SolidWorks® 97Plus User's Guide

Document Number: UG0797-E

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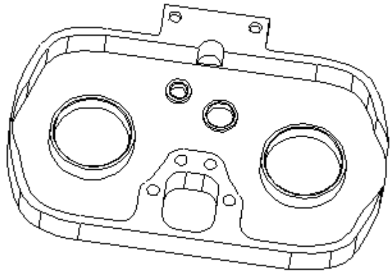
SolidWorks Fundamentals

What is SolidWorks 97Plus?



SolidWorks® 97Plus 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.

This chapter discusses some basic concepts and terminology used throughout the SolidWorks 97Plus application. It provides an overview of the following topics:

- ❑ Installing the SolidWorks 97Plus Software
- ❑ SolidWorks Terms and Document Windows
- ❑ FeatureManager™ Design Tree
- ❑ Right-Mouse Menus
- ❑ Selecting Items
- ❑ Toolbars
- ❑ Time Saving Tips
- ❑ Finding Out What is Wrong
- ❑ Rendering Models With PhotoWorks
- ❑ Using the SolidWorks API Development Environment

Installing the SolidWorks 97Plus Software

System Requirements

- Microsoft Windows NT 4.0 or Windows® 95
- Pentium®-based computer, or DEC AlphaStation™
- 32 MB RAM minimum, 48 MB or more recommended
- Mouse or other pointing device
- CD-ROM drive

Required Information for Installation

The SolidWorks 97Plus 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.

  - **Server Only.** This installation should be selected for a computer that will *not* run the SolidWorks 97Plus 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.** Your serial number is located on the SolidWorks Registration Form and the back of the CD-ROM case.
- **SolidWorks Registration Code.** You will receive your registration code when you register your software. (You can proceed with the installation and use the SolidWorks application until you receive the registration number.)

- **PhotoWorks Serial Number.** If you purchased the PhotoWorks photo-realistic rendering application, you need to enter the PhotoWorks serial number, located on the SolidWorks registration form and 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 make changes to a drawing without having those changes reflected back to the model. (You must reinstall the SolidWorks software to change this option.) The default, and most usual configuration is to allow model changes when the drawing is changed.

Installing the SolidWorks 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 for Windows 95 or Windows NT 4.0:

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 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.

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

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 or Windows 95: **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.

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:

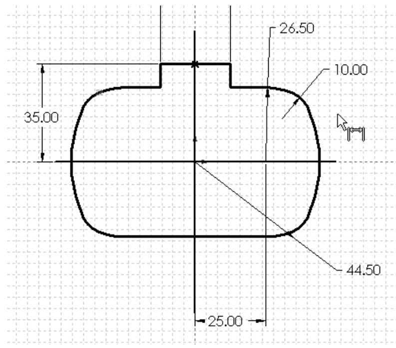
1. Click **Help, About SolidWorks 97Plus**.
2. Click the **Connect** button to visit the SolidWorks web site.

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

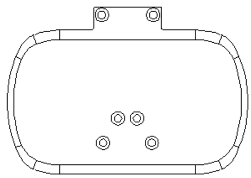
Basic Concepts

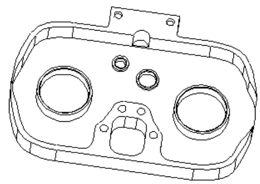
❑ **A SolidWorks model consists of parts, assemblies, and drawings.**

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

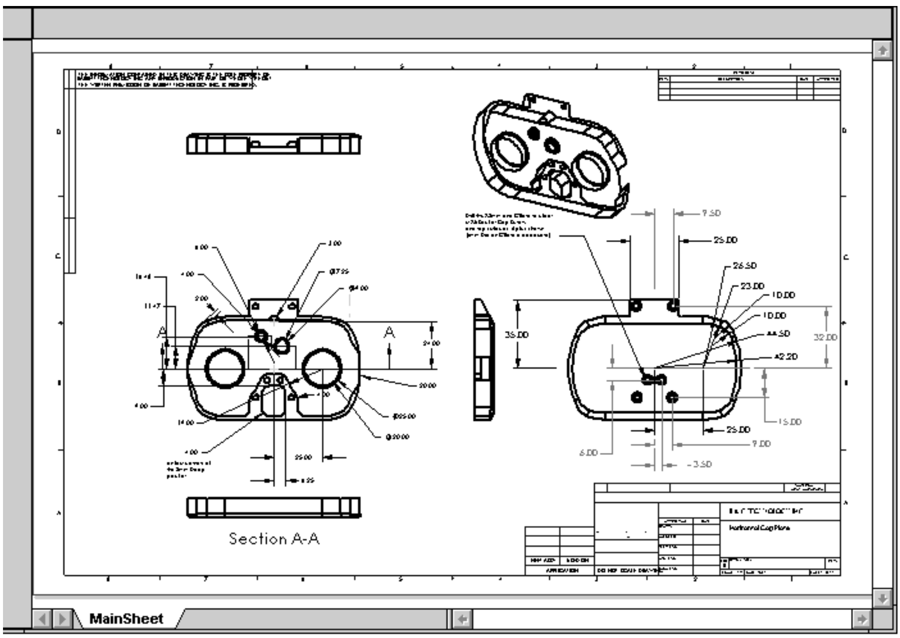


- You are free to refine your design by adding, changing, or reordering features.
- Associativity between parts, assemblies, and drawings assures that changes made to one view are automatically made to all other views.





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



- **The SolidWorks application lets you customize functionality to suit your needs.**

Click **Tools, Options** on the top menu bar to display tabbed pages of options for your selection. All of these options pages are described in detail in Appendix A. What follows is just a few of the ways in which you can customize the SolidWorks application:

Option page name:	Available options on page:
Color	Colