines appear and the item snaps to align with its neighbor.

3 Release the mouse button.

Inferencing line

Note $\boxed{\text{A}}$
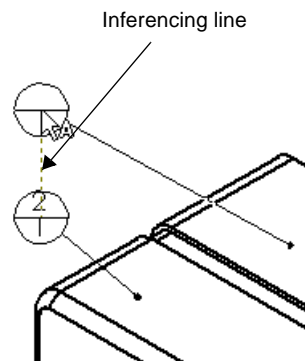
You can create a variety of notes. A note may be free floating or placed with a leader pointing to an item (face, edge, etc.) in the document. It may contain simple text, symbols, parametric text and/or hyperlinks. A note can have a maximum of 1023 characters.

You can set the default note properties (font, arrow and balloon styles, leader display, etc.) by clicking **Tools, Options**, and the **Detailing** tab.

To create a note:

1   Select a note location on the drawing sheet, or the attachment point on the model, if the note is to include an attached leader.

2   Click **Insert**, **Annotations**, **Note** or $\boxed{\text{A}}$ on the Annotations toolbar.

3   Type the text into the **Note Text** dialog box.

To add a *parametric* value to a note, select the corresponding dimension. Parametric text changes when the dimension changes. The dimension *name* is shown in the **Note text** box; the dimension *value* is shown in the note.

4   Modify the **Angle** and **Text Justification** (**Center, Left, Right**) if needed.

5   To specify a link to a file or URL, click **Insert Hyperlink** 🌐.

Dimension name as parametric text.
Resulting note:
2.00 is part height.

6   To link the text of the note to either a system-defined property, or to a user-defined custom property, click **Link to Property** 📇. See the next section for more information.

7   Select the necessary options for **Leader**, **Balloon**, and **Font**.

8   If a modifying symbol is needed, click **Add Symbol**. In the **Symbols** dialog box, click **Use Symbol**, select a library from the **Symbol Libraries** list, then select the symbol, and click **OK**.

9   Click **OK**.

10 If you selected **Display with Leader**, select the note location on the drawing.

You can cut and paste a note along with its leader by selecting the note and using **Ctrl-X** and **Ctrl-V**. You can copy and paste a note using **Ctrl-C** and **Ctrl-V**.

To edit a note:

1   Either double-click the note, or right-mouse click the note and select **Properties** from the menu.

2   Edit the properties as needed, and click **OK**.
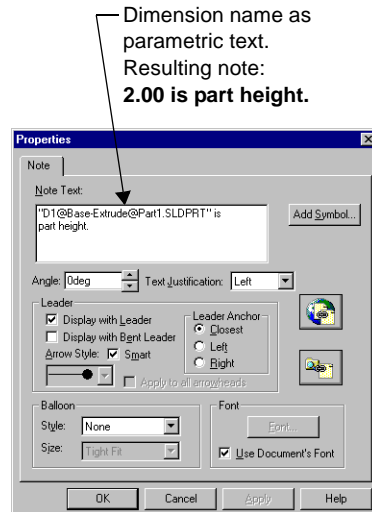
Links to Document Properties

You can link the text of a note to a document property. Properties may be either *system-defined,* or *user-defined* (custom file properties). In either case, if the value of the property changes, the text that is linked to the property changes automatically.

All SolidWorks documents have the system-defined property **SW-File Name** (the name of the document, without the extension). Drawings have these additional system-defined properties: **SW-Sheet Scale, SW-Template Size, SW-Current Sheet, SW-Total Sheets,** and **SW-Sheet Name**.

See **Customizing a Drawing Template** on page 8-5 for information about using these properties to fill in the title block of a drawing automatically.

To link a note to a property:

1   In the **Note** dialog box, click **Link to Property** ⊞.

2   Click the dropdown list under **Property Name**. Any properties that have values assigned are shown in the list, as well as the system-defined properties for the active document.

   To add or change a property, click **Edit Properties**. See **Custom Properties** on page 5-25 for information about how to add properties.

3   Select the property from the list, and click **OK**. The **Note Text** shown in the dialog box is the variable name **$PRP:"propertyname"**. However, in the active document, the note text is the *value* assigned to the property.

4   To link the note to the properties of the model in a drawing view, click **External model reference**, then select a **Property Name** from the list.

   • For a note that belongs to a *view*, the value of the property for the model in that drawing view is displayed. The dropdown list includes the properties of the model document, not the drawing document. (If a view is selected when you insert the note, or if the note is attached to the model in a view, the note belongs to the *view*.)

   • For a note that belongs to the *sheet* or the *template*, the value of the property for the model shown in *the first view* inserted into the sheet is displayed by default.

   If the sheet contains views of more than one model, you can specify which view contains the model whose properties you want to use. Right-mouse click the sheet, select **Properties**, then select the view from the list under **Use custom property values from model shown in.**

   • If no model is shown in the drawing, you may either select a property from the drawing's list, or add a new custom property. The property is resolved when a model is added to the sheet.

5   Click **Rebuild** 🔋 to update the drawing, if necessary.

Hyperlinks

You can add an embedded hyperlink in a note, or you may add a floating hyperlink in any SolidWorks document. The hyperlink may be to a document on the Internet, on your local network, or on your own hard drive.

If the link is to a website, selecting the hyperlink launches a web browser if one is not already running, and opens the associated URL (website). If the link is to a SolidWorks document, clicking the link opens the document. If the link is to another type of document, such as a Word or Excel file, clicking the hyperlink launches the appropriate application, and opens the document.

The actual path of a hyperlink embedded in a note does not appear in the note text, A hyperlinked note is displayed in blue; when the mouse moves over a note that contains a hyperlink, the cursor changes to a hand, and the tooltip indicates the path of the link.

A floating hyperlink (one that is not attached to note text) displays the full path of the link (for example, **HTTP://www.solidworks.com**) in blue.

To create an embedded hyperlink:

1   Create a **Note** as described above, and click **Insert Hyperlink** 🌐 in the **Note** dialog box. You can also add a hyperlink while editing a note's properties.

2   Enter the path (or browse to the document) to which you want to link the note text. For a World Wide Web location, enter the website name in the format **www.*website.com.***
(You do not need to enter **HTTP://**)

3   Click **Use relative path for hyperlink** if you want to the specify a directory path relative to the current location instead of the full pathname.

4   Click **OK** to accept the path.

To create a floating hyperlink:

1   Click **Insert, Hyperlink**.

2   Enter the path (or browse to the document) and click **OK**.

If you edit the properties of a floating hyperlink, you will see that it is a note, the text of which is **$URL** (a variable representing the path of the link).

Because clicking a hyperlinked note activates the link, moving or editing a hyperlinked object can be difficult. Move the cursor over the object until you see the annotation cursor ⌘, then drag the object to move it, or right-click the object to access its properties.

Notes with multiple leaders

You can select multiple attachment locations for a note. You can add, move, and remove leaders of existing notes.

To create a note with multiple leaders:

1 Click the attachment points while holding the **Ctrl** key.

2 Click **Insert**, **Annotations**, **Note**, and proceed as described above.

To add more leaders to an existing note:

1 With *nothing* selected, move the cursor over the attachment point of a note's leader until you see the ⌖ cursor.

2 Press the **Ctrl** key, drag the leader to a new attachment point, and release.

To delete extra leaders:

1 Select the note. The text and the attachment points are highlighted.

2 Click the attachment point on the leader to delete, watching for the ⌖ cursor.

3 Press the **Delete** key.

To modify the arrowheads on multiple leaders:

1 Move the cursor over the attachment point of a leader until you see the ⌖ cursor, then right-mouse click the leader, and select **Properties**.

2 Deselect the **Smart** option if it is selected, then choose a new arrowhead style from the list.

3 If you want all the leaders on this note to use the new arrowhead, click **Apply to all arrowheads**. Otherwise the new arrowhead is applied to only the selected leader.

4 Click **OK**.

Center Mark

(Drawings only)

You can place axis lines for showing center marks on circles or arcs that can be used as reference points for dimensioning. You set the default size of center marks and choose whether or not to display extended axis lines on the **Detailing** tab of the **Options** dialog box.

To create center marks:.

1   Click ⊕ on the Annotations toolbar, or **Insert, Annotations, Center Mark**.

2   Click the circles or arcs.
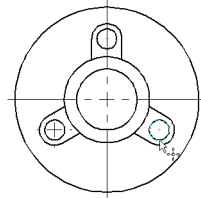
To change the display attributes of individual center marks:

1   Right-mouse click the center mark and select **Properties**.

2   Click to uncheck **Use document's defaults**.

3   Make the changes that you want and click **OK** to accept the changes; click **Cancel** to close the dialog box without saving any changes.

Hole Callout

(Drawings only)

You can add hole callout symbols to a drawing view.

1   Click **Insert, Annotations, Hole Callout** or ⊔ø on the Annotations toolbar.

2   Click the edge of a hole.

The **Modify Text of Dimension** box appears, with symbols appropriate to the selected hole. See **Modify Text** on page 9-13 for more information.

3   Modify as necessary, and click **OK**. The **Hole Callout** tool remains active, and you can continue to add callouts to other holes until you select another tool.
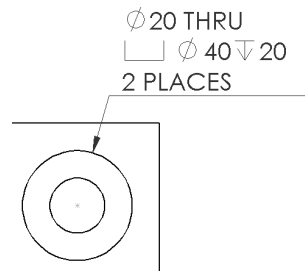
Cosmetic Thread

You can represent threads on a part, assembly, or drawing, and you can attach a thread callout note.

A cosmetic thread differs from other annotations, in that it is an absorbed feature of the item to which it is attached. For example, the cosmetic thread on a hole is in the FeatureManager design tree under the hole feature, along with the sketches used to create the hole.

Cosmetic threads added in a part or assembly can be imported to a drawing view. If you add a cosmetic thread while working in a drawing view, the part or assembly is updated to include the cosmetic thread feature.

You can specify the line style and weight to use for cosmetic threads on the **Line Font** tab of the **Tools, Options** dialog box.

To insert cosmetic threads:

1   On a cylindrical feature, click a circular edge where the thread begins.

2   Click **Insert**, **Annotations, Cosmetic Thread**.

3   In the **Cosmetic Thread** dialog box, select the thread to apply:

   • Select **Blind** to run the thread for a specified distance, and enter the distance in the value box.

   • Select **Up to Next** to run the thread to the next face.

4   Enter a value in the **Minor Diameter/Major Diameter** box.

5   In the **Thread Callout** box, enter thread callout text, if desired.

6   Click **OK**, then click **Rebuild** or on the Standard toolbar to update the part.

M24X2
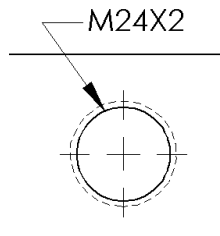
To edit a cosmetic thread:

1   In a part or assembly document, right-mouse click the **Cosmetic Thread** feature, and select **Edit Definition**. You can select it either in the graphics area, or in the FeatureManager design tree (under the feature to which it is attached).

2   Make the necessary changes in the **Cosmetic Thread** dialog box, and click **OK**.

To edit the thread callout on a cosmetic thread:

If you only need to edit the text of the thread callout note, double-click the note (or right-click the note and select **Properties**) in the drawing view, and edit it as needed in the **Properties** sheet for **Note**. See **Note** on page 9-18 for more information.

Surface Finish Symbol

You can specify the surface texture of a part face by using a **Surface Finish Symbol**. You select a model surface, then define a symbol that specifies the values and options for the finish that you want.

To insert a surface finish symbol:

1   Click a face or edge in a part, an assembly, or a drawing view.

2   Click ⎷ on the Annotations toolbar or **Insert, Annotations, Surface Finish Symbol.**

3   Construct the symbol for the selected surface using the options in the **Surface Finish Symbol** dialog box. As you build the symbol, a preview is displayed.

   • **Symbol.** Select a symbol type from **Basic**, **Machining Required**, **Machining Prohibited**, or one of the **JIS** types.

   • **Direction of Lay.** Select the direction of the surface pattern from the list.

   • **Roughness.** Specify the **Maximum**, and **Minimum** height deviation from the mean plane of the surface, and the **Spacing** between peaks and valleys that form the texture of the surface.

   • **Special Requirements.** Define any necessary requirements, such as **Production Method/Treatment**, **Sampling Length**, or **Other Roughness Values**.

   • **Material Removal Allowance.** Specify the amount of stock to be removed by machining.

   • **Leader.** Check **Show** to display the symbol with a leader. Check **Bent Leader** to allow the leader to have a bend. For **Arrow Style,** you can use the **Smart** option, or deselect the option, and choose a different type of leader end from the list.

   • **Rotated.** Check this box to rotate the symbol 90° (only if a leader is shown).

4   Click **OK** when you are satisfied with the surface finish symbol.
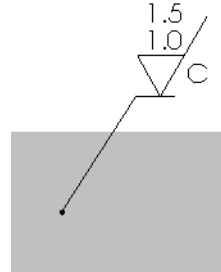
You can drag a surface finish symbol with a leader to any location. If you attach a leaderless symbol to an edge, then drag it off the model edge, an extension line is created.

To edit a surface finish symbol, double-click the symbol (or right-click the symbol and select **Properties**) and make the necessary changes in the **Surface Finish Symbol** dialog box.
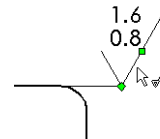
Surface finish symbols can have multiple leaders. Adding them is the same as it is for notes. See **Notes with multiple leaders** on page 9-21.

Datum Feature Symbol $\boxed{\text{H}\!\text{A}}$

You can attach a datum feature symbol to the following items:

- In a part or assembly, on a planar model surface or a reference plane
- In a drawing view, on a surface that appears as an edge (not a silhouette) or a section view surface
- A geometric control frame

To insert a datum feature symbol:

1   Click $\boxed{\text{H}\!\text{A}}$ on the Annotations toolbar, or **Insert, Annotations, Datum Feature Symbol.** The cursor changes to $\overset{\text{k}}{\text{Q}}$.

2   Click the surface on which you want to place a datum feature symbol.

The datum letters are assigned alphabetically (the letters **I, O, Q, S, X,** and **Z** are omitted in ANSI only).
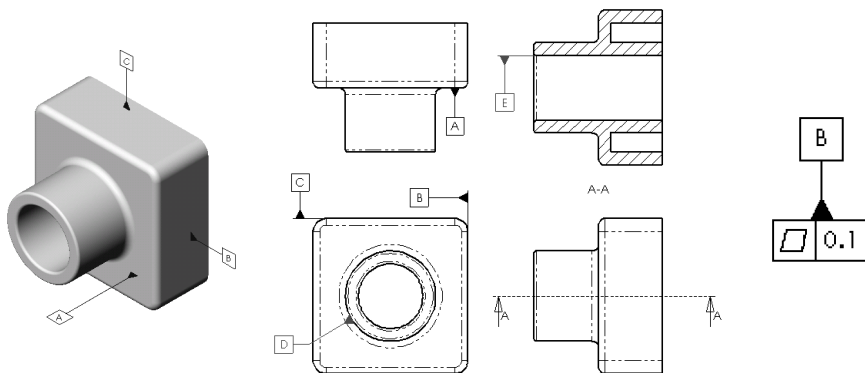
If you drag a datum feature symbol off the model edge, an extension line is added automatically.

To edit the datum feature symbol:

1   Double-click the symbol (or right-click the symbol and select **Properties**).

2   Edit as needed:

- Enter a new letter in the **Label** box to change the label.
- Click **Display with filled triangle** to use a filled triangle at the base; otherwise, an open triangle is used.
- Click **Display with shoulder** to allow a bent leader.

3   Click **OK** to save your changes; click **Cancel** to exit without saving the changes.

Datum Target

You can attach a datum target and symbol to a model face or edge in any document. Both the target (point or area) and the associated symbol are created in one step.

To create a datum target and symbol:

1   Click a model edge or face.

2   Click ⊡ on the Annotations toolbar, or **Insert, Annotations, Datum Target**.

   The **Datum Target Symbol** dialog appears.

3   Select a target shape: **Point** ⊠, **Circle** ⊘, or **Rectangle** ▨.

   If you choose circle or rectangle, enter the target area size in the boxes.

4   Select options as necessary:

   • **Display target area size outside.** Places the size values outside the target symbol balloon.

   • **Display target area.** Displays a crosshatch area of the specified size at the selected location.

   • **Display datum target symbol.** Displays a balloon containing the symbol information.

5   Enter the **Datum Reference** labels in the boxes.

6   Specify the **Leader Information**:

   • Check **Bent Leader**, if appropriate.

   • Check **Smart Arrow** to use the default attachment arrows, or deselect the option, then choose an **Arrowhead Style** from the list.

   • Select a **Line Style** (**Solid** or **Dashed**).

7   Click **OK** to accept the datum target symbol you constructed; or click **Cancel**.

To change the location of the datum target symbol (either the balloon or the crosshatch area or point), click and drag.

To edit the datum target symbol, double-click the symbol, or right-mouse click the symbol and select **Properties** from the menu. The **Datum Target Symbol** dialog appears. Make changes to the options as described above.

In a *drawing*, you can dimension to the center of a datum target area. Click either the symbol or the area to add the dimension.
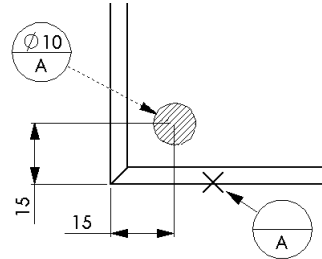
NOTE: The drawing view must be active in order to add dimensions to the symbol.

Geometric Tolerancing Symbol

SolidWorks supports the ANSI Y14.5 Geometric and True Position Tolerancing guidelines. You define the control symbols that are placed in the drawing in the **Geometric Tolerance** dialog box. The **Geometric Tolerance** dialog box displays the feature control frame as you develop it. You can also create a composite geometric tolerance frame.

To create a geometric control frame:

1 Select a model entity or a dimension. If you choose an edge, the leader of the frame will be normal to the selected edge.

2 Click **Insert**, **Annotations**, **Geometric Tolerance** or ⊞ on the Annotations toolbar.

3 In the first **Feature Control Frame** area, click **GCS (Geometric Characteristic Symbol)**. In the **Symbols** dialog box, under **Symbol Libraries**, make sure **Geometric Tolerancing** is selected. Choose a symbol, and click **OK**.

   As you make each selection, a preview of the resulting frame is displayed in the box.

4 Enter a tolerance value in the **Tolerance 1** box. Click ⊘ if you want to include a **Diameter** symbol.

5 Click **MC**, choose the material condition symbol for **Tolerance 1**, and click **OK**.

6 Repeat for **Tolerance 2.**

7 Enter tolerance values and material condition symbols for the **Primary**, **Secondary**, and **Tertiary** datums.

8 If you are creating a composite frame, enter the values for the second **Feature Control Frame**, then click the **Composite Frame** check box.

9 If the tolerance is between two points, click **Between Two Points**, and enter the labels of the points in the boxes.

10 Click the **Show PTZ** check box if you want to enter a **Projected Tolerance Zone** height. Enter a value in the **Height** box.

11 To make changes to the font size or style, arrow style, or leader type, click the **Options** button. Make your changes and click **OK** to close the **Geometric Tolerance Options** box.

12 When you are finished, click **OK** and drag the geometric tolerance frame to the desired position.
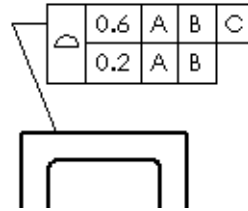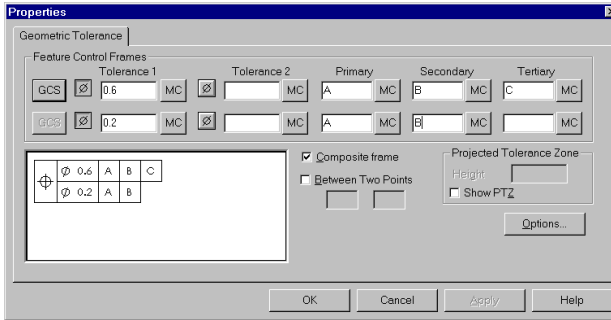
To edit a tolerance control frame:

1 Double-click on the control frame (or right-click the frame, and select **Properties**).

The **Geometric Tolerance** dialog box appears with the current values filled in.

2 Edit as necessary.

3 Click **OK** to update the geometric control frame.

To attach a geometric tolerance frame to a dimension:

You can attach a geometric tolerance frame to a dimension in a drawing view.

1 Create a geometric tolerance frame as described above. You can attach the symbol to a model entity, or create a floating frame, not attached to anything.

2 Select the frame. Note the handles at the corners.

3 Drag the frame by a corner handle until the frame is over the dimension value (watch for the dimension cursor   ), and release the mouse button.

4 Drag inside the frame to reposition it, if needed. You can place the frame either grouped with the dimension text, or outside the witness lines with a leader.
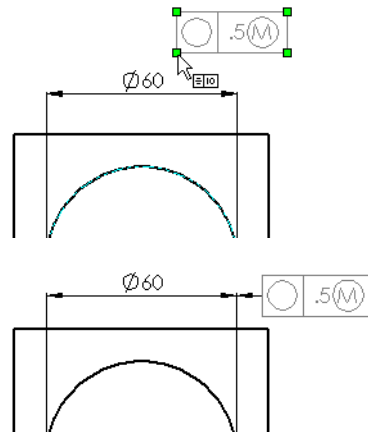
5 To detach the frame, select it and drag a corner handle again. A dimension may have only one attached geometric tolerance frame.

Bill of Materials

(Drawings only)

The **Bill of Materials** menu option lets you insert a bill of materials into the drawing of an assembly.

> **NOTE:** You must have the Microsoft Excel 97 spreadsheet program installed on your computer in order to insert a bill of materials (BOM) into a drawing.

If you select **Automatic update of BOM** in **Tools, Options**, on the **Drawings** tab, assembly changes (adding, deleting, or replacing components) are updated automatically. Otherwise, you may need to click **Rebuild** to update the BOM.

To insert a bill of materials into a drawing:

1 With a drawing view selected, click **Insert, Bill of Materials**.

2 Choose the way to list sub-assemblies and their components in the bill of materials (for assigning item numbers):

- **Show parts only**. Sub-assemblies are not listed as items; sub-assembly components are listed as individual items.

ITEM NO.	QTY.	PART NO.
1	1	box
2	3	handle
3	2	divider1
4	1	divider2

- **Show top level subassemblies and parts only.** Sub-assemblies are listed as items; sub-assembly components are not shown.

ITEM NO.	QTY.	PART NO.
1	1	box
2	3	handle
3	1	divider_assembly

- **Show assemblies and parts in an indented list.** Sub-assemblies are listed as items; sub-assembly components are listed below the sub-assembly item to which they belong, but are not given an item number.

ITEM NO.	QTY.	PART NO.
1	1	box
2	3	handle
3	1	divider_assembly
	2	divider1
	1	divider2

3 Choose from these options:

- **Use the document's note font when creating the table**. When checked, the BOM uses the text font specified for notes, under **Tools, Options, Detailing, Note Font**. Otherwise, the font specified in the bill of materials template file is used. The template is found in *install_directory***lang\English\Bomtemp.xls**.

- **Use Summary Info title as part number**. If you assigned a part identifier number in the title box of the **Summary Info** for the part (under **File, Properties**), use that identifier in the **Part No.** column of the bill of materials. Otherwise, the part name is used.

- **Add new item by extending top border of table.** Choose this option if you want the BOM to shift upward when new items are added, leaving the bottom row of the table in place. Otherwise, the BOM expands downward when new items are added. This option applies only when you do not use an anchor point.

- **Use table anchor point.** The standard templates supplied with the software include an anchor point for placing the BOM in a specific location on the sheet. To use the point, select this option, then select which corner of the BOM table to make coincident with the point (top right, bottom left, etc.).

  If you do not use the anchor point, the BOM is placed on the sheet near the selected view.

  For information about setting the location of the anchor point in the template, see **Customizing a Drawing Template** on page 8-5.

4   Click **OK**.

To move the bill of materials:

If the BOM is attached to an anchor point, you can change which corner of the table is coincident with the anchor point. Right-mouse click the BOM, select **Anchor**, then select a different corner from the list.

You can also detach the BOM from the anchor point, and move it anywhere on the sheet. Right-mouse click the BOM, and select **Anchor, Unlock from Anchor**. Then drag the BOM to a new location.

If you edit the template and move the anchor point, the BOM that is locked to the anchor point moves accordingly.

To attach an unlocked BOM to the anchor point, right-mouse click the BOM, and select **Anchor, Lock to Anchor**.

To edit the bill of materials:

1   Double-click the bill of materials. While the bill of materials is active, it is displayed with shaded borders, and row and column headers. Excel toolbars replace the SolidWorks toolbars.

2   Edit the worksheet as needed, using Excel functions. You can perform these kinds of edits:

- Edit the text in the **Description** column. If you edit text in **Item No.**, **Qty.**, or **Part No.** columns, these changes will be lost the next time that the BOM updates for any reason.

- Add columns or rows, before the cell marked **$$END** (but not above the header row). You can include rows for bulk items, such as oil or adhesive.

- Delete columns or rows (except you may NOT delete the **Part No.** column, or the header row).

- Change the text alignment (left, right, center) by column.

- Adjust the column width or row height.

- Change the text font. You can use one font for the body of the table, and a different font for the header row.

- Move the header row from the top to the bottom of the table. Insert a new row at the bottom (just before the row marked **$$END**), select and cut the header row, paste it into the new row, then delete the empty row at the top.

- Some editing operations apply to an entire row or column in the table. To select an entire row, click the numbered cell to the left of the row. To select an entire column, click the lettered cell above the column.

3 Click outside the worksheet to close it and return to editing the drawing sheet.

To show part or sub-assembly configurations in the bill of materials:

If you want the bill of materials to use a name other than the document name in the part number column (for example, if you want to use the configuration name for a part or sub-assembly that has more than one configuration), you need to edit the **Advanced** properties of the configuration.

1 In the part or sub-assembly document, click the Configuration tab at the lower left of the window to change to the Configuration Manager view.

2 Right-click the configuration you want to use, and select **Properties**.

3 Click **Advanced**. Under **Part No. displayed when used in a Bill of Materials**, select one of the following:

- **Use the document name.** Specifies that the part or assembly name is used.

- **Use name below.** When selected, a text box opens, containing a name to be used in the bill of materials. The configuration name is filled in by default. Alternatively, you may enter a name (such as a part number) of your own choosing.

4 If you *always* want a sub-assembly to be listed as a single component in the bill of materials, click **Don't show child components in BOM when used as sub-assembly.**
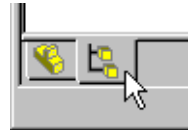
When unchecked, the child components are listed individually in the BOM, when **Show parts only** or **Show assemblies and parts in an indented list** is selected (see page 9-29).

5 Click **OK**.

To save a bill of materials as an Excel file:

1 Click the bill of materials on the drawing sheet.

2 Click **File, Save As**. The **Save as type** is set to **Excel Files (*.xls)** by default.

3 Enter a file name, and click **Save**.

You can use the saved bill of materials in Excel, or as an OLE object in other applications. The saved file is *not linked* to the bill of materials in the drawing; edits to one do not affect the other.

Custom properties in a bill of materials

You can use *custom properties* to add more information to a part or sub-assembly document. Then you can add columns for these properties to the bill of materials template found in *install_directory***lang\English\Bomtemp.xls**.

To add custom columns to the bill of materials:

1 Add custom properties to the part or sub-assembly files, or to the configurations within the file. See **Custom Properties** on page 5-25 for information about how to add custom properties.

2 Open the file *install_directory***lang\English\Bomtemp.xls** (in Excel).

3 Insert a column in the spreadsheet where you want the custom property to be listed.

4 In the top cell in the new column, enter the text you want to display in the column header cell, and press **Enter**. The name appears in the **Formula Bar**.

5 Click the cell to select it, then click **Insert, Name, Define**. Enter the name of the custom property (no spaces in the name), and click **OK**. The name appears in the **Name Box**.

The name in the **Name Box** must *exactly* match the name of the custom property in the part or sub-assembly documents. The text in the **Formula Bar** does not have to match the property name.
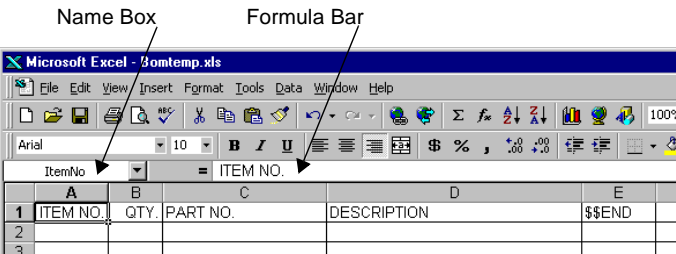
NOTE: DO NOT edit the name in the **Name Box** for these columns: ITEM NO. (**ItemNo**), QTY. (**quantity**), PART NO. (**partno**).

You may edit the text that appears in the header cells for these columns (in the **Formula Bar**).

6 Click **Add**, then click **OK**. A bill of materials will now include the added column, with the value specified in the custom properties of the part or sub-assembly document.

This example uses the custom property **Description**, part numbers were entered in the **Title** box of the file's **Summary Info** (**File, Properties**), and **Use Summary Info title as part number** was checked.

ITEM NO.	QTY.	PART NO.	DESCRIPTION
1	1	240-5892	bracket
2	1	N/A	crank assembly
3	1	001-5432	yoke, male
4	1	001-5431	yoke, female
5	1	003-7790	spider
6	1	120-1121	pin, long
7	2	120-1104	pin, short

Balloon Callout ⊕

You can create balloon callouts in a drawing document. The balloons label the parts in the assembly and relate them to item numbers on the bill of materials. However, you do not have to insert a BOM in order to add callouts.

To set the default BOM balloon properties (size, style, text), click **Tools, Options, Detailing, Balloons**, and select the desired options under **BOM Balloons**. If the default BOM balloon **Type** is **Circular Split Line**, you can choose what to display in the **Upper** and **Lower** portions of the balloon: **Item Number**, **Quantity**, or **Custom.**

Balloons are automatically suppressed when the components they reference are suppressed.

Balloons may also be added in assembly documents. To import balloons from an assembly document into a drawing view, select **Notes** in the **Insert Models Items** dialog box.

To insert balloon callouts:

1 Click ⊕ on the Annotations toolbar, or **Insert, Annotations**, **Balloon**.

2 Click a component in a drawing view of an assembly, or click a component in an assembly model.

   A balloon containing an item number attaches to the part. The number corresponds to the item number in the bill of materials.

3 To move the balloon or leader arrow, select and drag the balloon, or drag the leader by the handle.
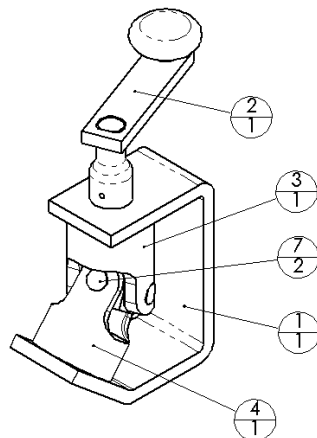
To change the balloon properties:

1 Double-click on the balloon, or right-click the balloon and select **Properties**.

2 Make the changes that you want in the **Note** dialog and click **OK**.

Weld Symbol

When you create a weld bead in an assembly, a weld symbol is created automatically on the **WeldBead** component, representing the parameters you specified. You can edit this symbol to add more information to it. You can also construct weld symbols independently in any SolidWorks document.

For information about creating weld beads, see Chapter 12, "Welding."

To insert a weld symbol:

1   Click a face or edge where you want to indicate a welded joint.

2   Click **Insert**, **Annotations**, **Weld Symbol** or click ⬚ on the Annotations toolbar. The **Properties** sheet for **Weld Symbol** appears.

3   In the **Dimension** box, enter a value for the weld dimension.

4   Click the **Weld Symbol** button and select a weld type from the list in the **Symbols** dialog box.

5   Enter additional dimensions or values as needed for this symbol in the box to the right of the **Weld Symbol** button.

In this example, the length and the pitch of the weld is entered in the box.

6   Continue constructing the weld symbol by choosing options from the dialog box. A preview of the symbol is displayed as you add more information.

The options available depend on the dimensioning standard you use (ISO, ANSI, etc.).
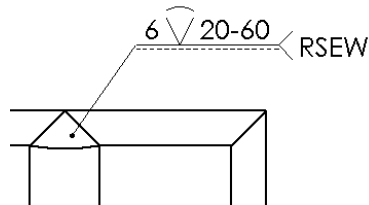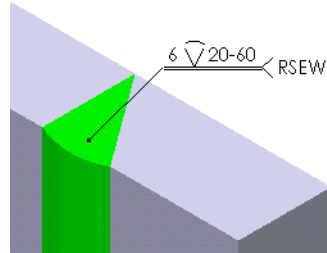
To edit a weld symbol:

Right-click the weld symbol, and select **Properties**. Edit as needed, and click **OK**.

Weld symbols can have multiple leaders. Adding them is the same as it is for notes. See **Notes with multiple leaders** on page 9-21.

Area Hatch

(Drawings only)

You can apply a crosshatch pattern to a selected face, or to a closed sketch loop.

To add crosshatch to a model face:

1 Activate a drawing view.

2 Select a model face.

3 Click **Insert, Area Hatch**.

The selected face is filled, using the default crosshatch pattern specified under **Tools, Options, Crosshatch**. See **Crosshatch Options** on page A-4 for more information.

Model face   Sketched area

To delete the crosshatch pattern from a model face, select the crosshatch, and press the **Delete** key.
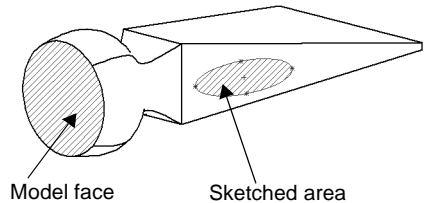
To add a crosshatch area to a sketched area:

1 Sketch a closed profile.

2 Select one segment of the profile.

3 Click **Insert, Area Hatch**.

You can change the shape and size of the sketched profile as needed. The crosshatch updates to fill the modified shape.

To delete the crosshatch pattern, select the crosshatch, and press the **Delete** key. The sketched profile remains.

To edit the crosshatch pattern:

1 Select the crosshatch area, and click **Edit, Properties,**

– or –

Right-mouse click the area and select **Properties**.

2 Choose a different **Pattern** and adjust the **Scale** and **Angle** if needed.

3 Click **OK**.

Custom Symbol

(Drawings only)

You can create, save, and insert custom symbols for drawing items that you use often, such as standard notes, label positions, etc. Custom symbols can include text and any type of sketch entity except points.

If you store your symbols in a specific folder, you can specify that folder as the default search path for adding custom symbols.

To specify a default path for locating custom symbols:

1 Click **Tools, Options, External References.**

2 Under **Show folders for**, select **Custom symbols** from the list, then click **Add**.

3 In the **Choose Directory** dialog box, locate the desired directory, and click **OK**.

   The list can include more than one directory; the *first* path in the list is displayed by default in the **Open** dialog box when you insert a custom symbol.

To create a custom symbol:

1 In a drawing, click **Tools, Custom Symbol, New**. A symbol editing window opens.

2 Sketch the entities that make up the symbol. To add text, use the **Note** tool ⏹A (see **Note** on page 9-18). You can also insert an existing custom symbol in the symbol you are creating.

3 To specify the location for attaching a leader to the symbol, right-click a blank area, and select **Make Custom Symbol**. The entities are grouped, and a leader attachment point is placed at the lower left corner of an imaginary rectangle that includes all the entities.

   If you want the leader to be attached in a different place, click the symbol to display the rectangle and a green handle at the default location. Hold the **Ctrl** key, and drag the handle to a new location.

4 To save the symbol, click **File, Save**. In the **Save As** dialog box, navigate to the directory where you want to store the symbol, and enter a filename (the extension **.sldsym** is added automatically).

To insert a custom symbol:

1 In a drawing, click the sheet, view, or model where you want to add the symbol, then click **Insert, Annotations, Custom Symbol**.

2 In the **Open** dialog box, double-click the filename (***.sldsym**), or click the filename and click **Open**. The symbol is copied to the sheet, with the leader attachment point where you clicked.

   If you pre-selected a model, the symbol includes a leader automatically. If you did not pre-select a model, you can add a leader.

3 Double-click the symbol, or right-mouse click the symbol and select **Properties**.

4 Click **Display with Leader**, and **Display with Bent Leader** if desired, and choose a different **Arrow** style, if desired.

5 Click **OK**, then drag the symbol to extend the leader.

To edit a custom symbol:

1 If a custom symbol includes text, you can edit the text after the symbol is added to the drawing. Double-click the text, type the new text, and press **Enter**.

2 To edit the sketch entities of a symbol, click **Tools, Custom Symbol, Edit**.

3 In the **Open** dialog box, navigate to the directory that contains the symbol, and either double-click the filename (***.sldsym**), or click the filename and click **Open**.

4 Right-mouse click the symbol in the symbol editing window, and select **Explode Custom Symbol**. The symbol items are un-grouped, and you can edit the sketch entities or text as needed.

5 When you are finished, click **Save** to save the symbol with the same name, or **Save as** to save the symbol with a new name. All the sketch entities in the window are grouped to create the symbol.

> **NOTE:** Changes that you make to the symbol text while working in the drawing are not applied to the original symbol file. Likewise, changes you make in the symbol editing window are *not* reflected in any existing instances of the symbol in drawings. Symbols are *embedded* in drawings; the files are *not linked*.

Library Features and the Feature Palette Window

A *library feature* is a feature, or combination of features, that you create once and then save in a library for frequent use.

You can create commonly used features, such as holes or slots, in common sizes, and save them as library features to use whenever you need them. You can use several library features as building blocks to construct a single part. This can save time, and it can help ensure consistency in your models.

The Feature Palette™ window helps you organize and use library features, sheet metal forming tools, and commonly used parts. The Feature Palette window gives you quick access to the features, forming tools, and parts you use most, by simply dragging and dropping them into the SolidWorks window.

This chapter describes:

❑ Creating and editing library features

❑ Adding library features to parts

❑ Organizing features, forming tools, and parts in the Feature Palette window

❑ Inserting palette items in parts and assemblies

Library Features

A library feature can contain one or more extruded bosses or cuts, ribs, fillets, drafts, holes, domes, cosmetic threads, reference planes or axes, or sketches.

You can edit a library feature once it has been placed in a part. After a library feature is added to a part, there is no link between the target part and the original library feature part; if you edit one, the other does not change.

You cannot insert a library feature that includes a base feature into a part that already has a base feature (you cannot have two base features in a single part). Library features usually consist of features built on a base feature, but not the base feature itself.

You can only add a library feature to a part, or while editing a part in the context of an assembly. You cannot add a library feature to an assembly itself.

Creating a Library Feature

To create a library feature, you first create a base feature, then create the features that you want as part of the library feature on the base. You save library features with the **.sldlfp** extension.

To create a library feature:

1   Open a new part, sketch a profile, and create a base.

2   Create the feature(s) you want to have in the library feature.

Dimension the features to the base if you want to use the dimensions to locate the library feature when you place it on the target part.

Library feature with three features: a boss, a hole, and a fillet.

3   In the FeatureManager design tree, select the feature(s) you want to save as the library feature.

To select more than one feature, hold the **Ctrl** key while you select. To select several features in a row, click the first feature, hold the **Shift** key, then click the last feature.

Base

Do not select the **Base** feature unless you plan to insert the library feature into new, empty parts only.

4   Click **File, Save As**. The **Save As** dialog box appears.

5   From the **Save As Type** list, select **Lib Feat Part Files** (***.sldlfp**). Enter a name in the **File name** box and click **Save**.

The part icon beside the part name in the FeatureManager design tree changes to the library feature icon, and each feature that is included is marked with a letter **L**.

NOTE: To create a library feature from an existing part, open the part, select those features that you want to use in your library feature, and save them as a library feature part (***.sldlfp**).

The source part document may contain features that are not related in any way to the features that you selected. The software recognizes such cases, and gives you a choice when you save the library feature part. You can create either a simplified library feature part that does not include the unrelated features, or one that contains all the features. It is important to note that removing the unrelated features may cause rebuild errors in the source part, due to unresolved dependencies.

To edit a library feature:

You can change which features are included as part of a library feature in the library feature part document (**.sldlfp**).

To add another feature, right-click the feature to add, and select **Add To Library**.

To remove a feature, right-click the feature, and select **Remove From Library**.

Adding a Library Feature to a Part

There are two ways to add a library feature to a part:

- Place the library feature in the Feature Palette (available for most, but not all types of features), then drag and drop the feature into the part. See **The Feature Palette Window** on page 10-6.

- Use **Insert, Library Feature** (for those types of features that are not supported in the Feature Palette).

To add a library feature to a part:

1. With the target part open, click **Insert, Library Feature**. The **Insert Library Feature** dialog box appears.

2. Navigate to the directory where the library feature is located, and select the library feature file (**.sldlfp**). The library feature's image appears in the **Preview** area of the dialog box if the **Preview** option is checked.

3. Click **Open.** Two windows and a dialog box appear, and tile automatically:

   - the library feature window
   - the target part window
   - the **Insert Library Feature** dialog box

   In the **Insert Library Feature** dialog, there is at least one mandatory **Reference,** and there may be optional references also. A mandatory reference is preceded by an exclamation point; an optional reference is preceded by a question mark. Dimensional references are optional references.

4   To locate the library feature on the target part, click the **Reference** entity (**Plane**, **Edge**, etc.) that is listed as **Mandatory**. When you click the entity on the target part, the exclamation point in the **Reference** area changes to a check mark.
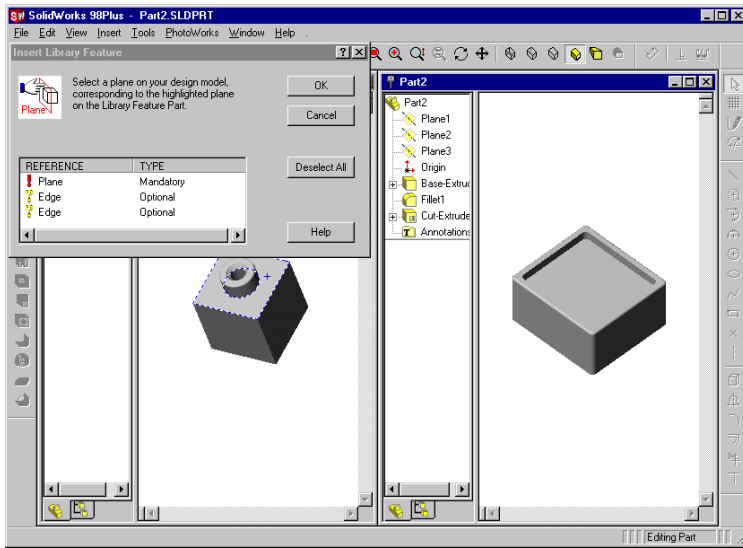
5   Select any optional references. As you click each item in the **Reference** list, notice that the corresponding entity is highlighted *in the library feature window.* Select the corresponding entity *in the target part window.*

To deselect an entity, either double-click the checkmark or click **Deselect All**.

6   Click **OK**.

The library feature is added to the target part.
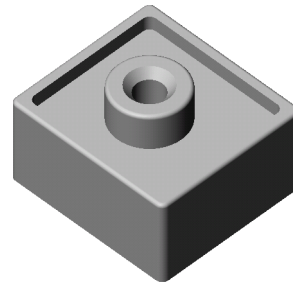
Editing a Library Feature

After the library feature is inserted into a part, you can edit a library feature using the same techniques that you would use to edit any SolidWorks feature, such as **Edit Sketch** or **Edit Definition**.

You can move the library feature to a different location on the target part by modifying the locating dimensions. Double-click the library feature icon in the FeatureManager design tree to expose the dimensions.

You can edit an existing library feature and save it with a new name (and **.sldlfp** extension) to create additional, similar library features.

To delete a library feature from a part, click the library feature icon in the FeatureManager design tree. Press the **Delete** key and click **Yes** to confirm the deletion.

Adding Color

You can apply color to a library feature either in its own document or in a target part.

To change the color of library features:

1   Click **Tools, Options**, and select the **Color** tab.

2   In the **Features** box, select **Library Feature**.

3   Click the **Edit** button and select a color from the color palette (or create a custom color), and click **OK**.

4   Select an option from the **Apply to** list to specify the scope of the changes:

- **Active Document**. Apply the settings to *only* the currently active document.

- **System Defaults**. Apply the settings to all *new* documents. The currently active document does not change.

- **All Possible**. Apply the settings to both the **System Defaults** and the **Active Document**.

5   Click **OK**.

In a library feature document (**.sldlfp**), all the features that are included in the library feature use the new color.

In a target part, all inserted library features use the new color.

Dissolving a Library Feature

After the library feature is inserted into a part, you can dissolve (break down) a library feature into the individual features it contains.

To break down a library feature into the features it contains:

1   In the FeatureManager design tree, right-mouse click the library feature icon.

2   Select **Dissolve Library Feature** from the menu.

The library feature icon is removed, and each of the features it contains is listed individually in the FeatureManager design tree.

The Feature Palette Window

The Feature Palette window gives you quick access to the features and parts you use most often. It helps you organize features and parts in a way that suits your specific needs.

Palette features and library features are similar in many ways. You create a palette feature in the same way as a library feature. The palette feature is displayed as a library feature in the FeatureManager design tree of the target part. You can dissolve a palette feature the same way as a library feature.

Palette features differ from library features in these ways:

❑ When you add a palette feature to a part, you can simply drag and drop the feature roughly where you want it to be, then place it precisely by re-attaching and modifying any locating dimensions saved in the feature.

❑ You can edit the dimensions of the palette feature as you insert it into the part.

❑ Not all library features are suitable for use in the Feature Palette window. A palette feature is limited to *one* mandatory reference (one face, plane, edge, or vertex) on the target part.

❑ Library features that have multiple mandatory references (such as draft features that depend the selection of several faces) may not be added through the Feature Palette window.

You can also use the Feature Palette window to add parts to an assembly. You can organize parts in logical groups, and you can see a miniature graphical view of each palette part. This makes it easy to locate, select, and insert the parts you want.

The Feature Palette is also used to apply forming tools to sheet metal parts. Forming tools are special parts that punch, stretch, or shape the sheet metal part, to create embossed features, lances, flanges, louvers, ribs, etc.

Mandatory References

The limitation of a single mandatory reference does not mean that a palette feature can consist of only one feature. As long as the *parent* feature uses a single reference, the palette feature may include multiple *child* features. To examine the parent/child relationships, right-mouse click the feature, and select **Parent/Child**.

You can determine if a library feature is suitable for use in the Feature Palette by inserting it as a library feature in a target part (see **Adding a Library Feature to a Part** on page 10-3). If there is only one mandatory reference listed, you may use the library feature as a palette feature.

Because of the limitation of a single reference, lofts and sweeps are allowed only in certain cases:

- For lofts, the single mandatory reference is the sketch plane of one of the profiles. This must be the face or plane where you drop the feature. Additional sketch planes must be **Offset** from this plane, or from each other. The library feature must include the planes.

- For sweeps, the sweep *path* is the single mandatory reference. This must be a model edge. The sketch plane for the sweep *section* must be a **Perpendicular to Curve** plane, with the model edge as the curve (when you select the edge, click near the vertex where the plane should be placed). Include this plane in the library feature. Then, when you drop the palette feature in the target part, drop it on the edge that is the sweep path, near the appropriate vertex.

Displaying the Feature Palette Window

The Feature Palette window has several default folders. Each folder represents a directory; each palette item contained in that directory is displayed as a "thumbnail" graphic. Palette items can include parts and sheet metal forming tools (**.sldprt** files), and library features (**.sldlfp** files).

To display the Feature Palette window:

Click **Tools, Feature Palette.** The Feature Palette window stays open, and on top of the SolidWorks window, while you work. If the Feature Palette window is displayed when you exit SolidWorks, it will be displayed when you start SolidWorks the next time.

To dismiss the Feature Palette window, either click ☒ in the upper right corner of the window, or click **Tools, Feature Palette** again to *uncheck* the item in the menu.

You can display the folder icons and the thumbnail graphics of palette items in two sizes. To change the size, right-mouse click a blank area of the window, and select **View, Large Icons** or **Small Icons**.

Navigating in the Feature Palette Window

When you display the Feature Palette window, it opens at the top level, or **Palette Home**. To open a folder, double-click it. These toolbar controls are similar to the controls in Internet Explorer.

- ⬅ **Go Backward**
- ➡ **Go Forward**
- 🔄 **Reload**
- 🏠 **Home** (to **Palette Home**)
- The dropdown history list, with the names of the last folders opened (up to 16).

Organizing Palette Items

The Feature Palette window has several default folders. You can create and name new folders to suit your specific needs, and you can place items in each folder according to the way you want to organize the contents.

The SolidWorks software includes some palette items to get you started. They are stored in these directories:

*install_directory***\data\Palette Features***folder_name*

*install_directory***\data\Palette Parts***folder_name*

*install_directory***\data\Palette Forming Tools***folder_name*

You add individual items to a folder by placing parts, forming tools (**.sldprt**) or library feature parts (**.sldlfp**) in the appropriate directory.

A folder may contain individual palette items, and other folders. You may have as many folders, and as many levels of folders, as you want.

You can store palette items anywhere you want; the folders do not have to be located in the default directory path. You can have folders on different disk drives on your system, or on different computers on a network.

To specify which folders appear at the* Palette Home *level:

1 Click **Tools, Options, External References**.

2 From the **Show folders for list**, select the type of file for which you are defining a folder list: **Palette parts**, **Palette features**, or **Palette forming tools**.

The first time you edit the **Folders** list, the default paths for the palette folders are already listed. These folder lists are always used, whether **Search document folder list for external references** is checked or not (this option applies only to the **Documents** folder list).

You can have more than one folder in the **Palette Home** for each type of item.

- To add a new path to the list, click the **Add** button. In the **Choose Directory** dialog box, locate the path, and click **OK**.

- To delete a path from the list, select the path in the **Folders** list, and click the **Delete** button.

- To change the order of the list, select a path in the **Folders** list and use the **Move Up** or **Move Down** buttons to rearrange the list. Folders appear in the **Palette Home** in the order of the list.

3 Click **OK**. If you change the paths while the Feature Palette window is open, click **Reload** ⬜ to update the paths.

The icons in the Feature Palette window identify the type of item that the folder contains (with large icons only).

Creating a Palette Item

There are several ways to add items to the Feature Palette.

❑ Move or copy existing parts, forming tools (**.sldprt**) or library feature parts (**.sldlfp**) into the appropriate directory, using Windows Explorer utilities.

❑ Drag the item from an open document and drop it into the Feature Palette window.

❑ Drag a document from Windows Explorer and drop it into the Feature Palette window.

❑ Drag a hyperlink to a palette item from the Internet Explorer (version 4.0 or higher), and drop it into the Feature Palette window.

To add a palette item from an open document:

1 Create and save a new part, forming tool, or library feature part, or open an existing document (**.sldprt** or **.sldlfp**).

2 Display the Feature Palette window, and navigate to the folder where you want to store the palette item.

3 Drag the part from the graphics area, or the part icon from the top of the FeatureManager design tree of the active document, and drop it on the Feature Palette window. The **Save As** dialog box appears.

4 Either accept the current name, or enter a new name, and click **Save**.

To add a palette item from Windows Explorer:

1 Display the Feature Palette window, and navigate to the folder where you want to store the palette item.

2 In Windows Explorer, navigate to the directory that contains the palette item.

3 Drag the icon from the Explorer window and drop it in the Feature Palette window.

To add a palette item from a hyperlink in Internet Explorer:

1 Display the Feature Palette window, and navigate to the folder where you want to store the palette item.

2 In Internet Explorer (version 4.0 or higher), navigate to a location that contains hyperlinks to SolidWorks palette item.

For example, on the SolidWorks website, in the Support section, navigate to the Solid Modeling Library page. You may also be able to obtain SolidWorks palette items from certain third-party vendors.

3 Drag a hyperlink from the Internet Explorer window, and drop it in the Feature Palette window. The document is saved in the specified folder.

You can also just *open* a hyperlinked item, without adding it to the palette. To open the document, drag the item and drop it into an empty area of the SolidWorks window. Then you can decide whether you want to add the item to the palette, or save it to a different directory, or close it without saving.

Thumbnail Graphics

When you save a palette item, the thumbnail graphic reflects the current view. Be sure to orient the part so that the thumbnail graphic looks the way you want it to look.

To make it clear which features are included in a palette feature, consider changing the color in the library feature part. Thumbnail graphics use a palette of 16 colors; if the color you choose for the feature is not available, the software attempts to replace it with a similar color in the thumbnail graphic. See **Adding Color** on page 10-5.

NOTE: In the target part, palette features that consist of *more than one feature* use the color specified for **Library Features** under **Tools, Options, Color**.

Palette features that consist of *a single feature* are automatically dissolved when you add them. Therefore they use the color specified for the individual feature type.

Another way to emphasize the features is to select them, then click **Zoom to Selection** 🔍 before you save the library feature part. That way, the thumbnail graphic provides a close-up view of the included features.

Working with Palette Items and Folders

To copy a palette item:

Hold the **Ctrl** key down, then drag and drop a palette item within the Feature Palette window. You can only copy an item this way in the same folder. The new item is named **Copy of *<item>***.

To rename a palette item:

You can rename the palette items and the folders, except the folders at the **Palette Home** level. Click-*pause*-click the name in the Feature Palette window, and enter a new name.

To edit a palette item:

1 Right-mouse click the item in the palette window, and select **Edit Palette Item**.

– or –

Drag the item from the Feature Palette window and drop it in an empty area of the SolidWorks window (not an empty area of a document window).

The part (**.sldprt**) or library feature (**.sldlfp**) document is opened in a new window.

2 Edit the document as needed, then **Save**. The palette updates automatically to reflect the changes.

NOTE: The thumbnail graphic reflects the newly saved view. Be sure to orient the part so that the graphic looks the way you want it to look.

To change the contents of a palette folder:

1 In Windows Explorer, navigate to the directory you want to modify.

2 Add, move, or delete the files as needed, using Windows file management commands.

3 Click **Reload** 🔃 on the toolbar in the Feature Palette window to update the folders.

To delete a palette item:

Click the item in the Feature Palette window, and press **Delete**. The item is removed from the window, and the file is deleted from the corresponding folder.

To delete a palette folder:

1 Exit SolidWorks.

2 Navigate to the directory to delete, move the contents to a new location, if necessary, then delete the directory.

Adding Palette Items to Documents

Adding a Palette Feature to a Part

Try using either **Hidden Lines Removed** or **Hidden in Gray** view mode, in order to see the preview and dimensions easily.

To add a palette feature to a part:

1 With a part open, click **Tools, Feature Palette**, and navigate to the folder that contains the feature.

2 Drag the feature from the Feature Palette window, observing the preview as you drag. Drop the feature in the general area of the face where you want the feature to be placed.

> **NOTE:** You can also drag a hyperlink to a library feature part (**.sldlfp**) from Internet Explorer (4.0 or higher) and drop it on the face of the part.
>
> If you want to save the hyperlinked palette feature for future use, be sure to also drag a copy into the palette, or into an empty area of the SolidWorks window, then save it. Otherwise, no local copy of the document is saved.
>
> Hyperlinked library feature parts have the same limitations and behaviors as library feature parts in the Feature Palette window.

The **Edit this Sketch** dialog box appears, and remains open while you position the sketch.

3 Click **Zoom to Selection** 🔍 to get a closer view of the feature.

4 Click **Modify Sketch** , and move or rotate the sketch as needed. See **Modify Sketch** on page 2-19 for more information.

Modify Sketch cursor

If the feature has locating dimensions, they are left dangling, and are displayed in brown.

5 Re-attach the dangling dimensions. Select a dimension, then drag the red handle on the dimension line.

The cursor has the $\bigcirc$ shape until it is over a suitable entity for re-attachment. When you release the handle over an edge or vertex, the dimension re-attaches and the handle turns green.

Dangling dimensions

You can also add geometric relations to control the position of the sketch.

6 Adjust the values of the locating dimensions. (Double-click the dimension, change the value, and press **Enter**.) You can also adjust the locating dimensions in the next step, or later, by editing the appropriate sketch.

7 Click **Next** in the **Edit this Sketch** dialog box. The **Change Dimensions** dialog box appears.

The **Name** and **Value** of each available dimension is displayed. If the dimensions are named in the part where you created the library feature, those names are used. Otherwise, the default dimension names *D1* through *Dn* are used. See **Dimension Properties** on page 2-26.

Some dimensions of the feature may be unavailable, because they are specified as **Internal Dimensions**. See **Controlling Access to Dimensions** on page 10-14 for more information.

8 To display a feature dimension on the model, click either the **Name** or **Value**.

To modify a dimension, double-click the **Value**, and enter a new value. Click **Apply** to see the changes.

9 When you are satisfied with the dimension values, click **Finish**.

If the palette item consists of more than one feature, a library feature icon appears in the FeatureManager design tree. If the palette item consists of a single feature, the item is automatically dissolved, and the appropriate icon for the feature type appears in the FeatureManager design tree. In either case, the name of the feature is the name of the item in the Feature Palette window.
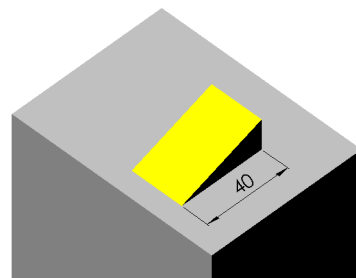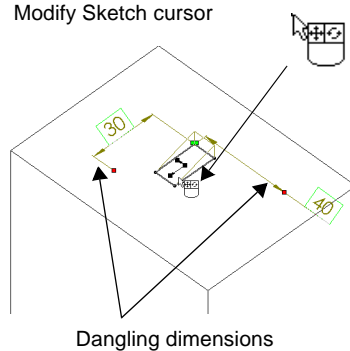
Adding a Palette Part to an Assembly

To add a palette part to an assembly:

1   With the assembly open, click **Tools, Feature Palette**, and navigate to the folder that contains the desired part.

2   Drag the part from the Feature Palette window and drop it into the assembly window.

If the part has multiple configurations, the **Select a configuration** dialog box appears, with a list of the configurations in the part. Choose the configuration you want, and click **OK**.

For information about positioning and mating assembly components, see Chapter 6, "Working with Assemblies."

Applying a Palette Forming Tool to a Sheet Metal Part

You can only apply forming tools through the Feature Palette window, and you can only apply forming tools to a part that has sheet metal features. You must have already inserted bends in the part before you attempt to apply a forming tool.

For information about creating sheet metal parts and forming tools, see Chapter 11, "Sheet Metal."

To apply a palette forming tool to a sheet metal part:

1   With a sheet metal part open, click **Tools, Feature Palette**, and navigate to the folder that contains the forming tools.

2   To apply forming tools to the sheet metal part in a flattened state, drag the rollback bar above the **Process-Bends** feature.

3   Drag the forming tool from the Feature Palette window to the face you want to deform.

The face where you apply the forming tool corresponds to the stopping surface of the tool itself. By default, the tool travels downward. The material is deformed when the tool strikes the face where it is dropped. To toggle the direction of travel, and strike the opposite side of the material, press the **Tab** key. Observe the preview as you toggle the direction.

4   Drop the feature where you want it to be applied. If the forming tool includes an optional positioning sketch, that sketch is displayed when you drop the tool on the face.

5   Use **Dimension** 🖉 or **Modify Sketch** 🖫 to position the feature.

As you add dimensions, the positioning sketch moves as a single entity. You cannot edit the dimensions of an applied forming tool. The absorbed sketch in the feature controls only the *location* of the feature, not its dimensions.

6   Click **Finish**. The forming tool is applied to the face, and the feature is added to the FeatureManager design tree, with the name of the forming tool.

Palette Parts as Derived Parts

You can use the Feature Palette window to create a derived part, based on the selected palette part.

1 With an *empty* part document open, click **Tools, Feature Palette**, then navigate to the folder that contains the desired part.

2 Drag the part from the Feature Palette window and drop it into the part window.

3 When you are asked if you want to create a derived part,

- Click **Yes** if you want to create a derived part (a part which has another part as its first feature). See **Derived Parts** on page 5-13 for more information.

- Click **No** if you want to edit the palette part itself. The palette part is opened in the current window.

- Click **Cancel** to abandon the operation.

Controlling Access to Dimensions

For some palette items, you may want to prevent other users from changing certain dimensions. You can specify which dimensions of a palette item are displayed and available for editing through the palette, and which ones are not.

You can also use this function to reduce the clutter of unnecessary dimensions when adding items through the palette.

To control dimension access:

1 Right-mouse click the item in the palette window, and select **Edit Palette Item**. The part (**.sldprt**) or library feature (**.sldlfp**) document is opened in a new window.

2 Right-mouse click the part icon at the top of the FeatureManager design tree, and select **Edit Dimension Access**.

In the **Dimension Access** dialog box, there are two lists: **User Dimensions** (those that may be edited) and **Internal Dimensions** (those that may not).

3 Specify which dimensions are available:

- To move a dimension from one list to the other, click the name, then click the left or right arrow button. As you click each name, the dimension is displayed in the model.

- To move all the dimensions, use the double arrow button.

When you insert a palette feature that has restricted dimension access, only the **User Dimensions** are displayed and available for editing through the wizard. The **Internal Dimensions** are not displayed, and they are not available for editing.

11

Sheet Metal

This chapter introduces the SolidWorks sheet metal functionality and describes:

- Designing sheet metal parts
- Bend types
- Sheet metal features in the FeatureManager design tree
- Rolling back and rebuilding the design
- Bend allowances
- Auto relief
- Creating sheet metal parts
- Adding additional features
- Editing bends
- Creating a flat pattern configuration
- Sheet metal drawings
- Forming tools

Designing Sheet Metal Parts

Sheet metal parts are generally used as enclosures for components or to provide support to other components.

You can design a sheet metal part on its own without any references to the parts it will enclose, or you can design the part in the context of the assembly containing the enclosed components. In the latter case, you can reference the enclosed components to create the geometry for the sheet metal part.

Creating Bends

Before...   ...and after inserting bends

You can create a solid model and then create bends in the model by specifying appropriate bend parameters such as bend radius and allowance.

Inserting bends in the model yields the results shown here. The sharp corner is replaced by a filleted bend.
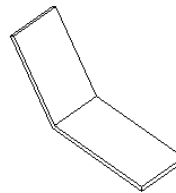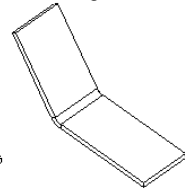
Creating Bend Reliefs, Walls, and Tabs

Walls or tabs can be added to a sheet metal part as designs dictate.

The **Auto Relief** option automatically adds bend reliefs where necessary.
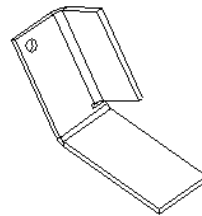
Creating Flat Patterns

When the design is complete (or at any time during the design process) you can create a flat pattern of your sheet metal part.
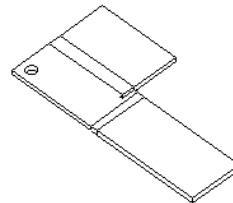
Bend Types

There are three types of bends in SolidWorks: sharp bends, round bends, and flat bends.

Sharp Bends

A *sharp bend* is created by adding bends to a model that was extruded as a thin feature or that was extruded as a solid feature of uniform thickness. The original bends were defined in the sketch. Click **Insert Bends** 🔧 or **Insert, Features, Bends** to create the model on the right. See **Creating a Sheet Metal Part Using Sharp Bends** on page 11-10.
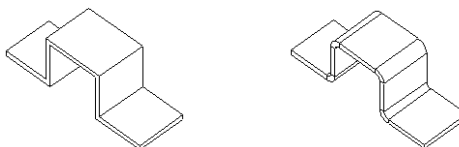
Round Bends

A *round bend* is created from filleted corners created in the sketch, during the thin feature extrusion using the **Auto Fillet** option, or from a cylindrical face. When you click **Insert Bends** 🔧 or **Insert, Features, Bends**, the model does not change in appearance. However, bend lines and allowances are added to the part, and are visible when the model is flattened. See **Creating a Sheet Metal Part Using Round Bends** on page 11-13.
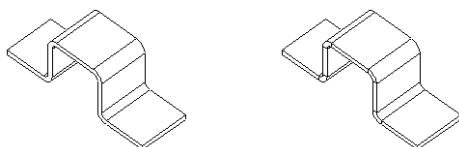
Flat Bends

A *flat bend* is created from a line sketched in the flat state of a sheet metal part. Bend lines are sketched as shown in the flat model on the left. Exiting the sketch creates the model on the right. See **Creating a Sheet Metal Part from a Flat Model** on page 11-15.
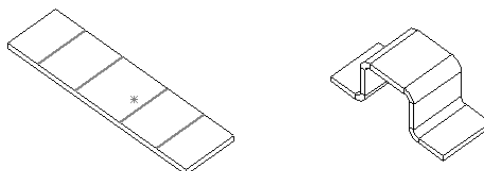
Sheet Metal Features in the FeatureManager Design Tree

When you click **Insert Bends** � or **Insert, Features, Bends**, two distinct stages are applied to the sheet metal part. First, the part is flattened and a bend allowance is added. The developed length is calculated, based on the bend radius and bend allowance. Then, the flattened part is restored to the folded state to create the formed, folded version of the part. Three new features and icons appear on the FeatureManager design tree that are specific to sheet metal operations. These features are:

❑ **Sheet-Metal**

❑ **Flatten-Bends**

❑ **Process-Bends**

These three features represent what can be thought of as a process plan for the sheet metal part.
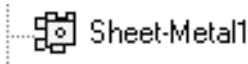
Sheet-Metal contains the definition of the sheet metal part itself and represents the result of the **Insert, Features, Bends** operation.

This feature stores the default bend parameter information (thickness, bend radius, bend angle, and auto relief offset ratio) for the entire part.

Flatten-Bends represents the flattened part. This feature contains information related to the conversion of sharp and filleted corners into bends.

Each bend generated by SolidWorks from the model is listed as a separate feature under **Flatten-Bends**. Bends generated from filleted corner edges are listed as **Round-Bends**; bends generated from sharp corner edges are listed as **Sharp-Bends**.

The **Sharp-Sketch** listed under **Flatten-Bends** is the sketch that contains the bend lines of all sharp and round bends generated by the system. This sketch cannot be edited but can be hidden or shown.
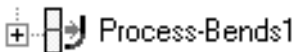
Process-Bends represents the transformation of the flattened part into the finished, formed part.

Bends created from bend lines specified in the flat are listed under this feature. **FlatSketch,** listed under **Process-Bends**, is a placeholder for these bend lines.

Features that are listed after the **Process-Bends** icon in the FeatureManager design tree do not appear in the flattened view of the part. (You flatten the view of the part using Rollback. See **Rolling Back and Rebuilding the Design** on page 11-6.)

For example, in the sheet metal part shown here, the cut was added after the **Process-Bends** feature.
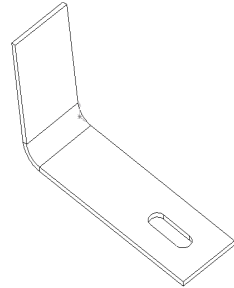
If you rollback to **Process-Bends**, the flattened part does not include the cut.
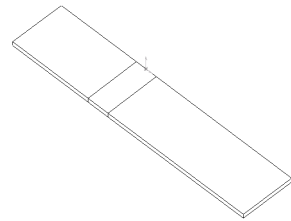
To see the cut in the flattened state, reorder the cut to occur before the sheet metal feature.
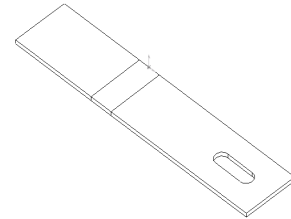
Rolling Back and Rebuilding the Design

A *rollback* reverts the model to the state it was in before the selected feature was created. When you design sheet metal parts, you may use rollback often to revert to previous stages in the design process. This section discusses the ways you can perform a rollback.

To rollback (Method 1):

1   In the FeatureManager design tree, select the feature you want to rollback to.

2   Click **Edit, Rollback**.

To rollback by dragging the rollback bar (Method 2):

1   Place your cursor over the rollback bar in the FeatureManager design tree. The cursor changes to a hand.

2   Click to select the rollback bar. The bar changes color from yellow to blue.

3   Drag the rollback bar up the FeatureManager design tree until it is above the feature you want rolled back

– or –

Use the up and down arrow keys on the keyboard to move the rollback bar up or down. (Check **Arrow key navigation** in **Tools, Options, General** to enable this functionality.)

> **NOTE:**   You do not have to select a feature before you move the rollback bar. Also, the rolled-back icons are gray.
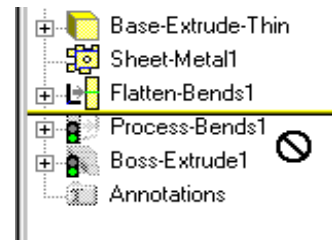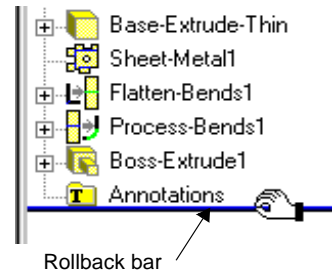
To rollback to before Process-Bends (Method 3):

1   Click the **Flattened** ⬚ tool on the Features toolbar.

2   Click **Flattened** ⬚ again to restore the bends.

To rollback to before Sheet-Metal, before any bends were inserted (Method 4):

1   Click the **No Bends** ⬚ tool on the Features toolbar.

2   Click **No Bends** ⬚ again to restore the bends.

Throughout the following sections, whenever you need to perform a rollback, you can use any of these methods.

To rebuild (roll forward):

To rebuild (roll forward) your part after performing a rollback, drag the rollback bar down the FeatureManager design tree. You can drag the rollback bar to the bottom of the FeatureManager design tree to rebuild the entire part, or drag it one or more features at a time to step through the regeneration of each rebuilt feature. Throughout this section, whenever you need to perform a rebuild, drag the rollback bar down to the bottom of the FeatureManager design tree.

Bend Allowances

There are three ways to specify a sheet metal bend allowance. You can:
- Specify a K-Factor
- Specify an explicit value
- Use a bend table

K-Factor

K-Factor is a ratio that represents the location of the neutral sheet with respect to the thickness of the sheet metal part. Bend allowance using a K-Factor is calculated as follows:

$$BA = \Pi/2(R + KT) \, A/90$$

where:

BA = bend allowance
T = material thickness
R = inside bend radius
A = bend angle in degrees
t = distance from inside face to neutral sheet
K = K-Factor, which is t / T
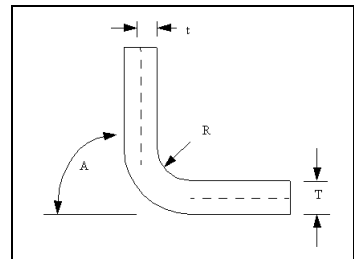
NOTE: The angle A is the angle through which the material is bent, hence it is not always the angle shown on the drawing.

Explicit Bend Allowance Value

You can specify an explicit bend allowance for any sheet metal bend by entering the value when creating the bend.

NOTE: For a given bend radius and angle, the specified bend allowance value should be between the length of the inner edge and the outer edge of the bend.

Bend Table

A sample bend table is provided in *lang\english\sample.btl*. To use your own bend table when performing sheet metal operations, copy and rename this table and edit it to specify required bend allowances.

NOTE: Bend table units must be specified in metric units.

You can edit the bend table using any text editor. A portion of the sample bend table is shown below:

```
 sample.btl - Notepad                                                 _ □ ✕
File  Edit  Search  Help

Bend Allowance Tables
---------------------
Version: 1
Material: Steel
Units: meters


Thickness: 0.0005
Bend Radius (read across)      0.0000  0.0005  0.0010  0.0015  0.0020  0.0025  0.0030
Opening Angle (read down)
                       5       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                      10       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                      20       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                      30       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                      45       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                      50       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                      70       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                      80       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                      90       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                     100       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                     110       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                     120       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                     130       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                     140       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                     145       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                     160       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                     170       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002
                     180       0.0002  0.0002  0.0002  0.0002  0.0002  0.0002  0.0002


Thickness: 0.0010
Bend Radius (read across)      0.0000  0.0005  0.0010  0.0015  0.0020  0.0025  0.0030
```

NOTE: The sample bend table is provided only for informational purposes. The values in this table do not represent any actual bend allowance values.

If the thickness of the part or bend angle falls between values in the table, the software interpolates the values to calculate the bend allowance.

Auto Relief

You can automatically add relief cuts wherever needed when inserting bends.

For example, you can use the **Auto Relief** option to add relief cuts in the model shown here.
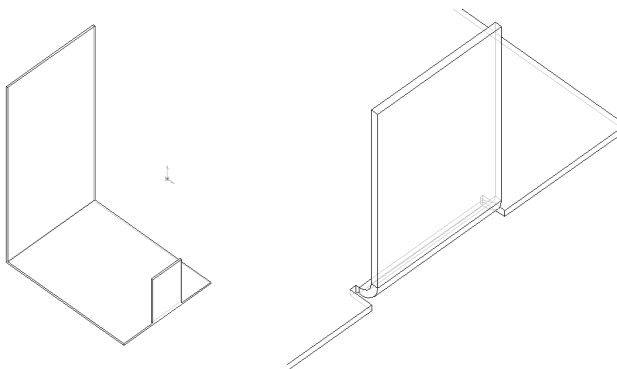
Relief cuts are added automatically at both sides of the wall in this example. (If only one relief cut is needed, only one is added.) The cuts added are of the minimum size required to insert the bend and flatten the part.

When you choose the **Auto Relief** option, you must specify the **Offset Ratio.**

Offset Ratio

The **Offset Ratio** represents the distance, *d*, by which the side of the square relief cut extends past a bend region, expressed as a ratio of material thickness.

d = (offset ratio) * (part thickness)

NOTE: Offset Ratio must have a value between 0.05 and 2.0.

The higher the value of the offset ratio, the larger the size of the relief cut automatically added during insertion of bends.
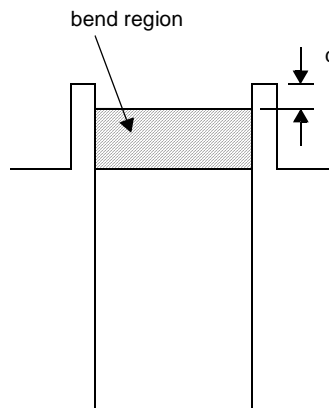
Creating a Sheet Metal Part Using Sharp Bends

The following example demonstrates the creation of a sheet metal part based on a thin feature base. You can create a sheet metal part from any part that has a uniform thickness, and a thin feature extrusion is a quick way to create such a base.

To create a thin-feature base:

1 Create a new part, open a sketch, and sketch an open profile for the base.

2 Click **Extruded Boss/Base** 🔲 or **Insert, Base, Extrude**.

3 In the **Extrude Feature** dialog box, specify the desired **Depth** and **Type**.

4 On the **Thin Feature** tab, specify the desired **Type** and **Wall Thickness**.

5 Click **OK**.

   For more information about thin feature extrusions, see **Thin Features** on page 4-6.
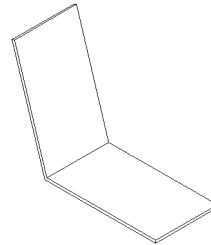
To add a wall:

1 Open a sketch on the face of the part where the new wall will be attached.

2 Select the edge of the model and click **Convert Entities** 🔲 or **Tools, Sketch Tools, Convert Entities**.

3 Drag the vertex nearest to the bend a small distance away from the bend to allow for the bend radius.

4 Click **Extruded Boss/Base** 🔲 or **Insert, Boss, Extrude**.

5 On the **End Condition** tab, set the **Type** to **Blind** and specify the **Depth**.

6 On the **Thin Feature** tab, specify the same **Thickness** as the base feature.

7 Click **OK**.
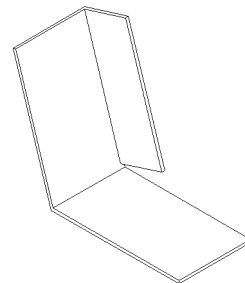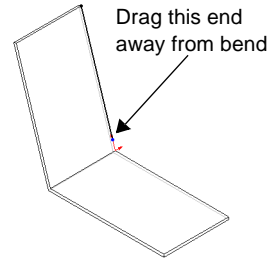
   The new wall is added. It is generally a good practice to add all the walls before inserting the bends.

Drag this end away from bend

To insert the bends:

To insert bends, you must first pre-select a *fixed face*. The fixed face remains in place when flattening the part during the creation of the **Flatten-Bends** feature.

1 In the model, select the face you want to remain fixed.

2 Click **Insert Bends** 🔧 or **Insert, Features, Bends**.

The **Flatten-Bends** dialog box appears.
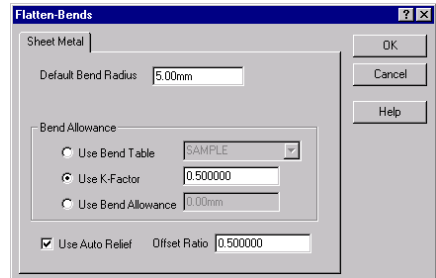
3 Enter a **Default Bend Radius**.

4 Select a **Bend Allowance** type and value:

   • If you select **Use Bend Table**, select the name of the table file. See **Bend Table** on page 11-8 for more information.

   • If you select **Use K-Factor**, specify the value based on the calculation. See **K-Factor** on page 11-7 for more information.

   • If you select **Use Bend Allowance**, specify the value.

5 Make sure **Auto Relief** is checked, and specify an **Offset Ratio**. See **Offset Ratio** on page 11-9 for more information.

6 Click **OK**.

The sharp corner edges of the part are replaced by bends, and the necessary relief cut has been added at the side wall.

7 Double-click the **Flatten-Bends** feature (or click the ⊞ beside its name) to expand the feature.

Notice the **Sharp-Bend** features and a **Sharp-Sketch** feature are listed under it.
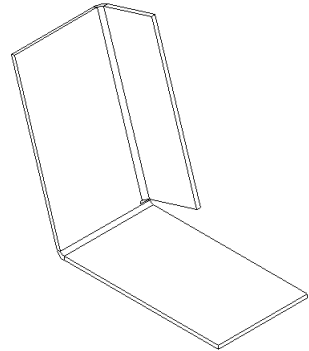
Adding Additional Features

When designing sheet metal parts, it may be necessary to work on the flattened state to add features such as an oblong cut. You can rollback to the flattened state of a sheet metal part by clicking the **Process-Bends** feature and then clicking **Edit, Rollback**, or by dragging the rollback bar in the FeatureManager design tree.

To create a cut across a bendline:

1   Click **Flattened** ![icon] to rollback above the **Process-Bends** feature.

The part is shown in the flattened state.

2   Select the top face of one of the flat regions and open a sketch.

3   Sketch the profile of the cut.

4   Click **Extruded Cut** ![icon] or **Insert, Cut, Extrude**.

5   Set the **Type** to **Through All**, and click **OK**.
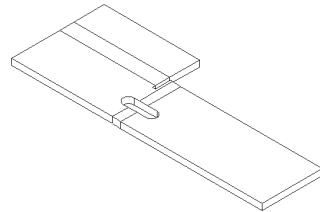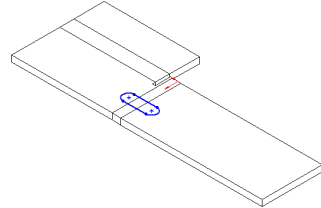
To create a tab:

1   While the part is still rolled back, open a sketch on one of the faces.

2   Sketch a tab with one side coincident with a model edge.

3   Click **Extruded Boss/Base** ![icon] or **Insert, Boss, Extrude**.

4   In the **Extrude Feature** dialog box:

   • Click **Link to Thickness** to set the boss thickness equal to the base thickness.

   • Click **Reverse Direction** if the extrusion is extending the wrong way.

5   Click **OK**.

6   Click **Flattened** ![icon] to rebuild the part.
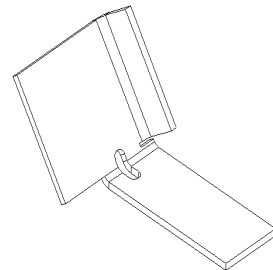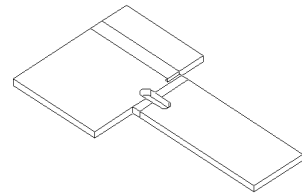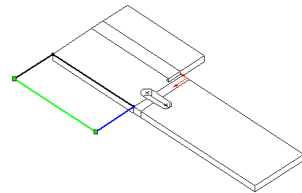
The part is folded back with the added cut and tab.

Creating a Sheet Metal Part Using Round Bends

To create a sheet metal part using round bends:

1 Create a thin feature part:

- Check the **Auto Fillet** option in the **Thin Feature** tab of the **Extrude Thin Feature** dialog box.
- Specify the **Fillet Radius** (inner bend radius).

2 Select the fixed face.

3 Click **Insert Bends** 🛠 or **Insert, Features, Bends**.

4 In the **Flatten-Bends** dialog box:

- Specify a **Default Bend Radius** of 0.
- Specify a **Bend Allowance** method and value.

5 Click **OK**.
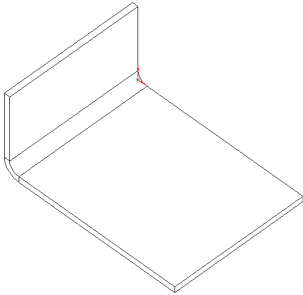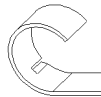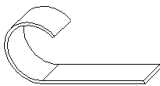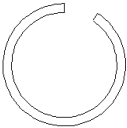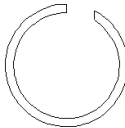
A **RoundBend** appears in the FeatureManager design tree under **Flatten-Bends**.

Creating a Sheet Metal Part with Cylindrical Faces

To create a sheet metal part with cylindrical faces:

1 Create a thin feature part with one or more cylindrical faces:

- Each cylindrical face must have four bounding edges.
- Any adjacent planar and cylindrical faces must be tangent.
- Cylindrical end faces must be normal to the surface of the cylinder.

Correct		Incorrect	
Four bounding edges		More than four bounding edges	
Faces tangent		Faces not tangent	
End faces normal		End faces not normal	

1 Select a linear edge of an end face of a cylindrical face or select a fixed planar face.

2 Click **Insert Bends** ![icon] or **Insert, Features, Bends**.

Select this edge
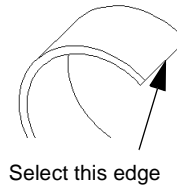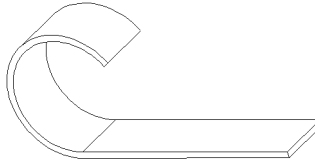
3 In the **Flatten-Bends** dialog box:

   • Specify a **Default Bend Radius** of 0.

   • Specify a **Bend Allowance** method and value.

4 Click **OK**.

   A **RoundBend** appears in the FeatureManager design tree under **Flatten-Bends**.

NOTE: You cannot add bosses or apply forming tools to cylindrical faces on sheet metal parts.

Creating Hems

You can make hems of various types in sheet metal parts. SolidWorks supports open, closed, double, and tear-drop hems.

To create a hem:

1 In an existing sheet metal part, click the ![+] beside a **Process-Bends** feature.

2 Right-mouse click the sketch and select **Edit Sketch**.

3 Sketch a line to define the fold of the hem, and exit the sketch.
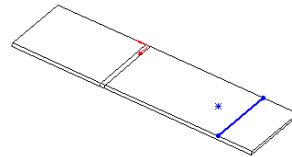
   The part folds on the sketched line, using the default radius and angle. You need to edit the bend to give it radius and angle values suitable for a hem.

4 Right-click the **FlatBend** feature and select **Edit Definition**.

5 Enter new values for **Radius** and **Angle**.

NOTE: The bend radius should allow for a small gap between overlapping faces so that they do not touch.

---

6 Click **OK**. The hem returns to the folded state using the new values.
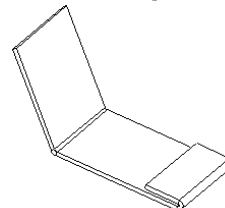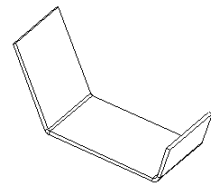
Creating a Sheet Metal Part from a Flat Model

While you should design sheet metal parts in the bent-up state (using either sharp corners or filleted corners), you can also model your parts from a flat sheet. You need to know the developed length required for your part and the locations of the bend lines for all bends in the part.

To create a sheet metal part from a flat model:

1   Create a flat part. You can use either a solid or thin feature extrusion. The requirement here is that the model be of uniform thickness.

2   Select the *fixed face*.

3   Click **Insert Bends** or **Insert, Features, Bends**.

4   In the **Flatten-Bends** dialog box, enter the **Bend Radius**, and specify the **Bend Allowance** method and value.

    The sheet metal features appear in the FeatureManager design tree even though the part appears unchanged.

5   Expand the **Process-Bends** feature in the FeatureManager design tree.

6   Right-click the **Flat-Sketch** under **Process-Bends** and select **Edit Sketch**.

7   Sketch lines where you want bend lines.

8   Place a sketch point on the face that you want to remain fixed.

9   Exit the sketch to rebuild the part.

    Bends are created using the sketched bendline and the specified bend parameter information.

    NOTE: You cannot specify a bendline for a flat bend that lies between existing sharp or round bends.
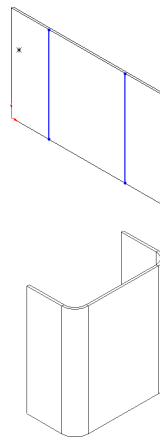
Editing Bends

You can edit bend information for sheet metal features. The scope of the changes depends upon which feature is edited.

To edit bend parameters for the entire part:

1   Right-click the **Sheet-Metal** feature in the FeatureManager design tree and select **Edit Definition**.

2   In the **Sheet-Metal** dialog box, change the **Default Bend Radius** or **Bend Allowance** method as desired.

3   Click **OK**.

To edit bend parameters for all bends listed under Flatten-Bends:

1   In the FeatureManager design tree, right-mouse click the **Flatten-Bends** feature and select **Edit Definition**.

2   In the **Flatten-Bends** dialog box, change the **Default Bend Radius** or **Bend Allowance** method as desired.

3   Click **OK**.

To edit bend parameters for all bends listed under Process-Bends:

1   In the FeatureManager design tree, right-mouse click the **Process-Bends** feature and select **Edit Definition**.

2   In the **Process-Bends** dialog box, change **Default Bend Radius** or **Bend Allowance** method as desired.

3   Click **OK**.

To edit bend parameters for an individual bend:

1   In the FeatureManager design tree, right-mouse click the bend you want to edit and select **Edit Definition**.

2   Change **Radius**, **Angle**, **Bend Direction,** and **Bend Allowance** as desired.

3   Click **OK**.

Notes on editing bends:

❑   You cannot change the bend angle for sharp bends. This angle is calculated from the model.

❑   You cannot change the bend angle or radius for round bends. The bend angle is calculated from the model, and the bend radius is specified in the sketch or in the definition of the fillet feature.

Creating a Flat Pattern Configuration

Creating a flat pattern is equivalent to suppressing the last **Process-Bends** feature and all features after it. You can save this as a configuration to use in a drawing.

To create a flat pattern configuration:

1 Create and name a new configuration. (See **Working with Part Configurations** on page 5-15 for information about creating a configuration.)

2 Select the last **Process-Bends** feature in the model and all features after it.

3 Click **Edit, Suppress**. Also, you can right-mouse click the **Process-Bends** feature, select **Properties**, and click the **Suppressed** check box.

4 Save the configuration.

You should create the flat pattern configuration after the entire design of the part is completed. This ensures that all features are shown in the flat pattern.

Sheet Metal Part Drawings

You can create drawings of sheet metal parts. For more information on creating drawings, see Chapter 8, "Drawings."

To create a drawing of a flat pattern:

1 Create a part configuration showing a flat pattern.

   If none of the standard views shows the flat part in such a way that the bend lines are clearly displayed, create a named view in a suitable orientation.

2 Create a new drawing.

3 Click **Insert, Drawing View, Named View**.

4 Change to the part window, click anywhere, then select the desired view from the **View Orientation** list.

5 Return to the drawing window and click to place the view on the sheet.

To remove display of the bend region lines:

1 In the drawing window, right-click the drawing view in the FeatureManager design tree.

2 Select **Tangent Edge, Tangent Edges Removed**.

To show the bend lines in the view of the flat pattern:

1 Either in the part document or the drawing view, expand the **Flatten-Bends** and **Process-Bends** features.

2 Right-click **Sharp-Sketch** and **Flat-Sketch** in the FeatureManager design tree, and click **Show**.

Tips for Designing Sheet Metal Parts

The order in which you add features is key to getting the desired results. The recommended approach for designing sheet metal parts is as follows:

❑ Design the part in the bent-up state, if possible.

❑ Add **Insert, Features, Bends** only after the functional design is complete; that is, add all the features before inserting bends.

❑ If some features such as bend reliefs and additional walls need to be added after inserting bends, add these features after doing a **Rollback** to the **Process-Bends** feature. This step will rollback the model to the flattened state.

❑ If you do not want certain features to be visible in the flattened state (features such as holes that are drilled after the sheet metal part is bent), add these features *after* the **Process-Bends** feature.

❑ To ensure that the part has uniform thickness, use the **Link to Thickness** features in the **Extrude** dialog box.

Forming Tools

Forming tools act as dies that bend, stretch, or otherwise form sheet metal. You apply forming tools to sheet metal parts to create such form features as louvers, lances, and ribs.

Create forming tools using bosses, cuts, fillets, chamfers, etc., and save them as SolidWorks parts on the Feature Palette. You can only use forming tools from the Feature Palette, and you can only apply them to sheet metal parts. For more information about using forming tools, see **Applying Forming Tools** on page 11-22.

The SolidWorks software includes some sample forming tools to get you started. They are stored in this directory:

> *install_directory***data\Palette Forming Tools***folder_name*

Examine the sample forming tools before creating your own. You may find that by editing a sample forming tool, you can create a forming tool that meets your needs. For more information about editing an existing forming tool in the Feature Palette, see **Working with Palette Items and Folders** on page 10-10.

Creating Forming Tools

To create a forming tool:

1 Open a new part document, then open a sketch on the horizontal plane. By default, the horizontal plane is **Plane2.**

2 To create the base feature, sketch a profile, and extrude the profile *above* the horizontal plane.

3 To create the boss feature, select the bottom face of the base feature, or select the horizontal plane, and open a sketch.

The horizontal plane is the *stopping surface* of the forming tool. To strike and deform sheet metal using a forming tool, the direction of travel of the forming tool must be along the vertical axis. Thus, the entire forming tool, except the base on which you create it, must lie below the stopping surface.
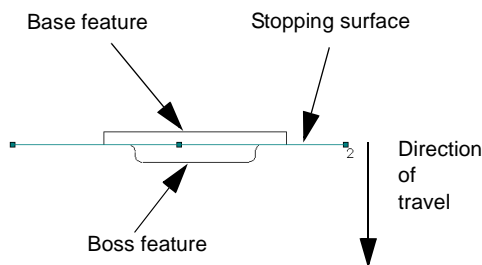
4 Sketch a profile centered around the origin.

The origin determines the drop point of the forming tool when applying the forming tool to a sheet metal part.

5 Dimension the profile, then extrude the profile *below* the stopping surface. The boss can be extruded or revolved.

To shape the forming tool, add features such as cuts, fillets, chamfers, etc., to the boss.

> **NOTE:** The minimum radius of curvature of the forming tool must be greater than the thickness of the sheet metal part to which it will be applied. To find out the minimum radius of curvature of the forming tool, use **Tools, Check**. For more information, see **Check Entity** on page 5-23.

6 Remove the base feature of the forming tool.

When you apply the forming tool to a sheet metal part, the stopping surface of the forming tool corresponds to the sheet metal face to which it is applied.

> **TIP:** To remove the base feature of the forming tool, you can select the bottom face of the base feature, and open a sketch. Project the face onto the sketch plane by clicking **Convert Entities** ⬚ . Click **Extruded Cut** ⬚ , and add a **Through All** cut.

To create an opening on a forming tool:

Some types of form features, such as louvers and lances, create openings on sheet metal parts. To indicate which forming tool faces create openings, change the color of these faces to *red*.

1 Right-mouse click the face designed to create an opening, and select **Face Properties**.

2 In the **Entity Property** dialog box, click **Change Color**.

3 Select the red box in the first column in the second row, then click **OK**. (Under **Entity Color, Red** should be 255; **Green** and **Blue** should be 0.)

Make this face red to indicate an opening on the sheet metal part

4 Click **OK** again to close the **Entity Property** dialog box.

Creating Positioning Sketches

When you apply a forming tool to a sheet metal part, you cannot change the dimensions of the forming tool. You can, however, edit the position of the forming tool on the sheet metal face.

NOTE: To change the dimensions of the forming tool after you apply it, you must edit the dimension access. See **Controlling Access to Dimensions** on page 10-14 for more information.

You can create a *positioning sketch* on the top face of the forming tool. When you apply the forming tool to the sheet metal part, you use the positioning sketch to locate the form feature on the face of the sheet metal part.

To create a positioning sketch:

1 Select the top face of the boss feature created in the previous section, and open a sketch.

2 Create a profile to locate the form feature on the sheet metal part. Close the sketch, and rename it if desired.

Positioning sketch

TIP: To create a profile to locate the form feature, you can select the top face of the forming tool and open a sketch. Project the face onto the sketch plane by clicking **Convert Entities** 🗇, then close the sketch.

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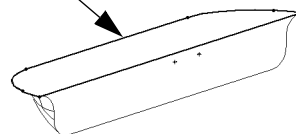
3 To hide the positioning sketch so that it does not appear in the thumbnail graphic of the forming tool displayed in the Feature Palette, right-mouse click the sketch in the FeatureManager design tree, and select **Hide**.

Saving and Adding Forming Tools to the Feature Palette

1 Save the part document **(.sldprt)**.

2 Add the forming tool to the Feature Palette using one of the following methods:

- Move or copy the forming tool part document to the desired Palette forming tools directory using Windows Explorer utilities. The default Palette forming tools directory path is *install_directory***data\Palette Forming Tools***folder_nam*e.

- Click **Tools, Feature Palette**, then navigate to the desired forming tools folder. Drag the part icon from the FeatureManager design tree of the forming tool part, and drop it on the Feature Palette window.

  If you want to store the forming tool in a different location, you must specify the directory path for the Feature Palette to use. For more information, see **Organizing Palette Items** on page 10-8.

Applying Forming Tools

You can only apply forming tools to a part that has sheet metal features; you must have already inserted bends in the part before you attempt to apply a forming tool.

Apply the forming tool to the sheet metal part by dragging the forming tool from the Feature Palette window and dropping it on a sheet metal face. By default, the forming tool deforms the opposite face of the material to which it is applied. To deform the face to which the forming tool is applied, press the **Tab** key as you drag. When you press the **Tab** key, the preview updates automatically.

A form feature is added to the FeatureManager design tree. Locate the form feature on the sheet metal part by using its positioning sketch. For more information, see **Applying a Palette Forming Tool to a Sheet Metal Part** on page 10-13.

You can create patterns of form features. See **Pattern/Mirror** on page 4-42 for more information.

12
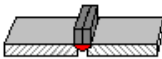
Welding

This chapter introduces the SolidWorks welding feature and describes:

❑ Weld types

❑ Top surface delta and radius calculations

❑ General procedures for creating a weld

❑ Examples of creating V Butt and Square Butt welds

❑ Editing a weld

Weld Types

You can add a variety of weld types to an assembly. The weld types available depend on the dimensioning standard you use (see page 9-2).

The ISO weld types and their symbols are shown in the following table:

ISO Weld Type	Symbol	Illustration
Butt		
Square Butt		
Single V Butt		
Single Bevel Butt		
Single V Butt with Root		
Single Bevel Butt with Root		
Single U Butt		
Single J Butt		
Backing Run		
Fillet		
Seam		

The ANSI weld types and their symbols are shown in the following table:

ANSI Weld Type	Symbol	Illustration
Square			
Scarf		
V Groove		
Bevel		
U Groove		
J Groove		
Flare-V		
Flare-Bevel		
Fillet		
Seam		
Back or Backing		
Flange-Edge		
Flange-Corner		

Top Surface Delta and Radius Calculation

When you are adding any type of weld to the model, you must specify the top surface delta. For a fillet weld, you also specify the radius, and for a backing run weld, you specify the bottom surface delta.

Top Surface Delta

The top surface delta is the distance between the edge of the bead and the top surface of the welded components.

These examples illustrate the top surface delta for a convex weld and a concave weld. The calculation is the same but the distance is measured *below* the top surface for a concave weld rather than *above* for a convex weld.

> **NOTE:** A backing run weld has both a top surface and a bottom surface delta.
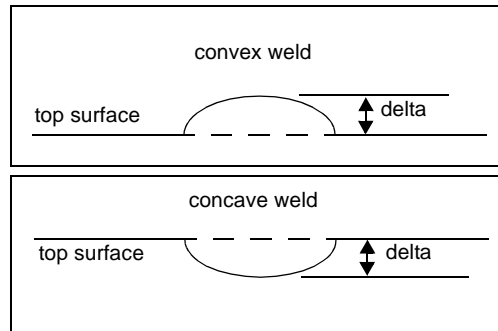
Radius (Fillet Welds Only)

For a fillet weld, you need to specify the radius of the weld. The radius, *r*, is measured from the point where the parts join.
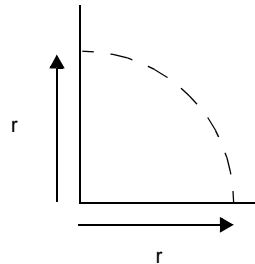
General Procedure for Welding Components

To weld two components:

1   Create an assembly and mate the components, using mates that are suitable to the specific weld type you plan to use. Refer to the section **Examples of Creating Weld Beads** on page 12-7 for more information.

2   Click **Insert, Assembly Feature, Weld Bead**.

3   In the **Weld Bead Type** dialog box, select a weld **Type** from the list.

4   Click **Next**.

5   In the **Weld Bead Surface** dialog box:

• Select a **Surface Shape** from the list.

• Specify the **Top Surface Delta**.

• Depending on the type of weld you are adding, specify the **Bottom Surface Delta** or **Radius** as required.

6   Click **Next**.

The **Weld Bead Mate Surfaces** dialog box appears. Be sure to read the information that is provided about selecting faces.

7   Click the **Contact Faces** box, then select the faces of the components where you want the weld bead to be added.

Depending on the type of weld bead you are adding, you may need to select the following:

• **Stop Faces.** Select the faces (usually four faces) where you want the weld bead to start and stop.

• **Top Faces**. Select the faces from which to measure the **Top Surface Delta**.
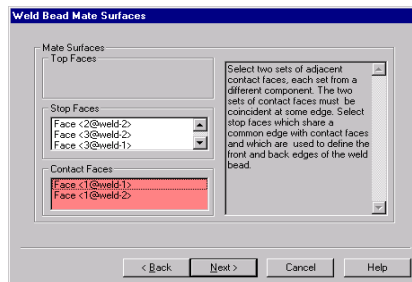
NOTE: Some **Mate Surfaces** edit boxes may be grayed out because they are not needed for the selected weld bead.

---

8   Click **Next**.

9   In the **Weld Bead Part** dialog box, accept the default name for the **Weld Bead Part** or enter a new part name, and click **Finish**.

Weld Face and Edge Selection Tips

To make face and edge selection easier, you can,

- View the assembly with hidden lines in gray.

- Place the assembly in isometric view orientation.

- Edit the distance in a distance mate to enlarge the space between the contact faces temporarily. After you select the faces and complete the creation of the weld bead, you can edit the distance mate again. The weld bead between the two components is updated to reflect the final mating distance.

- Use the **Selection Filter**.
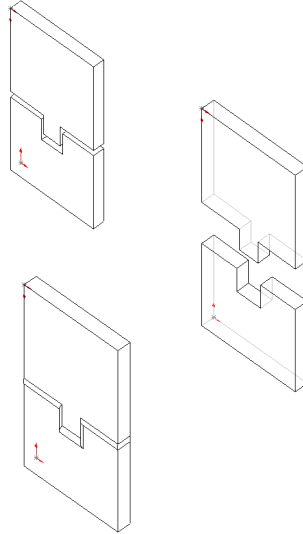
Annotations for Weld Beads

Weld symbols are automatically generated by the system when you create a weld bead.

To move the symbol into position:

1   Click where the arrow meets the weld edge.

   Green drag handles appear on the symbol and the arrow.

2   Drag the symbol by the handle and release the mouse button when it is in the position you want.
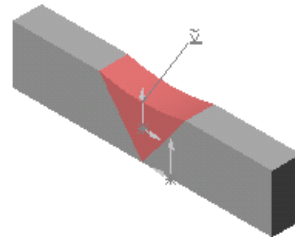
To add the weld symbol to the active assembly drawing:

1   Select a drawing view, click **Insert, Model Items**, and select **Welds** from the menu.

2   Click **OK**.

Refer to **Weld Symbol** on page 9-34 for more information.

Examples of Creating Weld Beads

Example 1: V Butt Weld

The assembly shown here calls for a V Butt type weld. The components are mated using three coincident mates:

- Between the edges where the contact faces meet
- Between the top faces
- Between the stop faces on the front of the assembly
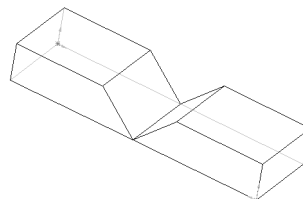
To add a V Butt weld:

1   Click **Insert, Assembly Feature, Weld Bead**.

2   In the **Weld Bead Type** dialog box, select **Single V Butt**, and click **Next**.

3   In the **Weld Bead Surface** dialog box, select a **Surface Shape**, and specify the **Top Surface Delta** value.

4   Click **Next**.

5   In the **Weld Bead Mate Surfaces** dialog box:

- Click the **Contact Faces** box and select the slanted faces of the components.
- Click the **Stop Faces** box and select the two sides of each component (four faces total).

6   Click **Next**.

7   In the **Weld Bead Part** dialog box, accept the default name for the **Weld Bead Part** or enter a new part name.

8   Click **Finish**.
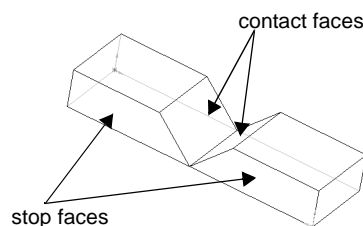
The weld bead is added between the two components.
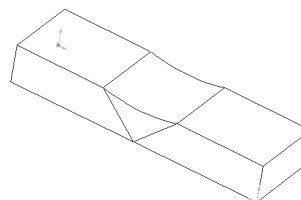
Example 2: Square Butt Weld

The assembly shown here calls for a Square Butt type weld. The components are mated as follows:

- A distance mate between the contact faces
- A coincident mate between the top faces
- A coincident mate between the stop faces
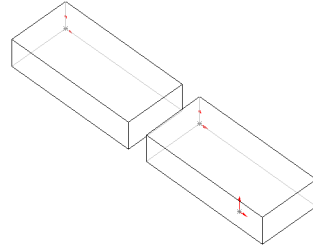
To add a square butt weld:

1 Click **Insert, Assembly Feature, Weld Bead**.

2 In the **Weld Bead** dialog box, select **Square Butt**, then click **Next**.

3 In the **Weld Bead Surface** dialog box, select a **Surface Shape**, and specify the **Top Surface Delta**.

4 Click **Next**.

5 In the **Weld Bead Mate Surfaces** dialog box, click each mate surface face type and the corresponding faces in the model.

- Click the **Top Faces** box, then select the two top faces of components (marked **A** in the illustration).

- Click the **Stop Faces** box then select the four **Stop Faces** of the components (the two faces marked **B** and the two faces opposite them).

- Click the **Contact Faces** box and select the two contact faces (the face marked **C** and the opposite face).

6 Click **Next**.

7 In the **Weld Bead Part** dialog box, name the weld bead part or accept the default name.

8 Click **Finish**.
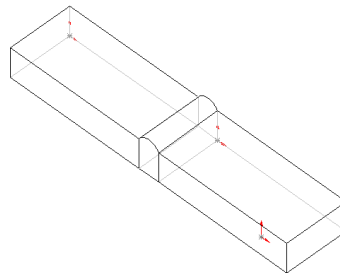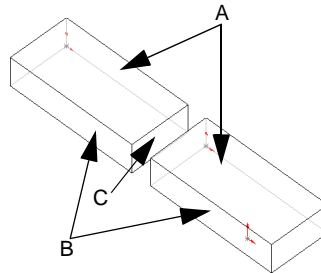
The weld bead is added between the components.

Editing a Weld

To edit a weld:

1 In the FeatureManager design tree, double-click the **Bead** component, or click the ⊞ beside its name to expand the component.

2 Right-click the **Weld Bead** feature and select **Edit Definition**.

3 Step through all the weld bead dialog boxes, changing parameters as desired. Click **Next** to continue, then click **Finish** when you reach the last dialog box.

4 Click **Rebuild** or **Edit, Rebuild**.

13

Importing and Exporting Files

This chapter describes these SolidWorks file import and export topics:

❑ Setting import/export options

❑ Exporting and importing the following file types:

- IGES
- Parasolid text and binary
- STEP
- ACIS™
- VDAFS
- STL (Export only)
- DXF/DWG
- VRML
- TIFF

❑ Recognizing features with FeatureWorks

Exporting/Importing SolidWorks Documents

You can export SolidWorks documents to a number of formats for use with other applications. You can also import files to SolidWorks using a number of formats from other applications. The following table displays the data translation methods available for SolidWorks documents:

	Parts		Assemblies		Drawings	
	Import	Export	Import	Export	Import	Export
IGES	√	√	√	√		
Parasolid	√	√	√	√		
STEP	√	√	√	√		
ACIS	√	√	√	√		
VDAFS	√	√				
STL		√		√		
DXF/DWG	√				√	√
VRML	√	√	√	√		
TIFF	*	√	*	√		√

NOTE: TIFF files can be imported to SolidWorks part and assembly documents using **Insert, Picture**. See **Exporting/Importing TIFF Files** on page 13-23.

To export a SolidWorks document with a new format:

Before you export a SolidWorks document, make sure that you have the Export options set for your purposes. Refer to **Setting Export Options** on page 13-3.

1   Click **File, Save As**.

   The **Save As** dialog box appears.

2   Select a format from the **Save as type** box.

3   Enter a name for the file.

4   Click **Save**, or click **Cancel** to exit without saving.

To import a file to SolidWorks from another application:

Before you import a document to SolidWorks, make sure that you have the Import options set for your purposes. Refer to **Setting Import Options** on page 13-4.

1   Click **File, Open**.

   The **Open** dialog box appears.

2   Select the desired format from the **Files of type** box.

3   Browse to the desired file, and click **Open**.

Setting Export Options

Before exporting a file make sure that the **Options** settings are appropriate for the target application. Options are available for the following file types:

IGES	**Parasolid**	**DXF/DWG**	**VRML**
STL	**ACIS**	**TIFF**	

To set export options:

1 Click **Tools, Options**.

2 Click the **Export** tab.

3 Choose options as desired, and click **OK**.
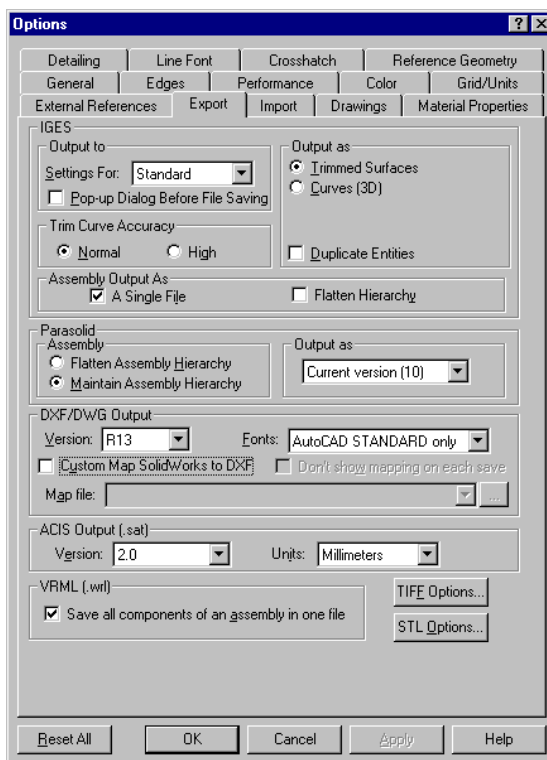
For details about these options, refer to the sections later in this chapter or Appendix A, "Export Options."

Setting Import Options

Before you import an IGES, STEP, SAT or VDAFS file, make sure the **Options** are set the way you want. The options apply only to these surface-based translators.

To set import options:

1   Click **Tools, Options.**

2   Click the **Import** tab.

3   **Error resolution.** From the pulldown lists next to the error types, select the default method that you want SolidWorks to use to resolve problems.

- The solutions for **Surface Errors**, **Surface Flaws**, and **Significant Digits** (IGES import only) are:

    Prompt (ask for a solution each time a problem occurs)
    Try to knit into solid
    Try to form reference surfaces
    Cancel operation

- The solutions for **Blanked Surfaces** (IGES import only) are:
    Prompt
    Ignore blanked surfaces
    Do NOT ignore blanked surfaces
    Cancel operation

NOTE: **Surface Errors** occur when a surface cannot be imported.

Surface Flaws occur when a surface can be imported but has some sort of defect.

Significant Digits problems occur when the data contains low precision data.

Blanked Surfaces are invisible surfaces included in a file, typically, for reference purposes.

---

4   **Try forming solid model(s) from surfaces**. If you do <u>not</u> check this option, the surfaces are automatically grouped into one or more reference surfaces (named **Surface 1, 2, ...**).

5   After you select the default responses, click **OK**.

IGES Files

The following IGES entity types are supported for import and export:

IGES Entity Type	Entity Name
416	External reference entity
408	Singular subfigure instance entity
406 Form 12	External reference file
402 Form 7	Group entity
314	Color definition entity
308	Subfigure definition entity
144	Trimmed (parametric) surface
143 Import only	Bounded surface entity
142	Curve on a parametric surface
140 Import only	Offset surface entity
128	Rational B-spline surface
126	Rational B-spline curve
124	Transformation matrix entity
122	Tabulated cylinder
120	Surface of revolution
118 Import only	Ruled surface
114 Import only	Parametric spline surface
112	Parametric spline curve
110	Line
106 Form 12 Import only	Copious Data - 3D piecewise linear curve
106 Form 11 Import only	Copious Data - 2D piecewise linear curve
104 Import only	Conic arc
102	Composite curve
100	Circular arc

If you want the SolidWorks software to try to form a solid from the IGES data for use in SolidWorks, export the solid or surfaces from the source system as trimmed surfaces (Entity type 144) or as untrimmed surfaces (Entity types 128, 122, 120, 118, 114).

Also, set the number of significant digits (precision) as high as possible to ensure high-fidelity translation.

Exporting IGES Files

You can export SolidWorks part and assembly documents to IGES format. Parts and surfaces exported to IGES format retain their part/surface color and display in color when in **Shaded** mode.

To specify the default export options for IGES files:

1   Click **Tools, Options**, and select the **Export** tab.

2   In the IGES area of the dialog box, select from the following options, then click **OK** to set the default format.

IGES Options

Output to

If you choose to output as **Trimmed Surfaces**, the following options are available:

- **Settings For:** Select the mechanical design application program to which you want to export your SolidWorks files.

- **Pop-up Dialog Box Before File Saving.** Check this option if you want the program to display the **Settings For** list every time you save an IGES file so you can select a different format type. You do not have to change the **Export** options page each time you export an IGES file to a new format.

Output as

- **Trimmed Surfaces.** The faces of the part, assembly, or the selected surfaces, are converted to trimmed surfaces (Entity Type 144) in the IGES file.

  The IGES entity types that compose the trimmed surfaces depend on the export format chosen. The following table shows the IGES entity types that compose the trimmed surfaces.

Export Format	Exported IGES Entity Types
Standard	144, 142, 128, 126, 122, 120, 110, 102, 100
ANSYS	144, 142, 128, 126, 110, 102, 100
COSMOS	144, 142, 128, 126, 110, 102, 100
MasterCAM	144, 142, 128, 126, 110, 102, 100
SURFCAM	144, 142, 128, 126, 110, 102, 100
SmartCAM	144, 142, 128, 126, 110, 102, 100
TEKSOFT	144, 142, 128, 126, 110, 102, 100
AlphaCAM	144, 142, 128, 126, 110, 102, 100

NOTE: If the system you are exporting to is not listed, refer to the manuals of that system to see which entities are supported and choose an appropriate setting.

- **Curves (3D).** The solid body is converted to a 3D wireframe representation in the IGES file. Unlike surfaces or faces, you cannot export individually selected model edges to an IGES file.

  Select either **B-Splines** or **Parametric Splines** depending on the entity types required by the target system. The following table shows which IGES entities are exported for each type of 3D curve.

Type of 3D Curve	Exported IGES Entity Types
B-splines (entity 126)	126, 110, 102[*], 100
Parametric splines (entity 112)	112, 110, 102[*], 100

  [*]Exported only if you select the **Duplicate Entities** option.

- **Duplicate Entities.** Check this option if you want to export composite curves (entity type 102) to any of the available export formats. Because each edge is included once in each of two composite curves forming the boundaries of the faces that meet at each edge, the curves are duplicated. For each pair of duplicate curves, one curve has the opposite sense (direction) from the other.

❑ **Trim Curve Accuracy**. Select **Normal** or **High**. Adjust this setting if you experience problems importing into another system.

- **Normal.** Select this option when smaller file size and faster export performance are preferred and high curve accuracy is not needed.

- **High.** Select this option when high curve accuracy is essential; the file size is larger than when using the **Normal** setting. This setting affects both files exported with **Trimmed Surfaces** and files exported with **Curves (3D)**. High curve accuracy can sometimes help if the target system has trouble importing the IGES file, or cannot knit the surfaces into a useful solid.

❑ **Assembly Output As.**

- **A Single File**. When checked, all assembly components, sub-assemblies, and sub-assembly components are saved in one file. Otherwise, the assembly components and the sub-assembly components are saved as individual IGES files in the same directory.

- **Flatten Hierarchy**. When checked, the assembly is flattened to one level of part bodies (no sub-assemblies). Components of sub-assemblies are at the same level as top-level components.

To export a SolidWorks part document or selected set of faces or surfaces to an IGES file:

1 Select the faces or surfaces you want to export. If no faces or surfaces are selected, the entire solid is exported.

2 Click **File**, **Save As**.

3 Select **IGES Files (*.igs)** in the **Save as type** box.

4 Enter a name for the file. SolidWorks automatically adds the **.igs** extension.

5 Select a coordinate system in the **Output coordinate system** box to apply a transformation matrix to the geometry, if desired. See **Creating a Coordinate System** on page 3-6.

6 Click **Save**.

7 If you selected any faces or surfaces, indicate whether you want to export the **Selected Surface(s), Selected Face(s)**, or the **Solid Body**, then click **OK**.

To export a SolidWorks assembly document or selected set of components to an IGES file:

1 Select the components from the FeatureManager tree that you want to export. If no components are selected, the entire assembly is exported.

2 Click **File, Save As**.

3 Select **IGES files (*.igs)** in the **Save as type** box.

4 Enter a name for the file. SolidWorks automatically adds the .**igs** extension.

5 Select a coordinate system in the **Output coordinate system** box to apply a transformation matrix to the geometry, if desired. See **Creating a Coordinate System** on page 3-6.

6 Click **Save**.

7 If the assembly document has lightweight components, you can choose to resolve the components to save them, or to omit the components from the exported file. See **Lightweight Parts** on page 6-31 for more information.

Importing IGES Files

You can import surfaces from IGES files into SolidWorks and use them to:

- Create a base feature from a group of IGES surfaces that form a closed volume
- Trim a part with an imported surface
- Create a base, boss, or cut feature by thickening an imported surface

You can import 3D curves from IGES files and use them to:

- Create planes used in pierce point relations in sketches
- Create path and guide curves for sweeps and lofts

Using Imported IGES Files

To import surfaces or to import 3D wireframe curves from an IGES file:

1 Click **File**, **Open**.

2 Select **IGES Files (*.igs)** in the **Files of type** box.

3 Select an IGES file.

4 Click **Open**.

If there are curves in the IGES file or if there are different levels (or layers) in the IGES file, the **IGES-In Surfaces, Curves, and Levels** dialog box appears. Levels are sometimes used by other systems to separate assembly components or to separate IGES entity types.

5 Edit the **IGES-In Surfaces, Curves and Levels** dialog box as needed.

- Select the types of entities to include: **Include Surfaces** and/or **Include Curves**.

- Select the levels to include:

    Include All Levels. All levels are imported to level zero.

    Include Levels. Only the specified levels are imported to level zero.

6 Click **OK** to exit the **IGES-In Surfaces, Curves and Levels** dialog box or click **Cancel** to abort the import process.

If there is wireframe geometry in the file, the SolidWorks software reads the IGES data and forms curves for the IGES wireframe entities. The 2D geometry on the XY plane is imported into a sketch rather than imported as reference curves.

If there are surfaces in the file, they are read into SolidWorks.

- If the attempt to knit the surfaces into a solid succeeds, the solid appears as the base feature (named **Imported1**) in a new part file. You can *add* features (bosses, cuts, etc.) to this base feature, but you cannot *edit* the base feature itself.

- If the SolidWorks software finds that the surfaces in the IGES file represent multiple closed volumes, then one part is made for each closed volume. An assembly file also is made that includes the imported parts positioned relative to the assembly origin, according to how the surfaces are defined in the IGES file.

- If the attempt to knit the surfaces fails, the IGES surfaces are grouped into one or more reference surfaces (named **Surface1, 2, ...**) in a new part file.

If any errors are encountered or if unsupported entities are present in the IGES file, error and report files (**.err** and **.rpt**) are written to the same directory as the IGES file you imported.

To import a surface from an IGES file:

1   With a part file open, click **Insert, Reference Geometry, Imported Surface.**

2   In the **Open** dialog box, select the IGES file to import and click **Open**.

The surface(s) is imported, and a **Surface** feature is added to the part. The surface is positioned relative to the part origin, using the global coordinates in the IGES file.

> **NOTE:**   Wireframe geometry and curves are ignored when importing surfaces as reference geometry.

To edit a feature created from an IGES file:

You can replace an imported IGES body or surface.

1   Right-click the feature created from the IGES file, and select **Edit Definition**.

2   In the **Open** dialog box, browse to another IGES file, and click **Open**.

The original imported body is replaced only if the data in the new IGES file can be successfully knitted into a body.

If you are editing an imported reference surface, the selected surface is replaced by the first reference surface in the IGES file, and all other reference surfaces in the file are added to the model.

For information about using imported IGES surfaces to create solid model features, see **Using Surfaces to Create Features** on page 3-20.

Exporting/Importing Parasolid Files

You can export SolidWorks part or assembly documents to Parasolid text files or Parasolid binary files. SolidWorks 98Plus supports Parasolid version 10.0 and earlier versions for import.

Data exported to or imported from Parasolid format retains its color when displayed in **Shaded** mode. Component names in assemblies are supported for both import and export. SolidWorks does not support curve or point data for export or import.

To specify the default export options for **Parasolid** *files:*

1 Click **Tools, Options**, and select the **Export** tab.

2 In the **Parasolid** section of the dialog box, select from the following options, then click **OK.**

Parasolid Options

Assembly

- **Flatten Assembly Hierarchy.** The default setting is to flatten the assembly to one level of only part bodies. A flattened file contains a top-level assembly and a series of parts that contain imported features.

- **Maintain Assembly Hierarchy.** You can select this setting to maintain the assembly hierarchy of assemblies, sub-assemblies, and parts. The structure mirrors the original assembly, its nested subassemblies, and parts.

Output as

From the pulldown list, select the type supported by the target system.

To export a SolidWorks part or assembly document to a **Parasolid** *file:*

1 Click **File**, **Save As**.

2 Select **Parasolid Files (*.x_t)** for a text file or **Parasolid Binary Files (*.x_b)** for a binary file in the **Save as type** box.

   Binary files are smaller than text files, but binary files are not supported in some target systems. Select the type supported by the target system.

3 Enter a name for the file. SolidWorks automatically adds the **.x_t** or **.x_b** extension.

4 Click **Save**.

5 If saving an assembly document with lightweight components, you can choose to resolve the components to save them, or to omit the components from the exported file. See **Lightweight Parts** on page 6-31 for more information.

To import a* Parasolid *file:

1 Click **File, Open.**

2 In the **Open** dialog box, set **Files of type** to **Parasolid Files (*.x_t)** or **Parasolid Binary Files (*.x_b)**.

NOTE: Some other systems use the **.xmt_txt** extension for Parasolid text files and the **.xmt_bin** extension for Parasolid binary files. Rename the extension to **.x_t** or **.x_b** for import to SolidWorks.

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3 Browse to the desired file, and click **Open**.

The selected file opens.

Exporting/Importing STEP Files

You can export or import SolidWorks part or assembly documents to STEP files. SolidWorks supports two STEP Application Protocols, AP203 and AP214.

❑ AP203 is supported for both import and export.

• Conformance Classes 1, 2 and 6 are fully supported.

• Conformance Classes 4 and 5 are partially supported.

❑ AP214 is supported for import only. Conformance Classes 1 and 2 are partially supported.

To export a SolidWorks part document or selected set of faces or surfaces to a STEP file:

1 Select the faces or surfaces you want to export. If no faces or surfaces are selected, the entire solid is exported.

2 Click **File, Save As**.

3 Select **STEP AP203 (*.step)** in the **Save as type** box.

4 Enter a name for the file. SolidWorks automatically adds the **.step** extension.

5 Select a coordinate system in the **Output coordinate system** box to apply a transformation matrix to the geometry, if desired. See **Creating a Coordinate System** on page 3-6.

6 Click **Save.**

7 If you selected any faces or surfaces, indicate whether you want to export the **Selected Face(s), Selected Surface(s),** or the **Solid Body,** then click **OK**.

To export a SolidWorks assembly document to a STEP file:

1 In an assembly document, click **File, Save As**.

2 In the **Save As** dialog box, scroll the **Save as type** box to select **STEP AP203 (*.step)**.

3 Enter a name for the file. SolidWorks automatically adds the .**step** extension.

4 Select a coordinate system in the **Output coordinate system** box to apply a transformation matrix to the geometry, if desired. See **Creating a Coordinate System** on page 3-6.

5 Click the **Save** button.

6 If the assembly document has lightweight components, you can choose to resolve the components to save them, or to omit the components from the exported file. See **Lightweight Parts** on page 6-31 for more information.

To import a STEP file:

1 Click **File, Open**.

2 In the **Open** dialog box, set **Files of type** to **STEP AP203/214 (*.step)**.

3 Browse to the desired file, and click **Open.**

The selected file opens.

If any errors are encountered, an error file (**.err**) is written to the same directory as the STEP file you imported.

Exporting/Importing ACIS Files

You can export or import SolidWorks part or assembly documents to ACIS files.

To specify the default export options for **ACIS** *files***:**

1 Click **Tools, Options**, and select the **Export** tab.

2 In the **ACIS Output (.sat)** area of the dialog box, select from the following options, then click **OK**.

ACIS Options

Version

From the pulldown list, select the type supported by the target system.

Units

From the pulldown list, select the default units of measure to use for ACIS files: **Millimeters**, **Centimeters**, **Meters**, **Inches**, or **Feet**.

To export a SolidWorks part document or selected set of faces or surfaces to an* ACIS *file:

1   Select the faces or surfaces you want to export. If no faces or surfaces are selected, the entire solid is exported.

2   Click **File**, **Save As**.

3   Select **ACIS Files (*.sat)** in the **Save as type** box.

4   Enter a name for the file. SolidWorks automatically adds the **.sat** extension.

5   Select a coordinate system in the **Output coordinate system** box to apply a transformation matrix to the geometry, if desired. See **Creating a Coordinate System** on page 3-6.

6   Click **Save**.

7   If you selected any faces or surfaces, indicate whether you want to export the **Selected Face(s), Selected Surface(s),** or the **Solid Body,** then click **OK**.

To export a SolidWorks assembly document to an* ACIS *file:

1   In an assembly document, click **File, Save As**.

2   In the **Save As** dialog box, scroll the **Save as type** box to select **ACIS Files (*.sat)**.

3   Enter a name for the file. SolidWorks automatically adds the .**sat** extension.

4   Select a coordinate system in the **Output coordinate system** box to apply a transformation matrix to the geometry, if desired. See **Creating a Coordinate System** on page 3-6.

5   Click the **Save** button.

6   If the assembly document has lightweight components, you can choose to resolve the components to save them, or to omit the components from the exported file. See **Lightweight Parts** on page 6-31 for more information.

To import an* ACIS *file:

1   Click **File, Open.**

2   In the **Open** dialog box, set **Files of type** to **ACIS Files (*.sat)**.

3   Browse to the desired file, and click **Open**.

   If the units of measure are not explicitly specified in the file, a dialog box gives you the opportunity to specify the units now. Files created in older versions of the ACIS modeler do <u>not</u> contain information about the units of measure.

   The selected file opens.

   If any errors are encountered, an error file (**.err**) is written to the same directory as the ACIS file you imported.

To edit a feature created from an ACIS file:

You can replace an imported ACIS body or surface.

1 Right-click the feature created from the ACIS file, and select **Edit Definition**.

2 In the **Open** dialog box, browse to another ACIS file, and click **Open**.

The original imported body is replaced only if the data in the new ACIS file can be successfully knitted into a body.

Exporting/Importing VDAFS Files

VDAFS is a neutral file format for the exchange of surface geometry. You can export or import SolidWorks part documents to VDAFS files.

To export a SolidWorks part document, or selected set of faces or surfaces, to a VDAFS file:

1 Select the faces or surfaces you want to export. If no faces or surfaces are selected, the entire solid is exported.

2 Click **File, Save As**.

3 Select **VDAFS (*.vda)** in the **Save as type** box.

4 Enter a name for the file. SolidWorks automatically adds the **.vda** extension.

5 Select a coordinate system in the **Output coordinate system** box to apply a transformation matrix to the geometry, if desired. See **Creating a Coordinate System** on page 3-6.

6 Click **Save.**

7 If you selected any faces or surfaces, indicate whether you want to export the **Selected Face(s), Selected Surface(s),** or the **Solid Body,** then click **OK**.

To import a VDAFS file:

1 Click **File, Open**.

2 In the **Open** dialog box, set **Files of type** to **VDAFS (*.vda)**.

3 Browse to the desired file, and click **Open**.

The selected file is opened.

If any errors are encountered, an error file (**.err**) is written to the same directory as the VDAFS file you imported.

Exporting STL Files

You can export SolidWorks part and assembly documents to STL format. STL format is intended for transfer to rapid prototyping machines. SolidWorks cannot import STL files.

To specify the default export options for STL files:

1 Click **Tools, Options**, and select the **Export** tab.

2 Click the **STL Options** button.

3 Select from the following options, then click **OK**.

4 Click **OK** to exit the **Options** dialog box.

STL Options

STL Output Format

- Choose **Binary** or **Ascii**. Binary files are smaller than ASCII files, but they are not supported in some other systems. Select the type supported by the target system.

- **Do not translate STL output data to positive space.** Check this box if you want exported parts to maintain their original position in global space, relative to the origin.

- **Save all components of an assembly in one file.** If you do <u>not</u> check this option, each component is written out relative to the assembly origin and saved in an individual file.

NOTE: Export of assemblies with coincident or interfering bodies produces a file that may be unsuitable for some rapid prototyping systems. It is highly recommended that you check for interference between the components prior to saving the assembly document. See **Detecting Interference Between Components** on page 6-29 for more information.

Quality

Coarse and **Fine** are preset values. Choose **Custom** if you want to set the resolution. If you choose **Custom**:

- Adjust the **Total Quality** slider to set the **Deviation**. **Deviation** controls whole-part tessellation. Lower numbers generate files with greater whole-part accuracy.

- Adjust the **Detail Quality** slider to set the **Angle Tolerance**. **Angle Tolerance** controls smaller detail tessellation. Lower numbers generate files with greater small-detail accuracy.

As you adjust the two sliders, note that the corresponding concentric circles adjust accordingly. The circles show, approximately, how the tessellation varies as a result of the settings. (See the **Preview** option.)

NOTE: Files generated with higher quality settings are larger in size and slower to generate. Experiment with these **STL Quality** settings to determine the best settings for your own equipment.

Preview. If you selected **Custom** resolution, check this option to preview a faceted version of the model while you are adjusting the sliders (move the dialog box out of your way to view the faceted model).

Show STL Info before saving. Choose this option to see a faceted display of the model and an information box each time you save an STL file. The information box reports the file size, number of triangles, and the path and name of the file. Click **Yes** to save the file in the indicated location; click **No** to cancel the operation.

To export a SolidWorks part document or assembly document to an STL file:

1 Click **File**, **Save As**.

2 Select **STL Files (*.stl)** in the **Save as type** box.

3 Enter a name for the file. SolidWorks automatically adds the **.stl** extension.

4 Select a coordinate system in the **Output coordinate system** box to apply a transformation matrix to the geometry, if desired. See **Creating a Coordinate System** on page 3-6.

5 Click **Save**.

6 If saving an assembly document with lightweight components, you can choose to resolve the components to save them, or to omit the components from the exported file. See **Lightweight Parts** on page 6-31 for more information.

Exporting/Importing DXF/DWG Files

Only drawing documents can be saved to **.DXF** or **.DWG** files.

To specify the default export options for DXF/DWG files:

1   Click **Tools, Options**, and select the **Export** tab.

2   In the **DXF/DWG Output** area of the dialog box, select from the following options, then click **OK.**

DXF/DWG Options

Version

From the pulldown list, select the type supported by the target system.

Fonts

Select either **Truetype Fonts** or **AutoCAD STANDARD only**.

Custom map SolidWorks to DXF

Click the checkbox to implement DXF mapping. See **DXF/DWG Mapping** on page 13-19.

Don't show mapping on each save

Select this option to prevent the **SolidWorks to DXF/DWG mapping** dialog box from appearing every time you save a drawing with **Custom map SolidWorks to DXF** selected.

Map file

To use a previously saved map file, enter the file name or browse to the desired file.

To export a SolidWorks drawing document to a DXF/DWG file:

1   Click **File**, **Save As**.

2   Select **DXF Files (*.dxf)** or **Dwg Files (*.dwg)** in the **Save as type** box.

3   Enter a name for the file. SolidWorks automatically adds the **.dxf** or **.dwg** extension.

4   Click **Save**.

5   If appropriate, the **SolidWorks to DXF/DWG mapping** dialog box appears.

To import a DXF/DWG file:

1   Click **File, Open.**

2   In the **Open** dialog box set **Files of type** to list **DXF files (*.dxf)** or **Dwg files (*.dwg)**.

3   Browse to the desired file, and click **Open**.

The **Open DXF/DWG File** dialog box appears.

You have the option of importing to either a drawing or a part.

4   If you choose **Import to Drawing**, adjust the drawing sheet size and units of measure, if necessary.

-   Click **Move entities onto sheet** to move the lower left-hand corner of the imported drawing to the origin of the SolidWorks drawing sheet.

-   Click **Import to template** to import the file as a SolidWorks template.

– or –

If you choose **Import to Part**, adjust the units of measure if necessary.

5   Click **OK**.

The selected file opens on a drawing sheet or in a part, as specified.

If the file opens to a part, it forms 3D curves.

DXF/DWG Mapping

You can map selected SolidWorks colors, line styles, or entities to different layers or colors in a DXF/DWG file. You create map files in SolidWorks to assign selected items to designated layers or colors in the DXF/DWG file.

To create a map file:

Map files are created using **Layer Mapping**, **Color Mapping,** or **Entity Mapping**.

1   Make sure that **Custom map SolidWorks to DXF** is checked and **Don't show mapping on each save** is deselected in the **DXF/DWG Output** area of the **Tools, Options, Export** dialog box.

2   With a SolidWorks drawing document open, click **File, Save As**.

3   Select **DXF files (*.dxf)** or **Dwg files (*.dwg)** in the **Save as type** box.

4   Click **Save**

The **SolidWorks to DXF/DWG mapping** dialog box appears.

5   Select the **Layer Mapping**, **Color Mapping,** or **Entity Mapping** tabs to create the map file. What you select depends upon the data to be mapped.

To map colors and line styles to layers:

Layer Mapping allows you to map a combination of colors and line styles to selected layers. **Layer Mapping** is intended to work independently of **Color Mapping** or **Entity Mapping**.

1 Click **Map File Settings** to use a previously saved map file.

   The settings from the previously saved map file are displayed.

2 To create a new layer, click the **New Layer** box and enter a name of up to 31 characters. Spaces are not allowed in a layer name.

   By default, the first layer is named 0 (zero).

3 Select an item in the **Color** column or select **Any** to map line styles to a selected layer, regardless of color.

4 Select an item in the **Line Styles** column or select **Any** to map colored entities to a selected layer, regardless of line style.

5 Click the layer name to which you want to assign the selected item and line style pair.

   The resulting set of *color*, *line style* and *layer* is displayed below the selection windows.

6 Repeat for each combination of entities that you want to map.

7 When you are done, click **Map File Settings** to save the map file for future use.

8 Click **OK** to accept the changes and save the drawing file, or click **Cancel** to cancel the save operation, or click **Reset All** to return to the default map settings.

To map SolidWorks colors to DXF standard colors:

Color Mapping allows you to map the colors used in SolidWorks drawings with the standard DXF color numbers. **Color Mapping** can either work on its own or with **Entity Mapping**. It is not intended to work with **Layer Mapping.**

1 Click **Map File Settings** to use a previously saved map file.

   The settings from the previously saved map file are displayed.

2 Click an item in the **SolidWorksColors** column.

3 Click the **DXFColors** item to which you want to assign the SolidWorks color.

   The resulting set of *SolidWorksColor* and *DXFColor* is displayed below the selection windows.

4 Repeat for each SolidWorks color that you want to map.

5 When you are done, click **Map File Settings** to save the map file for future use.

6 Click the **Entity Mapping** tab to include entity mapping in your map file.

7 Click **OK** to accept the changes and save the drawing file, or click **Cancel** to cancel the save operation, or click **Reset All** to return to the default map settings.

To map drawing entities to layers:

Entity Mapping allows you to map drawing entities to selected layers. **Entity Mapping** can either work on its own or with **Color Mapping**. It is not intended to work with **Layer Mapping.**

1   Click **Map File Settings** to use a previously saved map file.

The settings from the previously saved map file are displayed.

2   To create a new layer, click the **New Layer** box and enter a name of up to 31 characters. Spaces are not allowed in a layer name.

By default, the first layer is named 0 (zero).

3   Select an item in the **Entities** column.

4   Click the layer name to which you want to assign the selected entity.

The resulting set of *entity* and *layer* is displayed below the selection windows.

5   Repeat for each entity that you want to map.

6   When you are done, click **Map File Settings** to save the map file for future use.

7   Click the **Color Mapping** tab to include color mapping in your file.

8   Click **OK** to accept the changes and save the drawing file, or click **Cancel** to cancel the save operation, or click **Reset All** to return to the default map settings.

To use a previously saved map file:

1   Click **Map File Settings** from the **SolidWorks to DXF/DWG mapping** dialog box.

2   Make sure that **Load map file** is selected.

3   Click the **Browse** button and browse to an existing file.

4   Click **OK**.

> **NOTE:** Previously saved map files can contain information for the **Layer Mapping, Color Mapping,** and **Entity Mapping** tabs. Review all tabs prior to saving the SolidWorks drawing.

To save a map file:

1   Click **Map File Settings** from the **SolidWorks to DXF/DWG mapping** dialog box.

2   Make sure that **Save map file** is selected.

3   Enter a path and name for a new map file in the edit box, or click the **Browse** button and browse to an existing file.

4   Click **OK**.

Exporting/Importing VRML Files

VRML files can be used to display 3D graphics over the Internet.

To specify the default export options for VRML files:

1   Click **Tools, Options**, and select the **Export** tab.

2   In the **VRML (*.wrl)** area of the dialog box, check **Save all components of an assembly in one file** to include each assembly and sub-assembly component in one file.

3   Click **OK** to exit the dialog box.

To export a SolidWorks part or assembly document to a VRML file:

1   Click **File**, **Save As**.

2   Select **VRML Files (*.wrl)** in the **Save as type** box.

3   Enter a name for the file. SolidWorks automatically adds the **.wrl** extension.

4   Click **Save**.

5   If saving an assembly document with lightweight components, you can choose to resolve the components to save them, or to omit the components from the exported file. See **Lightweight Parts** on page 6-31 for more information.

> **NOTE:**   When you use **Save As** to a **VRML file (.wrl)**, the software takes the **View Section** option into account. If **View Section** is on, only the visible geometry is output to the **.wrl** file. Appropriately, this is different from the way the software behaves when you save to an IGES, STEP, Parasolid, or ACIS file.

To import a VRML file:

1   Click **File, Open.**

2   In the **Open** dialog box, set **Files of type** to **VRML Files (*.wrl)**.

3   Browse to the desired file, and click **Open**.

The selected file opens.

> **NOTE:**   The import of VRML files attempts to knit the VRML polygons into a solid. There are no contoured surfaces in the imported file; all surfaces are planar. This import option is intended as a last resort for importing data into SolidWorks. In many cases (especially complex models) the data does not knit into a usable solid, and is useful for viewing purposes only.

To edit a feature created from an VRML file:

You can replace an imported VRML body or surface.

1 Right-click the feature created from the VRML file, and select **Edit Definition**.

2 In the **Open** dialog box, browse to another VRML file, and click **Open**.

The original imported body is replaced only if the data in the new VRML file can be successfully knitted into a body.

Exporting/Importing TIFF Files

You can save any SolidWorks document as a TIFF image.

You can import a TIFF image and use it as a background of a part or assembly. You can capture the image as it appears on a normal view (screen) or you can capture the image as it appears on a print preview. For drawing documents, you can select the sheet size and scale factor.

To specify the default TIFF image format for export:

1 Click **Tools, Options**, and select the **Export** tab.

2 Click the **TIFF options** button.

3 Select from the following options, then click **OK**.

4 Click **OK** to exit the **Options** dialog box.

TIFF Options

Image Type

From the pulldown menu, select either **RGB Full Color** or **Black & White** image.

Compression Scheme

Select the type supported by the target system.

- **Uncompressed**.

- **Packbit**. Results in the smallest file size for a color image.

- **Group 4 FAX**. Results in the smallest file size of the three formats, but can only be used for a black and white image.

Capture Type

Select either **Screen capture** or **Print capture**.

- **Screen capture**, for all SolidWorks documents, captures the image at the resolution of the screen display.

- **Print capture**, for parts and assemblies, captures the image at the resolution of a print preview.

- **Print capture**, for drawings, captures the image at a user-defined **DPI** (dots per inch) and **Paper size**.

To export a SolidWorks document to a TIFF file:

1 Click **File, Save As**.

2 Select **TIFF files (*.tif)** in the **Save as type** box.

3 Enter a name for the file. SolidWorks automatically adds the **.tif** extension.

4 Click **Save.**

5 If saving an assembly document with lightweight components, you can choose to resolve the components to save them, or to omit the components from the exported file. See **Lightweight Parts** on page 6-31 for more information.

To import a TIFF file:

1 In a part or assembly document, click **Insert, Picture**.

2 In the **Open** dialog box, browse to the desired file, and click **Open**.

The selected file opens as a background image of your part or assembly.

You can import **RGB Full Color** and **Black & White** images and you can also import **Grayscale** and **Palette** images from other systems.

To turn the display of the TIFF image on or off:

Click **View, Display, Picture**.

When the **Picture** option is checked, the image is displayed; when the **Picture** option is not checked, the image is hidden.

To delete a TIFF image from a document:

Click **View, Modify, Picture, Delete**.

To replace one TIFF image with another:

1 Click **View, Modify, Picture, Replace**.

2 In the **Open** dialog, browse to the TIFF file that you want to use as a replacement and click **Open**.

FeatureWorks

FeatureWorks™ recognizes a limited set of features on an imported solid in a SolidWorks part document. These features can be modified similar to any other SolidWorks features. FeatureWorks is primarily intended for machined parts and sheet metal parts similar to these illustrated models:
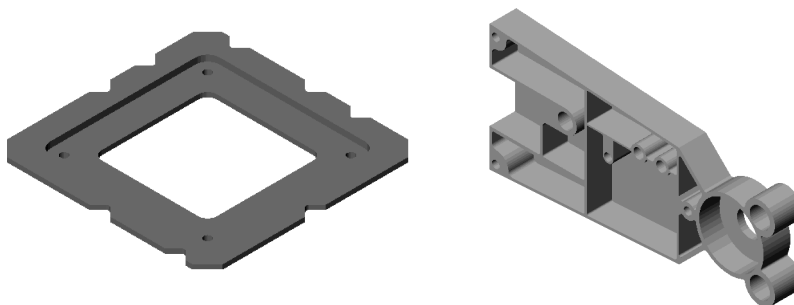
There are some things to note about FeatureWorks.

❑ **FeatureWorks is fully integrated with SolidWorks**. FeatureWorks is supplied as a SolidWorks dynamic link library (.DLL) add-in. You access all the controls for the FeatureWorks interface from the FeatureWorks item on the main SolidWorks menu bar. This menu bar is displayed whenever a SolidWorks part is open.

❑ **FeatureWorks recognizes the following features**:

- *Extrusion features* where the underlying sketch geometry is made up of the following sketch segments: line, circle, and circular arc.

- *Revolved features* that are conical or cylindrical shapes. These features are recognized as extrusions.

- *Chamfers* on linear edges

- *Constant radius fillets* on linear and circular edges

❑ **Features are recognized on imported parts**. Models imported from IGES, STEP, Parasolid (**.x_t, .x_b**), SAT, and VDAFS can have features recognized. Feature recognition is only performed if **Imported1**, the imported solid body, is the only feature in the FeatureManager design tree.

❑ **Features recognized by FeatureWorks are fully editable**. These features behave the same as other SolidWorks features, which can be moved, renamed or edited. You can edit the definition of recognized features to change their parameters. For features that are based on sketches, you can edit the sketches to change the geometry of the features.

> **NOTE:** During the recognition process, recognized features are removed from the graphics display. Only the features that have yet to be recognized are displayed. Once the recognition process is complete, the entire, original geometry is displayed.

To turn on FeatureWorks:

1   Click **Tools, Add-Ins**

2   Select **FeatureWorks**, and click **OK**.

FeatureWorks Options

To specify default options for Feature Recognition:

1   Click **FeatureWorks, Options**.

2   Select from the following options, then click OK.

Feature recognition

- **Basic features**. When checked, FeatureWorks includes basic features, extrusions and revolves, during **Automatic Feature Recognition**.

- **Chamfers and fillets**. When checked, FeatureWorks includes chamfers and fillets during **Automatic Feature Recognition**.

Part document

- **Overwrite existing part**. FeatureWorks creates the new features in the existing part document. The original imported body is replaced.

- **Create new part**. FeatureWorks creates the new features in a new part document.

Feature Recognition

FeatureWorks recognizes features on an imported solid body in a SolidWorks part document. As features are extracted from the imported solid body, the features are listed in the FeatureManager design tree and are fully editable within SolidWorks. **Imported1** changes in both the FeatureManager design tree and the graphics window to include only the geometry that *cannot* be recognized by FeatureWorks. If all of the imported geometry is recognized by FeatureWorks, **Imported1** is replaced in the FeatureManager design tree and the graphics window with fully editable features.

FeatureWorks has two feature recognition modes, automatic and interactive. You can use them individually or in combination.

NOTE: The SolidWorks option **Use fully defined sketches** must be deselected to **Recognize Features**. Click **Tools, Options, General,** and deselect **Use fully defined sketches,** if necessary.

To recognize features automatically:

1 Import a solid body to a part document using IGES, Parasolid (**.x_t, .x_b**), SAT, STEP, or VDAFS. **Imported1** must be the only feature in the FeatureManager design tree.

2 Click **FeatureWorks, Automatic Feature Recognition**.

3 Click **FeatureWorks, Recognize Features**.

FeatureWorks attempts to automatically recognize features. FeatureWorks recognizes and highlights as many basic features, chamfers, and fillets, as possible.

If FeatureWorks automatically recognizes all of the features in the imported solid body, the model is rebuilt and **Imported1** is replaced by the newly created features in the FeatureManager design tree.

To recognize features interactively:

1 Import a solid body to a part document using IGES, Parasolid (**.x_t, .x_b**), SAT, STEP, or VDAFS. **Imported1** must be the only feature in the FeatureManager design tree.

2 Click **FeatureWorks, Interactive Feature Recognition**.

3 Click **FeatureWorks, Recognize Features**.

4 In the **Recognize Features - Interactive** dialog box, select from these options:

- **Feature type**. Select the feature type to recognize, either **Boss extrude** or **Cut extrude**.

- **Selected entities**. Select either **Face** or **Edge**.
  - **Face**. Select a model face representing the sketch for the feature.
  - **Edge**. Select a set of edges representing the sketch for the feature.

    Click **Clear All Selection** to clear the **Selected Entities**.

- Click **Up to face**, if appropriate. Select a face for termination of the feature. FeatureWorks extends the feature from the sketch plane to the selected face.

5 Click **Recognize Feature** to interactively recognize the selected feature.

If successful, the feature is removed from the imported body, and a new feature is added in the FeatureManager design tree.

6 Click **Undo** to undo the newly created feature. Click **Redo** to recreate the undone feature.

7 Click **Finish** to return to the SolidWorks window, or click **Cancel** to return to the original imported solid body.

To recognize features both automatically and interactively:

1 Import a solid body to a part document using IGES, Parasolid (**.x_t, .x_b**), SAT, STEP, or VDAFS. **Imported1** must be the only feature in the FeatureManager design tree.

2 Enable both **Automatic Feature Recognition** and **Interactive Feature Recognition** in the **FeatureWorks** menu.

3 Click **FeatureWorks, Recognize Features**.

   FeatureWorks attempts to automatically recognize features first.

   • If FeatureWorks converts the entire imported solid into features, you return to the SolidWorks window.

   • If FeatureWorks does not convert the entire imported solid into features, the imported solid changes to show only the geometry that has *not* yet been recognized.

4 Click **Continue** to interactively recognize features, or click **Cancel** to return to the original imported solid body.
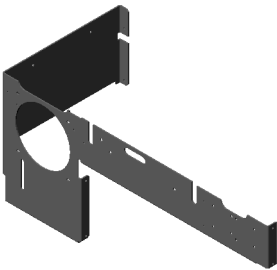
Imported solid body
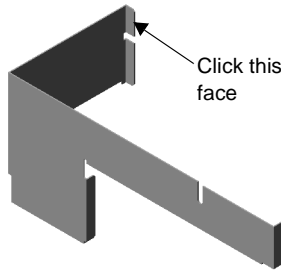
After Automatic Recognition (only geometry *not* recognized as features is displayed)

Click this face
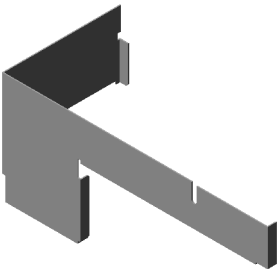
After Interactive Recognition of a Boss-extrude (boss is removed from geometry)
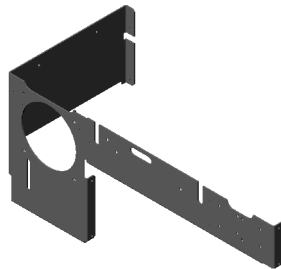
Once feature recognition is complete, original solid is displayed again with fully editable features

14

Object Linking and Embedding

You can use OLE to take advantage of features of other applications while in a SolidWorks document. You can also link or embed a SolidWorks part, assembly, or drawing document to another OLE-compliant application.

For example, you may need to perform advanced mathematical calculations to determine some dimensions on a part. Or you may want to include a SolidWorks part in a document such as a product data sheet.

OLE allows you to both bring data generated by other applications into the SolidWorks application and to place SolidWorks data into other applications.

This chapter introduces Object Linking and Embedding (OLE) and describes:

❑ Linking vs. embedding

❑ Using data from other applications in a SolidWorks document

❑ Bringing SolidWorks data into other applications

Linking vs. Embedding

When using OLE, you can *link* or *embed* files. Linking files allows you to continuously change the contents of a file in all places where it appears without having to edit each individual occurrence. Embedding a file allows you to keep the edits to the file specific to the place in which you embedded it.

Linking Files

When you link a file, the file remains in its original location. Anything you change in the original file affects all the files to which it is linked.

For example, if you edit a SolidWorks assembly document that is linked to several Microsoft Word documents, the changes you make are reflected in both the original SolidWorks file and all the Word documents. Also, when you double-click on a SolidWorks image in a Word document, the SolidWorks application launches allowing you to edit the original file (if you have SolidWorks installed on your PC).

The illustration below shows an example of a linked file.

Original SolidWorks file

File containing linked
SolidWorks file

Editing the original file changes the linked file and
editing the linked file changes the original file.
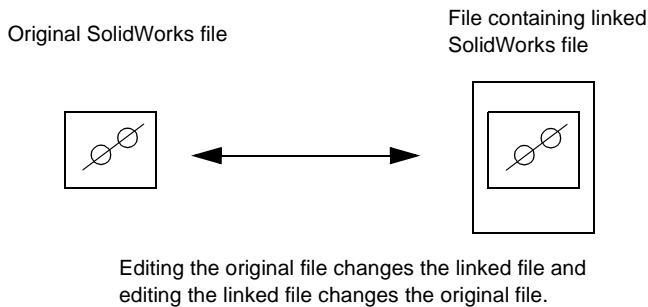
Linking is useful if you have data that may change and that you use in more than one place. By changing the original file, you automatically update the information in all the places to which it is linked.

NOTE: Keep track of the files to which you link data to avoid changing information in places you do not expect.

Embedding Files

If you embed one file in another, the original file becomes part of the file in which you embedded it. If you embed a SolidWorks document in a Word document, and if the SolidWorks application is loaded on the PC, double-clicking the image on the Word document page opens the SolidWorks application with the document active. You can edit the SolidWorks document and any changes you make affect only that document. Likewise, any changes you make to the original SolidWorks file do not affect the part embedded in the Word file.

Embedding is useful if you want discrete control over the data.

The illustration below shows an example of an embedded file.

Original SolidWorks file

File containing an embedded SolidWorks file. Changes were made to the embedded file only.

No link
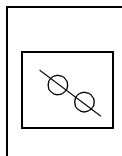
Editing the embedded file does not change the original file. Editing the original file does not change the embedded file.

Using Data from Other Applications in SolidWorks

With OLE, you can use other applications to generate data that you can then use in the SolidWorks application to control your parts, assemblies, and drawings. For example, you can calculate a dimension of a part using a math application. Or you can control the dimensions of a part using an Excel 97 spreadsheet. You can then link or embed the math application or Excel 97 spreadsheet to the desired dimension of the part in SolidWorks. Refer to **Design Table** on page 5-18 for details.

To use data from another application in a SolidWorks file:

1 With a SolidWorks file open, click **Insert**, **Object**.

2 To dynamically create and insert an object:

   • Click **Create New** in the **Insert Object** dialog box. The object appears in the SolidWorks file. Toolbars and menu options related to the object type, or application, that you selected are added to the SolidWorks window, allowing you to use the application's tools to edit the object you inserted.
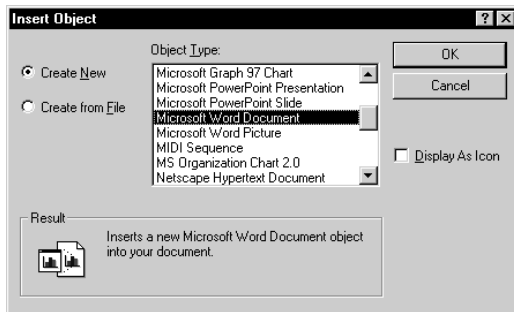
   To insert an existing object:

   • Click **Create from File** in the **Insert Object** dialog box. A file name appears in the **File** field.

   • Click **Browse** to select the desired object, and click **OK**.

3 To link the object to the SolidWorks file, click the **Link** check box in the **Insert Object** dialog**.** Otherwise the object is embedded.

4 To insert the object as an icon in the SolidWorks file, click **Display As Icon**.

5 Click **OK** to close the **Insert Object** dialog box.
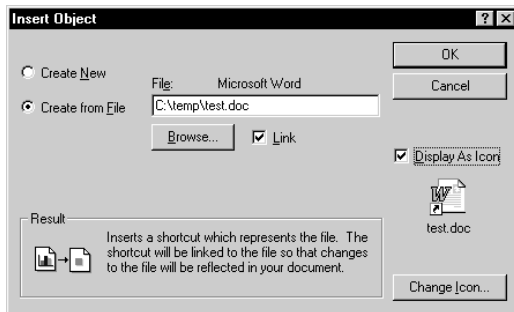
A Microsoft Word File in an Assembly

In this example, a Microsoft Word file, linked to the SolidWorks application, contains information about one of the components. Double-click on the Microsoft Word icon to view the contents of the file.
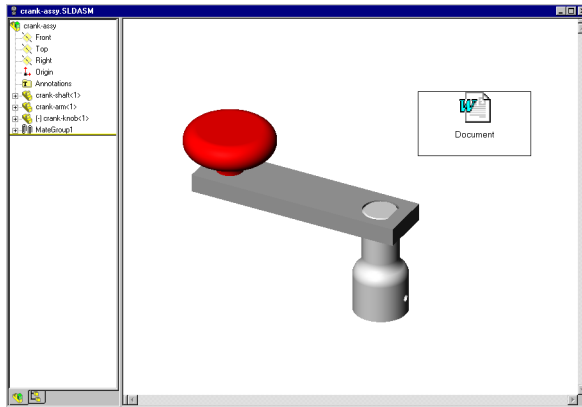
An OLE object in a SolidWorks Drawing

When you link an OLE object to a drawing document the default behavior is that the OLE object appears on every sheet of a multi-sheet document. If you want the OLE object to appear only on a single sheet, right-mouse click the icon and select **Show on this sheet only** from the menu.
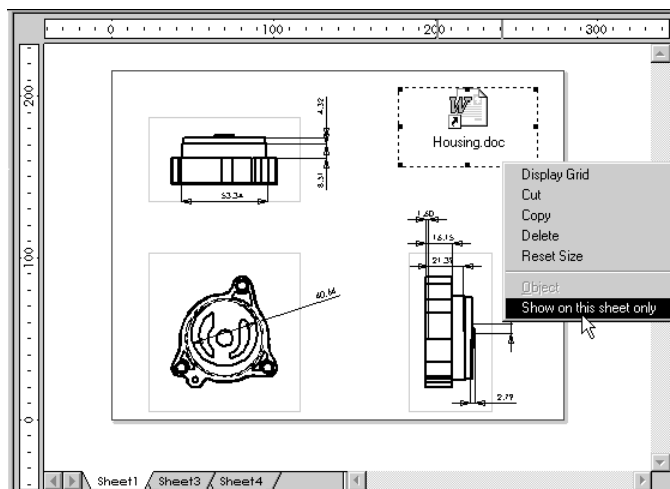
You can drag and drop selected OLE objects into SolidWorks documents. For example, if you drag highlighted text from a Microsoft Word file into an open SolidWorks document, the text becomes an embedded object.

Bringing SolidWorks Data into Other Applications

You may want to use a SolidWorks part, assembly, or drawing document in another application. Using OLE, you can link or embed the SolidWorks document with any other OLE-compliant application. For example, you can place a part in a product data sheet you created in Microsoft Word.

To use a SolidWorks file in another application:

1   In the desired application, click **Insert**, **Object**.

A dialog box appears allowing you to insert an existing object or to dynamically create and insert a new object. The dialog box that appears depends on the application you are using.

2   Select the desired options, including whether you want to link the SolidWorks file to this file and whether you want the SolidWorks file to appear as an icon in this file.

The SolidWorks file appears.

Example: An Assembly in a Word Document

This example shows a Word document containing a linked SolidWorks assembly document with the contents displayed.
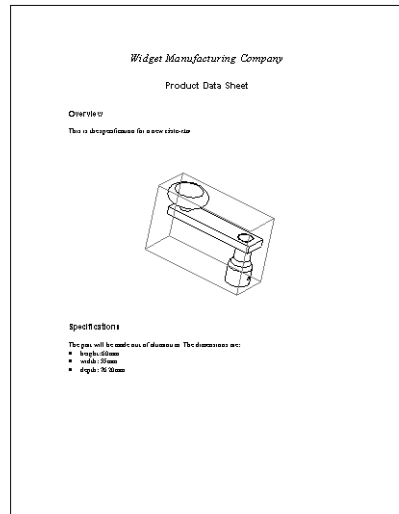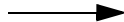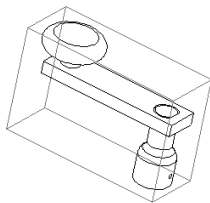
You can embed SolidWorks documents in other OLE containers by selecting the component's icon in the FeatureManager design tree and using the **Copy** and **Paste** functionality.

A

SolidWorks 98Plus Options

This appendix provides a list of all the options available for customizing part, assembly, and drawing documents. To access these options, click **Tools, Options** and select one of the tabbed pages. Change the settings as needed, then either click another tab, or click one of the following:

- **OK.** Accept the changes and exit the dialog. If you made changes on more than one page, *all* the changes are accepted.

- **Cancel.** Discard the changes and exit the dialog. If you made changes on more than one page, *all* the changes are discarded.

- **Reset All.** Return to the installed system defaults. This restores the original values for *all* the pages, whether you changed them in this session, or in an earlier session.

You specify some options at the *system level.* Options in these categories control certain system behaviors, and they apply to *all* documents:

- General
- Performance
- Export
- Drawings

- Edges
- Reference Geometry
- Import
- External References

You specify other options at the *document level*. You can choose default settings for all new documents, and you can change the values in any individual document, for options in these categories:

- Color
- Grid/Units
- Crosshatch

- Detailing
- Line Font
- Material Properties

When you change document level options, you also need to select an option from the **Apply to** list to specify the scope of the changes:

- **Active Document**. Apply the settings to *only* the currently active document.

- **System Defaults**. Apply the settings to all *new* documents. The currently active document does not change.

- **All Possible**. Apply the settings to both the **System Defaults** and the **Active Document**.

To review the system default settings on any of the pages, click **View System Defaults.** When checked, the system default for each of the options is displayed; when not checked, the option selections used by the active document are displayed.

Color Options

Specifies the color display of view modes, features, and lines. Click the **Edit** button to change the color used to display the items you selected. You can select a color from the standard color palette or create a custom color.

❑ **Part and assembly documents.** Setting colors used to display model lines and shaded surfaces in one of the view modes:

By default, the **System** box lists only the view modes for color selection:

- **Wireframe/HLR**, **Shaded**, or **Hidden.** Select the view mode for which you want to edit the color representation.

- **Apply same color to Wireframe/HLR and Shaded** box if you want to use the same color for those view modes.

If you edit the color of the **Shaded** mode, you can click **Advanced** and use the slider controls to change the display properties:

- **Ambient** – light reflected and scattered by other objects
- **Diffuse** – light scattered equally in all directions on the surface
- **Specularity** – ability to reflect light from a surface
- **Shininess** – a glossy, highly reflective surface
- **Transparency** – ability to pass light through the surface
- **Emission** – ability to project light from the surface

❑ **Part, assembly, and drawing documents.** Setting colors used to display lines, annotations, temporary graphics, highlighting, grid lines, borders in drawings, and many other items in addition to the view modes:

If the **System** box does not already display a list of lines, borders, dimensions, temporary graphics, etc., click the **View System Defaults** checkbox. Now, you can select from a large list of items or view modes for which you can change the color. Click **Edit** to select a color.

❑ **Part and drawing documents.** Setting colors used to display features:

The **Features** box lists the kinds of features and surfaces to which you can apply a color change.

- Click the feature type for which you want to edit the color representation, and click **Edit**.

- **Reset Feature.** Restores the original default color settings for the feature.

- **Ignore Feature Colors.** Specifies that the assigned feature colors are not used in the display.

Crosshatch Options

Specifies a crosshatch pattern to be used in section views, and in user-defined hatched areas.

See also **Modifying the crosshatch** on page 8-22, and **Area Hatch** on page 9-35.

Type

Displays the currently selected pattern.

Properties

- **Pattern.** Select a pattern that is appropriate for the part material from the list.
- **Scale.** Specifies the scale used for the pattern.
- **Angle.** Specifies the angle used for the pattern.

Apply To:

Crosshatch options are different than other options in the way that they are applied. You should specify the crosshatch pattern in any *part* document for which you will create a section view. When you use the part in an assembly, the appropriate crosshatch pattern for each component is used in assembly section views, also. That way, you do not have to edit the crosshatch pattern on the section faces for each component or region in the section view.

If you change the crosshatch options while an *assembly* or *drawing* document is active, you cannot choose an **Apply To** option. The changes you make are applied to the **System Defaults** (for new documents).
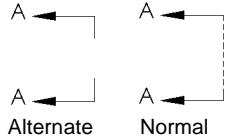
Detailing Options

Lets you set options for detailing and dimensioning in your parts, assemblies, and drawings.

Dimensioning Standard

- Specifies the standard to use: **ANSI, ISO, DIN, JIS, BSI,** or **GOST**.
- **Display with Broken Leaders.** Orients the text of vertical dimensions horizontally; inserts bends in the leaders if the text is outside the witness lines (if allowed by the selected dimensioning standard).
- **Dual Dimensions Display.** Specifies that dimensions are displayed in both primary and alternate units, and specifies the placement of the alternate dimension: either **On Top** or **On Right** (on the same line).
- **Fixed size weld symbols.** When unchecked (default), the size of the weld symbol is scaled according to the **Dim Font** size, and changes if the **Dim Font** size changes. When checked, the size of weld symbols is dependent on the selected dimensioning standard, and remains constant regardless of changes to the **Dim Font** size.

   This option is unavailable for the GOST dimensioning standard.

- **Display datums per 1982.** Click this checkbox to use the 1982 standard for the display of datums (available only if you use the ANSI dimensioning standard).
- **Trailing Zeroes.** Specifies the display of trailing zeroes. Tolerances are not affected.
  Smart – Trailing zeroes are trimmed for whole metric values. (Conforms to ANSI and ISO standards.)
  Show – Displays trailing zeroes up to the number of decimal places specified in **Tools, Options, Grid/Units**.
  Remove – All trailing zeroes are removed.
- **Alternate Section Display.** Displays an alternate style of section line (available with ANSI standard only). When checked, the section line does not cross the drawing view; the arrow ends stop at the ends of the section cut.

  Alternate   Normal

- **Centerline extension.** Controls the length of the centerline extension beyond the geometry in a drawing view, when the section cuts through the center of a hole, a circular extruded boss or cut, or a revolved feature. The default value is set according to standards, but you can modify the value in the edit box. The new centerline length is used the next time the section is rebuilt.

Notes

- **Note Font.** Specifies the font type and size used for notes.
- **Balloons.** Specifies the default balloon style, size, and text for Notes Balloons and BOM (bill of materials) Balloons.
- **Leader Anchor.** Specifies to which side of the text the leader attaches: **Left**, **Right**, or the side **Closest** to the attach point.
- **Text Justification.** Select **Left, Right,** or **Center**.
- **Display Notes with Bent Leader.** Specifies whether notes are displayed with a bent leader.
- **Bent Leader Length.** Specifies the distance between the leader bend and the text of the note.

Center Marks

- **Size.** Specifies the size of **Center Marks,** used with arcs and circles in drawings.
- **Show Lines.** Specifies whether the center mark lines are displayed.

Witness Lines

Sets the **Gap** (distance from model) and **Extension** (beyond the arrows) of witness lines.

Break Lines

Break Gap. Specifies the distance between break lines in a broken view.

Datum Feature Symbols

Specifies the letter of the alphabet that will be used for the next Datum Feature Symbol. The default letter is "A." Successive labels are in alphabetic order.

Dimensions

- **Dim Font.** Lets you specify the font type, style, and size used for dimensions.
- **Tolerance.** Lets you specify options for display of tolerances.

  Tolerance Display. Select from these types: **None**, **Basic**, **Bilateral**, **Limit**, **Symmetric, MIN, MAX,** or **Fit**. Depending on the type you choose, specify tolerance values (**Minimum/Maximum Variation**). You can specify tolerance display options for both **Linear** and **Angular** dimensions.

  Font. Select the font to use for the dimension tolerance text (not available for all types). By default, the tolerance is displayed in the same font as the dimension. To change the size of the tolerance text, deselect **Use dimension's font**, then select either **Font Scale** (enter a number from 0 to 10.0) or **Font Height** (enter a value).

- **Precision**. Sets the precision (number of decimal places) for dimension values and for tolerances, for **Primary Units**, **Angular Units**, and **Alternate Units**.
- **Add Parentheses By Default.** Specifies that reference dimensions in drawings are displayed within parentheses.
- **Snap Text to Grid.** Specifies that the placement of dimension text snaps to the grid in a drawing or a sketch.
- **Use System Separator.** Specifies that the default system decimal separator is used in the display of decimal numbers. (To set the system default use **Control Panel, International** (or **Regional Settings**), **Number Format, Decimal Separator.**) To set a decimal separator different from the system default, click to deselect and enter the symbol that you want to use (usually the period or the comma).
- **Center Text.** Specifies that the dimension text is centered between its witness lines.

Virtual Sharp

Represents a sketch point at the virtual intersection point of two entities. Dimensions and relations to the virtual intersection point are retained even if the actual intersection no longer exists, such as when a corner is removed by a fillet. Select one style of virtual sharp from the list. (This applies to both drawings and sketches.)

Arrows

- **Size.** Sets **Height** and **Width** of dimension arrowheads, and **Length** of arrow.
- **Section/View Size.** Sets **Height** and **Width** of arrowheads, and **Length** of arrow used in section views.
- **Attachments.** Sets style of arrowheads for attaching annotations to different entities. You can select a different style for attachment to an **Edge/vertex**, to a **Face/surface**, and for **Unattached** annotations.
- **Dimensions.**

  Set the arrowhead **Style** to use for dimensions.

  Specify placement of arrows in relation to the witness lines: **Outside**, **Inside** or **Smart**. The **Smart** option changes the placement of the arrows to outside if inside arrows interfere with the text of the dimension.

Display 2nd Outside Arrow (Radial). Specifies that two outside arrows are displayed with radial dimensions.

Arrows follow position of text (Radial) (only available if **Smart** placement is checked). Allows arrows on radial dimensions to display inside the arc/circle when the text is inside the dimensioned arc/circle, and to move to the outside when the text position moves outside the dimensioned arc/circle.

Dimension offset distances. Lets you specify values for the distances between baseline dimensions. **From model (A)** is the distance between the model and the first baseline dimension. **From last dimension (B)** is the distance between dimension lines. This value is also used for **Align Dimensions Parallel** ⬚. See **Baseline Dimensions** on page 9-6 and **Aligning Dimensions** on page 9-8.

Annotations

- **Display filter.** Specifies which annotation types, if any, are displayed by default. To select specific types, deselect **Display all types**, and select the types you want.
- **Display items only in the view in which they are created.** When checked, any annotation is displayed only when the model is viewed in the same orientation as when the annotation was added.
- **Display annotations.** Toggles the display of annotations.
- **JIS surface finish size.** If you use JIS dimensioning standard, specify the size (1, 2, or 3 characters) or a custom scale for surface finish symbols.

The remaining options are only available in part and assembly documents. You can also set these options by right-mouse clicking the Annotations folder, and selecting **Details**.

- **Text scale.**
- **Always display text at the same size.** When checked, all annotations and dimensions are displayed at the same size, regardless of zoom.
- **Display assembly annotations.** Toggles the display of annotations, including annotations that belong to the assembly, and annotations that are displayed in the individual part documents.
- **Use assembly's setting for all components.** Sets the display of *all* annotations to match the setting for the assembly document, regardless of the setting in the individual part documents.

  You can use these two options together to display different combinations of annotations.

Section Font, Detail Font

Lets you specify different text fonts for the letter labels on section lines and detail circles, and the corresponding view labels.

See page A-2 for information about the **Apply To** list and the **View System Defaults** option.

Drawings Options

Lets you set default options for drawings.

Default sheet

> **Sheet Scale.** Specifies the default drawing scale for those cases when you choose **No Template** from the **Template to Use** dialog. This scale is used for all views (except **Detail** views) on the sheet.

Type of projection

> Specifies either **First Angle** or **Third Angle** projection for those cases when you choose **No Template** from the **Template to Use** dialog.

Default display for new drawing views

> Specifies the way models or assemblies are displayed in drawings.

- Select the default view mode:

  Wireframe – All edges are displayed.

  Hidden in gray – Displays visible edges normally; displays hidden edges in gray.

  Hidden lines removed – Displays only edges that are visible at the chosen angle; obscured lines are removed.

- If you selected **Hidden in gray** or **Hidden lines removed**, select the mode for viewing tangent edges:

  Tangent edges visible – The transition edge between rounded or filleted surfaces displays as a line.

  Tangent edges with font – The transition edge between rounded or filleted surfaces displays as a line using the default font for tangent edges defined in **Tools, Options,** on the **Line Font** page. See **Line Font Options** on page A-20.

  Tangent edges removed – The transition edge between rounded or filleted surfaces and other surfaces is not displayed.

Automatic placement of imported dimensions from model. Specifies that imported dimensions are automatically placed at an appropriate distance from the geometry in the view.

Display drawing view borders. When checked, displays a border (a gray box) around each drawing view. This is the default.

Automatic scaling of 3 view drawings. When checked, if you insert a drawing, all the views of a **Standard 3 View** are scaled to fit on the drawing sheet, regardless of the paper size selected.

Automatic update of BOM. When checked, the bill of materials in an assembly drawing is updated automatically to reflect component changes in the assembly. See **Bill of Materials** on page 9-29 for more information.

Automatic drawing view update when model changes. When checked, all views are updated automatically when the model changes. When unchecked, you must update the views manually. For more information, see **Updating Views** on page 8-28.

Show contents while dragging drawing view. When checked, the model is displayed as you drag a view. When unchecked, only the view boundary is shown while dragging. (Similar to **Show window contents while dragging** in Microsoft Windows.)

Dynamic drawing view activation

- When checked, the view closest to the cursor is activated automatically. To stop the dynamic activation temporarily, you can lock a view or the sheet. Right-mouse click the view or sheet, and select **Lock View Focus** or **Lock Sheet Focus**. Then right-mouse click again, and select **Unlock View Focus** (or double-click another view) or **Unlock Sheet Focus** (or double-click any view) to return to dynamic mode.

- When unchecked, you double-click a drawing view to activate it, or right-mouse click the view and select **Activate View**. To activate the sheet, double-click a blank area of the sheet, or choose **Activate Sheet** from the right-mouse menu.

Select hidden entities. Allows you to select hidden (removed) tangent edges.

Detail view scaling. Specifies the scaling for detail views. The scale is relative to the default scale of the sheet.

Detail item snapping (See also **Aligning Annotations** on page 9-17.) These options may be used together.

- **Infer when dragging corner.** When you select a detail item (note, dimension, etc.) and drag it by the corner handle, the corner can infer to the corners of other detail items.

- **Infer when dragging center.** When you select a detail item (note, dimension, etc.) and drag it by the center handle, the center can infer to the centers of other detail items.

Print out-of-date drawing views with crosshatch. Lets you specify what happens when you **Print** (or **Print Preview**) a drawing that contains out-of-date views, with **Auto view regeneration** OFF. (If **Auto view regeneration** is ON, the views update automatically, and the crosshatch is removed, whenever you print the drawing.)

- **Prompt** (default). Notifies you if the drawing contains out-of-date views, and asks how to proceed. When the dialog box appears, click **Yes** to print the drawing *with* crosshatch on the out-of-date views, or click **No** to print the drawing *without* crosshatch.

- **Always**. The printed drawing always includes crosshatch on out-of-date views.

- **Never**. The printed drawing never includes crosshatch on out-of-date views.

Edges Options

Lets you set options for the display of various edge types.

Hidden Edges

- **Gray** or **Dashed.** Specifies whether hidden edges are displayed as gray lines or dashed lines.
- **Select Hidden for Wireframe/HLG.** Allows you to select hidden edges or vertices in **Wireframe** and **Hidden In Gray** modes.
- **Select Hidden for HLR and Shaded.** Allows you to select hidden edges or vertices in **Hidden Lines Removed** and **Shaded** modes.

Part/Assembly Tangent Edge Display

Controls the display of tangent edges in **Hidden Lines Removed** and **Hidden In Gray** modes.

- **Tangent Edges Visible** – Tangent edges are displayed as solid lines.
- **Tangent Edges as Phantom** –Tangent edges are displayed using the Phantom style line font (a broken line).
- **Tangent Edges Removed –** Tangent edges are not displayed.

Shaded Display

- **No Edges** (default). No edges are shown in shaded view mode.
- **HLR Edges**. Any edges that would appear in **Hidden Lines Removed** mode are displayed in **Shaded** mode also.
- **Wire Edges**. All edges are shown in shaded mode (wireframe).

Repaint After Selection in HLR. When selected, specifies that a selected feature is repainted in HLR if you click on space. When *not* selected, specifies that the selected feature is repainted in wireframe.

Highlight All of Selected Feature. Specifies that the entire feature is highlighted when you click on it.

Dynamic Highlight from Graphics View. Specifies whether model faces, edges, and vertices are highlighted when you move the cursor over a sketch, model, or drawing.

Export Options

Specifies the settings for exporting SolidWorks files.

For more information, see Chapter 13, "Importing and Exporting Files."

IGES

Output to
- **Settings For.** Lists the formats for various mechanical design application programs to which SolidWorks can export IGES files: Standard, ANSYS, COSMOS, Mastercam, SURFCAM, SmartCAM, and TEKSOFT.
- **Pop-up Dialog Box Before File Saving.** If **Trimmed Surfaces** is also selected, the program displays the **Settings For** list every time you save an IGES file so you can select a different format type. (You do not have to change the **Options** each time you export an IGES file to a new format.)

Output as
- **Trimmed Surfaces.** Specifies the output of trimmed surfaces.
- **Curves (3D).** Specifies the export of 3D curves. The solid body is converted to 3D wireframe representation. Select either **B-Splines** or **Parametric Splines** depending on the entity types required by the system to which you are exporting.
- **Duplicate Entities.** Select this option to export composite curves (entity type 102) to any of the available export formats.

Trim Curve Accuracy
- **Normal.** Select this option when smaller file size is preferred and high curve accuracy is not needed.
- **High.** Select this option when high curve accuracy is essential; the file size is larger than when using the **Normal** setting.

Assembly Output As

When exporting an assembly file to IGES, use the **A Single File** and **Flatten Hierarchy** options alone or in combination to create the following files:
- Single file without hierarchy (both options ON)
- Single file with hierarchy (**A Single File** ON)
- Multiple files and a master file without hierarchy (**Flatten Hierarchy** ON)
- Multiple files and a master file with hierarchy (both options OFF)
  Flatten Hierarchy reduces an assembly to a top-level assembly with one level of components; sub-assembly components are at the same level as components of the top-level assembly.

Parasolid

Assembly (For export only.)

- **Flatten Assembly Hierarchy.** A file containing a flattened hierarchy consists of a top level assembly and one level of parts. All subassemblies are eliminated.
- **Maintain Assembly Hierarchy.** A file which saved the assembly hierarchy consists of assemblies, sub-assemblies, and parts, and the structure mirrors the original assembly and its nested subassemblies.

Output as. Select the appropriate version for the target system.

DXF/DWG

Version. Specify if a file saved with a **.dxf** or **.dwg** extension will be in **R12**, **R13**, or **R14** format.

Fonts. Select either **Truetype Fonts** or **AutoCAD STANDARD only**.

Custom map SolidWorks to DXF. Click the checkbox to implement custom DXF mapping. When the **Custom map SolidWorks to DXF** option is checked, you can either create a new map file or select a file from the **Map File** list. Use the **Browse** button to search for a map file.

Don't show mapping on each save. Select this option to prevent the mapping dialog box from appearing every time you save a drawing with **Custom map SolidWorks to DXF** checked.

ACIS (.sat)

Version. From the pulldown list, select the default version in which to save files with the **.sat** extension. Select the appropriate version for the target system.

Units. From the pulldown list, select the default units of measure to use for ACIS files: **Millimeters**, **Centimeters**, **Meters**, **Inches**, or **Feet**.

VRML (.wrl)

Save all components of an assembly in one file. Remove the check to specify that all assembly components will be saved in individual files in a directory that has the same name as the assembly.

TIFF Options

Click the **TIFF Options** button to specify the following options:

- **Image Type.** From the pulldown menu, select either **RGB Full Color**, or **Black & White** image.
- **Compression Scheme.** From the pulldown menu, select one of the following: **Uncompressed**, **Packbit**, or **Group 4 FAX**. (For Group 4 FAX compression, you must use the **Black & White** image type.)

- Select either **Print capture** or **Screen capture**.
- If you select **Print capture**, you can specify **DPI** (dots per inch), **Paper size**, and **Scale** factor, or click **Scale to fit** to make the image fit the selected paper size. These options apply only to *drawing* documents.

STL Options

Click the **STL Options** button to specify the following options:

Output Format

- Select either **Binary** or **ASCII** as the STL file format.
- **Do not translate STL output data to positive space.** Check this box if you want exported parts to maintain their original position in global space, relative to the origin.
- **Save all components of an assembly in one file.** If you do not check this option, each component is written out relative to the assembly origin and saved in an individual file.

Quality

Controls the tessellation of cylindrical surfaces for Stereolithography (STL) output. A lower deviation setting results in slower model rebuilding, but more accurate curves.

Coarse and **Fine** are preset resolutions. **Custom** allows you to specify the resolution by dragging the **Total Quality** and **Detail Quality** slider controls or by entering values for **Deviation** and **Angle Tolerance**.

- **Total Quality.** Reports the maximum chordal **Deviation** in effect at the various tuning levels. The chordal deviation value is greater at the **Coarse** setting, and decreases as the resolution becomes more finely tuned.
- **Detail Quality. Angle Tolerance** controls tessellation for smaller details on the model. Use a value from 0.5 to 30 degrees. A lower value gives greater small-detail accuracy, but takes longer to generate.

Preview. Select this option to see a preview of the model with the borders of the tessellation triangles visible.

Show STL Info Before File Saving. Displays a faceted model preview in the part window and displays a dialog box with the following information: triangles (number), binary file size, file format, and the directory path and file name.

External References Options

Specifies how assembly, part, and drawing files with external references are opened and managed.

Open referenced documents with read-only access. Specifies that all referenced documents will be opened for read-only access by default.

Don't prompt to save read-only referenced documents (discard changes). Specifies that when this part is saved or closed, no attempt will be made to save its referenced documents that are opened for read-only access.

Allow multiple contexts for parts when editing in an assembly. Allows you to create external references to a single part from more than one assembly context. Any individual feature or sketch within the assembly may only have one external reference, however.

Search document folder list for external references. Specifies that the document folder list is searched to locate or update any referenced documents. See **Searching for Referenced Documents** on page 7-3 for information about using the document folder list.

Load referenced documents. Lets you specify whether to load referenced documents when you open a part that has external references (such as a derived part, or a part that has been edited in the context of an assembly). See also **File Management with External References** on page 5-14).

- **Never**. Referenced documents are never opened.
- **Always**. Referenced documents are always opened, in a minimized window.
- **Prompt**. Ask about loading externally referenced documents each time you open a derived part. Click **Yes** or **No** in the dialog box that appears when you open the part. If you also click **Don't ask me again** (before you click **Yes** or **No**), the option is updated to reflect your choice (**Yes** changes the option to **Always**, **No** changes the option to **Never**).

Folders

Show folders for. Displays search paths for files of various types.

You can specify the search rules for locating referenced documents. Folders are searched in the order in which they are listed in the **Show folders for** list.

Select a file type from the list (**Documents, Palette parts, Palette features, Palette forming tools,** and **Custom symbols**), then specify the paths to locate files of the selected type:

- To add a new directory path to the list, click the **Add** button.
- To delete a directory path from the list, select the path, and click the **Delete** button.
- To change the order of the list, select a directory path and use the **Move Up** button or the **Move Down** button as needed.

> **NOTE:** The paths for **Documents** are only searched if **Search document folder list for external references** is checked.
>
> The paths for **Palette parts, Palette features, Palette forming tools**, and **Custom symbols** are *always* searched, whether **Search document folder list for external references** is checked or not.

Assemblies

Automatically generate names for referenced geometry. This option is OFF by default.

When this option is off, you can mate to parts for which you have read-only access because you are using the internal face IDs of the parts. Unless you will use component replacement, leave this option off, especially in a multi-user environment.

When this option is on, you automatically create surface identifiers (for example: Face1, Face2) at the time you mate the part; therefore, you need write access to the part, in most cases. Turn this option on if you intend to do component replacement using the same surface identifiers, remembering that you need write access to the parts you are using. (Rename the corresponding edges and/or faces on the replacement component to match the edge/face names on the original part.)

Update component names when documents are replaced. This option is ON by default.

When this option is on, the component name in the FeatureManager design tree is the name of the part file. If you replace a component, the name in the tree updates to use the name of the new part file.

When this option is off, you can use the **Component Properties** dialog box to assign a component name in the FeatureManager design tree that is different from the filename of the component. If you replace a component, the name in the tree does not change.

General Options

Allows you to customize SolidWorks behavior and set default values.

Model

- **Input dimension value.** Automatically displays the modify spin box for input of a dimension value when you place the dimension.
- **Single command per pick**. Sketch and dimension tools deselect after each use. (Double-clicking a tool will cause it to remain selected.)
- **Show dimension names.** Displays the dimension's name as well as its value.
- **Show errors every rebuild.** If errors are present in the model construction, display an error message each time the model rebuilds.

FeatureManager Design Tree

- **Scroll selected item into view.** Specifies that the FeatureManager design tree should automatically scroll to display the text that is related to the selected items in the graphics area.
- **Name feature on creation.** When you create a new feature, the feature's name in the FeatureManager design tree is automatically selected and ready for you to enter a name of your choice.
- **Arrow key navigation.** Lets you use the arrow keys to traverse the FeatureManager design tree, and expand or collapse the design tree and its contents, as follows:
  Up arrow – scrolls up the design tree
  Down Arrow – scrolls down the design tree
  Left arrow at top of design tree – collapses the design tree
  Right arrow at top of design tree – expands the design tree
  Left arrow on an item in the tree – collapses the item to hide its contents
  Right arrow on an item in the tree – expands the item to display its contents, if any
  Space bar – selects the item
- **Dynamic highlight.** Specifies that the geometry in the graphics area (edges, faces, planes, axes, etc.) is highlighted when the cursor passes over the item in the FeatureManager design tree.

Sketch

- **Use fully defined sketches.** Requires sketches to be fully defined before you can use them to create features.
- **Alternate spline creation.** Lets you create splines by clicking on through points instead of dragging out segments.
- **Display arc centerpoints.** Turns the display of arc centerpoints on or off in a sketch.
- **Display entity points.** Specifies the display of sketch segment endpoints as filled circles in a sketch. The color of the circle indicates the status of the sketch entity: Black = Fully defined, Blue = Under defined, Red = Over defined, Green = Selected. (Over defined and dangling points are always displayed, regardless of the **Options** setting.)
- **Infer from model.** When sketching on the face of an extruded part, inferencing lines and the inferencing cursor relate to the lines of the part.
- **Prompt to close sketch.** With this option selected, if you make a sketch with an open profile that can be closed with the model edges to extrude a boss, the system displays a dialog, **Close Sketch with Model Edges?** You can choose the model edges to close the sketch profile and the direction. The **Extrude Feature** dialog then appears. Otherwise, only the **Extrude Feature** dialog is available.
- **Create sketch on new part.** When you select **New, Part**, the part window opens with the sketching area and sketch tools immediately available.

- **Enable silhouettes.** With this option, silhouettes are selectable and you can use the following sketch tools on these edges: **Convert Entities**, **Offset Entities**, **Add Relation**, and **Dimension**.
  Enabling silhouettes may affect performance when activating a sketch for editing if the model is complicated.
- **Override dims on drag.** Lets you override dimensions by dragging sketch entities. The dimension updates after the drag is completed. (Also available from the menu: **Tools**, **Sketch Tools**, **Override dims on drag**.)
- **Automatic relations.** Specifies whether geometric relations are automatically created as you add sketch elements. (Also available from the menu: **Tools**, **Automatic Relations**.)
- **Display plane when shaded.** When you are editing a sketch in **Shaded** mode, this option displays the translucent plane. (By default, old models will have this checked.)

NOTE: If the display is slow, due to the translucent plane, it may be because of the **Transparency Quality** specified in **Tools, Options, Performance**. With some graphics cards, the display speed will improve if you use **Low (Screen door)** transparency.

Over defining Dimensions

- **Prompt to set driven state.** When checked, specifies that when you add an over defining dimension to a sketch, a dialog box asks you if the dimension should be *driven*. (The default is to ask.)
- **Set driven by default.** When checked, specifies that when you add an over defining dimension to a sketch, the dimension is set to be *driven* by default.
  The above two checkboxes can be used together or alone, resulting in different behaviors when you add an over defining dimension to a sketch.

View Rotation

- **Arrow Keys**. Lets you set the angle increment for view rotation when you use the arrow keys to rotate the model.
- **Mouse Speed.** Lets you set the speed of the rotation when you use the mouse to rotate the model or assembly component. Move the slider to the left to get finer control and slower rotation.

General

- **Open Last Used Documents at Startup.** Select either **Always** or **Never**. Select **Always** if you want the convenience of having the documents that you used most recently open automatically when you start SolidWorks.
- **Number of backup copies per document.** Lets you specify the maximum number of backup copies you want to save of part, assembly or drawing documents. Specify zero for no backups, or up to the maximum number of backups you want to save. The name of a backup copy is "Backup (*n*) of *original filename and extension.*"

- **Save backup copies in directory.** Lets you name a directory to which all backup copies will be stored by default. Use the **Browse** button to find the directory to use.
- **Maximize document on open.** When checked, each document opens to its largest size within the SolidWorks window.
- **Save AutoRecover info every (*n*) changes.** The software can automatically save information about your active part, assembly, or drawing documents so you will not lose your work in case of a system crash. Use the spin box to specify the number of changes that should occur before information is automatically saved.

NOTE: In a part or assembly document, a *change* is a rebuild or an action that requires a rebuild, such as the addition of a feature. In a drawing document a *change* is any action such as changing a dimension, creating a section view, or adding an annotation.

To disable **AutoRecover**, click to remove the check from the checkbox.

- **Use English language.** If you selected the use of a language other than English during the SolidWorks installation, you can change to English by selecting this check box. Note that you must exit and re-start SolidWorks for this change to take place.
- **Show thumbnail graphics in Windows Explorer**. When checked, a thumbnail graphic is displayed in Windows Explorer, instead of an icon, for each SolidWorks part or assembly document. The graphic is based on the view orientation of the model when the document was saved. A palette of 16 colors is used; if the model uses a color that is not available, a similar color is substituted. The thumbnail graphics also are used in the **File, Open** dialog box.

Grid/Units Options

Sets sketch grid properties and units of measure.

Grid

Properties

- **Display Grid**. Turns the sketch grid on or off.
- **Dash**. Toggles between solid and dashed grid lines.
- **Automatic Scaling**. Adjusts display of the grid when you zoom in and out.
- **Major Grid Spacing**. Specifies the space between major grid lines.
- **Minor-Lines Per Major**. Specifies the number of minor grid lines between major lines.

Snap Behavior

- **Snap to Points**. Turns grid snap on or off. When snap is on, points that you sketch or drag snap to the nearest intersection of grid lines (or to intermediate points, if **Snap Points Per Minor** is greater than 1).
- **Snap Points Per Minor.** Specifies the number of snap points between minor grid lines.
- **Snap to Angle.** Specifies the number of degrees between snap points when sketching arcs.
- **Snap only when grid is displayed.** Turns off snap behavior when the grid is not displayed.

Units

Length Unit

- Select either: **Millimeters, Centimeters, Meters, Inches, Feet,** or **Feet and Inches**.
- If you select **Inches**, or **Feet and Inches** as your unit of measure, select **Decimal** or **Fractions**.
- If you choose **Decimal**, specify the number of **Decimal Places**.
- If you choose **Fractions**, specify the default **Denominator**. (Only dimensions that are evenly divisible by this denominator are displayed as fractions.)
- If you choose **Fractions** and **Inches** or **Feet & Inches**, you can select **Round to Nearest Fraction**.

Angular Unit

- Select either: **Degrees**, **Deg/Min**, **Deg/Min/Sec**, or **Radians**.
- If you choose **Degrees** or **Radians**, specify the number of decimal places.

Spin Box Increments

- **Length.** The number of units added/subtracted when you click on a spin box arrow to change a linear dimension value.
- **Angle.** The number of degrees added/subtracted when you click on a spin box arrow to change an angular dimension value.

See page A-2 for information about the **Apply To** list and the **View System Defaults** option.

Import Options

Specifies the default response to make when there is an error in importing a file.

Error resolution. From the pulldown lists next to the error types, select the way that you want SolidWorks to resolve problems.

- The solutions for **Surface Errors**, **Surface Flaws**, and **Significant Digits** are:
  Prompt
  Try to knit into solid
  Try to form reference surfaces
  Cancel operation
- The solutions for **Blanked Surfaces** are:
  Prompt
  Ignore blanked surfaces
  Do NOT ignore blanked surfaces
  Cancel operation

Significant Digits and **Blanked Surfaces** apply to IGES only.

Try Forming Solid Model(s) From Surfaces. For IGES, STEP, SAT, or VDA, check this option if you want the software to attempt to knit the imported surfaces, in order to create a solid model.

For more information, see Chapter 13, "Importing and Exporting Files."

Line Font Options

Specifies the style and weight of edge lines for selected kinds of edges.

Type of Edge. Select an edge type from a list of line types: **Visible, Hidden, Sketch Curves**, **Dimension**, **Section Line**, etc.

Line Style. From the list, choose a line style to apply to the previously selected edge type. You can choose line styles such as **Solid, Dashed, Phantom**, etc.

Line Weight. From the list, choose a line weight to apply. Choose from **Normal, Thin,** or one of several **Thick** lines. The thickness of the lines corresponds to the **Line Weights** for printing that you specify under **File, Page Setup**, on the **Printer** tab.

The **Preview** box displays the selected line.

Apply To
- If the active document is a *part* or *assembly*, you cannot choose an **Apply To** option. The changes you make are applied to the **System Defaults** (for new documents).
- If the active document is a drawing, you can specify where the changes are applied. See page A-2 for information about the **Apply To** list.

Material Properties Options

Specifies the material properties for the current part. This option is available when you are working with an active *part* document.

This page is also used by some **Add-In** applications. Refer to the instructions of the specific application for further information.

Properties. Lets you specify material properties by selecting the text in the box, and entering the appropriate **Density** specification.

See page A-2 for information about the **Apply To** list and the **View System Defaults** option.

Performance Options

Sets performance options. You may choose the display quality that is best for your needs, understanding that higher display quality impacts the speed of redrawing the model.

Shaded Display Quality

Controls the tessellation of cylindrical surfaces for shaded rendering or Stereolithography (STL) output. A higher resolution setting results in slower model rebuild, but more accurate curves.

- **Coarse** and **Fine** are preset resolutions. **Custom** allows you to specify the resolution by dragging the slider control or by entering a **Deviation** value.
- **Deviation.** Reports the maximum chordal deviation in effect at various tuning levels. The chordal deviation value is greater at the **Coarse** setting, and decreases as the resolution becomes more finely tuned. The range of values for the deviation is relative to the overall size of the part.

Wireframe Display Quality

- **Optimal.** A preset quality that provides a faster redraw without greatly sacrificing the display quality.
- **Custom.** Allows you to choose the display quality by dragging the slider control. Choose the **Low** end of the slider if you want to redraw the screen more quickly and the display quality is not of great importance; choose the **High** end of the slider for high display quality, but a slower redraw.

NOTE: If you are noticing problems with your **HLR** image display, you may want to select a higher quality wireframe display.

Rebuild. **Verification on Rebuild** controls the level of error checking when creating or modifying features. For most applications, the default setting (off) is adequate, and results in faster model rebuild. See also **Force Regeneration with Verification on Rebuild** on page 5-24.

Windows95 Zooming. When using SolidWorks on Windows95, there is a limit beyond which you cannot zoom in on a model. The **Enable clipping for zoom limitation** option lets you select a portion of the model and zoom in on only the selected portion. You can zoom in on small details very closely, but the display is slower.

Transparency Quality. Select either **Low** or **High**. **Low** quality transparency (Screen Door) is similar to viewing an object through a mesh or screen; **High** quality transparency (Alpha Blending) is similar to looking through clear glass.

Curvature Generation

- When **Only on demand** is checked (the default), the initial curvature display is slower, but it uses less memory.
- When **Always (for every shaded model)** is checked, the curvature displays more quickly on the first display, but extra memory is always used (RAM and disk) for every part that you create or open.

Assemblies

Automatically load parts lightweight. You can only change this setting when no assemblies, or drawings of assemblies, are open. While this option is enabled, all the individual parts in assemblies that you open are loaded **Lightweight**. Sub-assemblies are not lightweight, but the parts that they contain are lightweight. See **Suppression States** on page 6-31 for more information.

Resolve lightweight parts. Some operations (such as **Mass Properties, Interference Check**, displaying non-shaded views and section views, exporting to other file formats) require certain model data that is not loaded for lightweight parts. This option controls what happens when you request one of these operations in an assembly that has lightweight parts.

- **Always**. Lightweight parts are automatically resolved.
- **Prompt** (default). The system offers to resolve lightweight parts for you each time one of these operations is requested. In the dialog box that appears, click **Yes** to resolve the parts and continue, click **Cancel** to cancel the operation. If you also click **Always resolve** (before you click **Yes** or **Cancel**), the option is set to **Always**.

Rebuild assembly on load. Lets you specify whether you want assemblies to be rebuilt, to update any changed components, when you open them.

- **Never**. Open assemblies without rebuilding.
- **Always**. Always rebuild assemblies when opening.
- **Prompt**. Ask if rebuild is desired each time an assembly is opened. Click **Yes** or **No** in the dialog box that appears when you open the assembly. If you also click **Don't ask me again** (before you click **Yes** or **No**), the option selected under **Rebuild assembly on load** is updated to reflect your choice (**Yes** changes the option to **Always**, **No** changes the option to **Never**).

Reference Geometry Options

Display Filter

Lets you set a filter for the default display of reference geometry items. Click the checkbox beside the types of items to display: **Planes**, **Axes**, **Temporary Axes**, **Origins,** and **Coordinate Systems**.

NOTE: To change the visibility of a type of reference geometry in the *current* document only, use the options under the View menu. To change the visibility of an individual reference geometry item, right-click the item and select **Hide** or **Show**.

Plane Default Names

Lets you specify default plane names for parts and assemblies. For example, you may want to name planes Front, Top, and Right, instead of Plane1, Plane2, and Plane3. Enter new names in the boxes that correspond to the original plane names.

SolidWorks Application Programming Interface

This appendix contains an overview of the SolidWorks Application Programming Interface (API). The SolidWorks API is an OLE programming interface to SolidWorks. The API contains hundreds of functions that can be called from Visual Basic, VBA (Excel, Access, etc.), C, C++, or SolidWorks macro files. These functions give you direct access to SolidWorks functionality; for example, creating a line, extruding a boss, or verifying the parameters of a surface. Complete online documentation is supplied with the SolidWorks API.

Online Documentation:

The online documentation supplied on your SolidWorks CD documents every object available in the SolidWorks API. Use it as you would use any conventional Windows online help, accessing topics through the table of contents, the index, or by using the Find capability to search for key words or phrases. The complete SolidWorks API documentation and a list of the new functionality added since the previous release of the SolidWorks software is located in the **...\Samples\Appcomm** subdirectory of your SolidWorks installation. Double-click on **API_help.hlp**. The SolidWorks API documentation can also be found on the SolidWorks web page:

http://www.solidworks.com/html/Products/api

Topics in this Appendix:

❑ Recording and playing SolidWorks macros

❑ Getting started and installing with C++ and Visual Basic

❑ Compiling and running your application with C++ and Visual Basic

❑ Programmer's guide and overview of the OLE automation interface

❑ Programming topics

❑ SolidWorks API objects

Recording and Playing SolidWorks Macros

You can record operations performed with the SolidWorks user interface and replay them using SolidWorks Macros. These macros are actually the equivalent calls to the API functions that were made when the operations were performed with the user interface.

To add the Macro toolbar:

Click **View, Toolbars, Macro**.

To record a SolidWorks macro

1 Click ▐▐● or **Tools, Macro, Record**.

2 Perform the operations you wish to have in your macro.

3 Click ▣ or **Tools, Macro, Stop** and enter a filename for the macro. (Macros have a **.swb** extension.)

To run a SolidWorks macro:

1 Click ▶ or **Tools, Macro, Run**.

2 Choose the filename of the macro you wish to run.

To edit a SolidWorks macro

1 Click ▦ or **Tools, Macro, Edit**.

2 Choose the filename of the macro you wish to edit.

3 You can now modify, run, and debug your macro.

Note: SolidWorks Macros recorded in this way usually have some context-dependent elements that require minor changes before they can be used with other documents. For example, a macro recorded with Part1 may need to be changed so that it plays back with another document.

Installing and Getting Started with C++

Installing SolidWorks to compile and run your C++ applications requires the following:

MSDEV

MSDEV, Microsoft Developer Studio, must be installed with all appropriate libraries. It is recommended that you perform a Full Install to avoid unforeseen problems. However, if you wish to customize your setup, refer to the requirements below. Your installation should be for Visual C++ revision 5.0.

❑ **Windows NT** must have all UNICODE libraries installed

❑ **Windows 95** must have MBCS libraries installed (these are the default libraries for MSDEV)

❑ **Alpha** must have all UNICODE libraries installed

SolidWorks recommends that all operating systems also have the Shared MFC and MSVCRT Libraries installed.

SolidWorks

Development using the SolidWorks API is done with a regular installation of SolidWorks.

Note: The /API installation option is no longer available when you install SolidWorks because the debug libraries are no longer supplied on the SolidWorks CD. If you wish to DEBUG your code in a dynamic link library, then you can still build the debug information in your DLL file. (See the section Compiling Your C++ Applications.)

Compiling Your C++ Applications

Visual C++ 5.0 should be used to develop your C++ applications. SolidWorks allows you to create and run your project as a standalone **.exe** file or as a User DLL or Extension DLL.

1 Use a **MAK** file from a project in the *Samples* subdirectory to determine the build properties or simply load an existing project (**.mdp** file) and begin cutting and pasting your own code.

2 If you chose not to use the setups from one of the existing projects, you need to bring in the SolidWorks API declarations by yourself. You can generate these from the **\SldWorks\solidworks.tlb** type library, or you can simply include **\SldWorks\Samples\AppComm\swdisp.cpp** and **swdisp.h** into your project. The **swdisp.cpp** and **swdisp.h** files expose all objects available in SolidWorks. If you use a COM interface instead of Dispatch, include **amapp.h** instead of **swdisp.h** and **swdisp.cpp** .   The **amapp.h** file also includes the header information needed for anyone using event notification.

3 Choose the correct build configuration (Win32 Release, Win32 Debug, Win32 MBCS Release, or Win32 MBCS Debug):

- Windows 95 should use the MBCS configurations (Win32 MBCS Release or Win32 MBCS Debug.):

- Windows NT should use the Unicode configurations (Win32 Release or Win32 Debug.)

4 If you choose a Debug configuration you *must* also change project settings for that configuration as follows:

- Click **Project, Settings** to display the settings dialog.

- Click the **C/C++** tab and select **Code Generation** from the **Category** list. Select **Multithreaded DLL** from the Use run-time library (the previous setting should be **Debug Multithreaded DLL**).

- Click **Preprocessor** from the **Category** list and change _DEBUG in the **Preprocessor** definitions text box to _NDEBUG.

- If you wish to use ASSERTions or TRACE, then select **Preprocessor** from the **Category** list and at the end of the **Preprocessor** definitions text add _PSEUDO_DEBUG. You need to add two Microsoft files to your project, debug.cpp and debug.h, which can be downloaded from the SolidWorks web site at:

   http://www.solidworks.com/html/Products/api/solpartner.htm.

- Click the **Link** tab and select **General** from the **Category** list. Make sure the libraries in the Object/library modules text box are release versions and not debug. As an example, use mfc42u.lib and not mfc42ud.lib.

5 Add your own code.

6 Build your project.

Note: SolidWorks uses the function `InitUserDll3` to initialize your DLL. Please refer to the sample projects in the **../Samples** directory for implementation guidelines. If your DLL is not initialized properly or you do not use `InitUserDll3`, then you will receive *invalid add-in* and *incompatible version* messages.

Running Add-In Applications

Running your application as a Release DLL

Loading 3rd party applications is handled by the SolidWorks Add-In manager, which is accessed from the Tools menu in SolidWorks. To add an application to the Add-Ins available to the user you have to register the application in the windows registry (this is described in the section *Using the Add-In Manager* in the API Help documentation.)

Once the application is registered, follow these steps to load (or unload) it:

1 Start a SolidWorks session.

2 Open any document or create a new document.

3 Select **File, Tools, Add-Ins...** .

4 Check (uncheck) the checkbox for the Application to be loaded (unloaded).

5 Select **OK**.

Once the Add-In is loaded then it will be automatically loaded whenever SolidWorks starts up.

Note: As of SolidWorks 98, the \APPS directory is no longer supported for automatic loading of applications.

Debugging Add-In Applications

If you are debugging your application, then you should start SolidWorks from your development environment. This allows you to set break points in your application and step through your code. If your application was built as a Debug DLL (see above), follow these steps:

1 From your development environment, select **GO**.

2 When you are prompted for the executable name, enter *SldWorks.exe* with its path name.

   If your application is registered in the windows registry, make sure it is loaded:

3 Select **File, Tools, Add-Ins...** .

4 Check the checkbox for the application to be loaded.

If your application is not registered you can still debug it once SolidWorks is running.

1 Click **File, Open** and change your file selection filter to **Add-Ins(*.dll)**.

2 Select the desired DLL file and click **OK**. This brings your application into SolidWorks.

Note: Only debug information for your DLL will be available because the release MFC libraries are used, so you will not be able to step into MFC functions or see stack information outside of your DLL.

Installing and Getting Started with Visual Basic

Installing SolidWorks to compile and run your Visual Basic applications requires no special steps. Simply install SolidWorks from the SolidWorks CD.

Compiling Your Visual Basic Applications

SolidWorks supports programs through any OLE controller (Visual Basic, Visual Basic for Applications, etc.).

Applications written in Visual Basic can be started from many different points. In any of the cases below, the code generated by you should be similar to code generated by the SolidWorks Macro utility. To get started, it may be helpful to generate a macro from within SolidWorks (**Tools, Macro, Record**) and then use that code as the foundation for your application.

❑ Programs to Run From SolidWorks

There is no need to compile your application. If your routine uses only SolidWorks API calls, create your program in Visual Basic and use a file extension of **.swb** instead of **.bas**.

Note: SolidWorks Macro files (*.swb) only recognize Visual Basic 3.0 commands. If you wish to use Basic commands in Visual Basic 4.0 or higher, then you must compile and run the program as a Visual Basic executable or from a VBA application.

❑ Programs to Run as Separate **.exe** File

Build your project with a standard utility such as Microsoft Visual Basic. From the Microsoft Visual Basic application, select:

File, Make EXE File…

❑ Programs to Run as a DLL

SolidWorks does not support Visual Basic DLL implementations.

❑ Programs to Run From Other Applications

1 Load the application (Access, Excel, etc.).

2 Use the embedded VBA to generate your utility or script.

3 Use the compile utilities within the application to build your project.

Running Your Visual Basic Applications

Applications written in Basic can be started from many different points.

❑ Running From SolidWorks

1 Click **Tools, Macro, Run**.

2 Choose the desired BASIC source file and click **OK**.

❑ Running From Customized SolidWorks Menu Buttons

Applications written as .swb files can be run from custom buttons in the SolidWorks user interface.

1 Place the .swb file in the Macros folder of the SolidWorks installation. (You may have to create it.)

2 Customize the user interface by clicking **Tools, Customize...** .

3 Click the **Menus** tab.

4 Click **Macros** under **Categories** and the Macro you added under **Commands**.

5 Choose the menu, position, and text for the new button, then click **Add**.

6 To run the macro, click the button you just added.

Note: Once added, the button will always appear in subsequent SolidWorks sessions.

❑ Running as a Separate **.exe** File

Execute your **.exe** file.

If a SolidWorks session is already running, then your program will attach to it. If not, then a new SolidWorks session will be started.

❑ Running From Other Applications

Load the application (Access, Excel, etc.) and run your program or script from the application.

If a SolidWorks session is already running, then your program will attach to it. If not, then a new SolidWorks session will be started.

Programmer's Guide

OLE Automation Interface

The functions listed in the following pages are specified in C++ format. They may be called from Visual BASIC, VBA (Excel), C or C++.

- C++ classes are generated in a header file and an implementation file by importing the type library for SolidWorks using the Visual C++ class wizard.

- In C implementations, you need to determine the program ID from the type library and set up an invoke handler.

- Calling from Visual Basic or other varieties of BASIC which support OLE Automation requires only that an object is defined (Set Object =) and the implementation of the Basic OLE implementation takes care of the rest.

The SolidWorks API interface uses an object oriented approach. All of the functions described in this document are methods or properties which apply to an object. Thus, there is an assumption that these methods will apply to the current state. For example, there is a method EditDelete with effectively no arguments. Instead of passing in the items to delete, the EditDelete method acts on the current set of selected items. This style of interface may be different from what you have encountered in previous products, however, it is consistent with the Windows-based approach of SolidWorks. By way of reference, all of the SolidWorks user interface is based on Microsoft Foundation Classes.

COM vs. Dispatch

SolidWorks exposes functionality through OLE automation using IDispatch and also through standard COM objects.

❑ The Dispatch interface packages arguments and return values as Variants so they can be handled by languages such as Basic.

❑ A COM implementation gives your application more direct access to the underlying objects, and subsequently, increased performance.

COM implementations also have an HRESULT return value for each API function to indicate successful or unsuccessful calls and provide slightly more functionality with operations such as enumeration.

Sample Projects

SolidWorks provides you with several sample applications to get started. These Visual Basic and C++ projects can be found in the **Samples** subdirectory of your SolidWorks installation. Each project directory includes a **readme.txt** file so you can become familiar with each project and the highlighted functionality within that project. Refer to **...\Samples\comuserdll**, **...\Samples\userdll**, **...\Samples\TestApp**, and **...\Samples\VisualBasic**.

In most cases it is necessary to recompile each project on your system before attempting to run. For instructions on compile settings and requirements, see Compiling Your C++ Applications on page B-3 and Compiling Your Visual Basic Applications on page B-6.

For C++ applications you have the option of using either a COM or a Dispatch interface. A sample project exists for each interface type in the **...\Samples\ComUserdll** and **...\Samples\Userdll** directories respectively. The **...\Samples\TestApp** project is a second Dispatch example. Many developers simply load the **.MDP** file from the appropriate directory and cut and paste their code into the sample project. In the least, you should refer to the **.MAK** file for appropriate build settings.

Note: The sample projects are supplied on an as is basis, and are intended to demonstrate the method of using the OLE capabilities of SolidWorks. SolidWorks Corp. makes no representations or warranties regarding these samples.

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Helpful Hints

Using Macros to get a Head Start

To get a head start on any project, it may be helpful to generate a macro from within the SolidWorks program (**Tools, Macro, Record**). By recording a macro and performing the desired function interactively, you can get a start on the commands and syntax needed for your code. Record your macro before you do any coding and use it as a foundation for your project. When you require additional functionality in your program, go back to the SolidWorks program and record additional macros. Cutting and pasting your macros into the existing sample projects can be beneficial for even the most advanced programmer.

Keeping Your Visual Basic Form On Top

For an example of keeping your Visual Basic form on top of all other windows, refer to the example provided in the Visual Basic 4.0 help for the `hWnd` Property.

Checking for Empty or NULL Variant Return Values

In many situations the SolidWorks API may return an empty `VARIANT`. It is always a good idea to check for valid return values before proceeding with your program (See also **Return Values** on page B-12). The following C++ example shows you one method of checking for an empty `VARIANT`:

```
VARIANT v = m_ModelDoc.GetMassProperties();        // Get the Mass Properties
if (v.vt == VT_EMPTY)  ||  (V_VT(&v) == VT_NULL))// Error occurred
return;
```

SolidWorks API, Programming Topics

Implementation Guidelines

As a general rule of OLE programming, the caller is responsible for allocating and deallocating memory. This includes data returned by a SolidWorks function.

Interface Pointers

Interface pointers can also be an area of concern with C++ programming. Each SolidWorks API method which returns an interface pointer will automatically increment the Reference Count on the interface pointer by 1.

❑ For COM implementations, you may call a SolidWorks API which returns an interface pointer. You can then use this pointer as you wish, but you are responsible for releasing it.

This C++ COM example demonstrates how to handle interface pointers:

```
{ LPMODELDOC m_ModelDoc = NULL;     // Retrieve IModelDoc pointer
HRESULT res = UserApp->getSWApp()->get_IActiveDoc( &m_ModelDoc );
if( m_ModelDoc == NULL )
    return;
LPPARTDOC m_PartDoc = NULL;      // Retrieve IPartDoc pointer
res = m_ModelDoc->QueryInterface(IID_IPartDoc, (LPVOID *)&m_PartDoc);
ASSERT( res == S_OK );
...                              // Use the interface pointers within your code
m_ModelDoc->Release();           // Release the IModelDoc pointer
m_PartDoc->Release();            // Release the IPartDoc pointer
    }
```

❑ For Dispatch implementations, the release of the interface pointer is hidden in the destructor of the dispatch objects (`IModelDoc`, `IFace`, etc.). This implies that attaching an interface pointer to more than one of these dispatch objects, would cause a release to be performed by each of the objects as they go out of scope. This would cause a problem since the Reference Count is only incremented once when the interface pointer is returned to you. To avoid this problem, you must manually increment the Reference Count (`pdisp->AddRef();`) if you are attaching the interface pointer to more than one object.

This C++ Dispatch example demonstrates how to handle the Reference Count on interface pointers:

```
{LPDISPATCH modDisp;
modDisp = UserApp->getSWApp()->GetActiveDoc();
                           // Get interface pointer to the active document
                           // Ref Count on modDisp automatically incremented by 1
if( modDisp == NULL )
     return;
IModelDoc m_ModelDoc( modDisp );        // Attach to the IModelDoc object
IPartDoc m_PartDoc( modDisp );          // Attach to the IPartDoc object
modDisp->AddRef();          // Manually increment the Ref Count on modDisp
                           // because we use modDisp a second time
...                        // Use objects within your code
    }                      // Variables go out of scope and destructor called for
                           // IModelDoc and IPartDoc which will decrement
                           // the Ref Count on modDisp by two.
```

SafeArrays

C++ programmers who use API functions with SafeArrays, must be careful when managing SafeArray memory. If you receive a SafeArray from SolidWorks, then you are responsible for destroying it. Also, if you are passing a SafeArray to SolidWorks, the SafeArray will not be destroyed by SolidWorks and you are again responsible for destroying it.

Return Values

All C++ COM implementations will have an HRESULT return value. The API Help documentation will only show the COM syntax when necessary (see **Programmer's Guide** on page B-8 for more details). If you are using COM, it is implied that the SolidWorks API function will return an HRESULT and that any additional return values should be passed by reference as arguments.

HRESULT return values are used by SolidWorks to indicate that the code was called successfully. It is not meant as an indication that your call achieved its objective. For example, calling IGetFirstFace will have an HRESULT return value of S_OK if the code for IGetFirstFace was called successfully. It does not mean that the code succeeded in finding the first face. In this case, you should check the LPFACE return value for a NULL condition.

If an API method returns an object, it is always a good idea to verify that the object returned is not NULL. This type of error checking is good programming practice and will avoid crashes in your code for unexpected conditions.

For example:

```
LPFACE m_Face = NULL;
HRESULT hres = S_ERROR;
hres = m_Body->IGetFirstFace( &m_Face );
if (hres != S_OK || m_Face == NULL)
{
AfxMessageBox( _T("Error in call to IGetFirstFace.") );
return;
}
```

This type of error checking is also a good habit for Visual Basic programmers. For example:

```
Dim FaceObj As Object
Set FaceObj = BodyObj.GetFirstFace
If FaceObj Is Nothing Then          // See if we found a Face
Msg = "Error in call to GetFirstFace"   // Define message
Style = vbOKOnly                     // Show OK button only
Title = "Error !"                    // Define title
Call MsgBox(Msg, Style, Title)
Exit Sub                             // If no face, then exit
End If
```

For more information about NULL `Variant` return values, see **Helpful Hints** on page B-10.

SolidWorks API Objects

Accessing Objects

To call any of the methods or properties that are found in the API, you first need to obtain the object. For example, to use the `Face::GetArea` method you need to have a Face object.

There are many different ways to access objects within the SolidWorks API. For example, to obtain the Face object mentioned above, you could:

- Obtain the Body object using `PartDoc::Body` and then traverse the faces on the body using the `Body::GetFirstFace` and `Face::GetNextFace` methods.

- Or you could get the Face object from the current set of selected items using `SelectionMgr::GetSelectedObject`.

- Or maybe you prefer to get the face object by its name using `PartDoc::GetEntityByName`.

In either case, once you have the Face object, you are then able to access the properties and methods found in the Face Class. For example, once you have the Face object, you could get the number of edges on the face using the `Face::GetEdgeCount` method, or you could get the normal vector for the face using the `Face::Normal` property.

Do not get confused with API methods and properties which also require a selected item, such as `AssemblyDoc::OpenCompFile`. To call `OpenCompFile` you need to have the `AssemblyDoc` object AND have a component selected. The component can be selected in one of two ways:

- The user can interactively select the component.

- Or you can programmatically select the component using `ModelDoc::SelectByID`.

For more information on object relationships, please refer to the API Object Diagram in the API documentation located in the **...\Samples\Appcomm** subdirectory of your SolidWorks installation. Double-click on **API_help.hlp**.

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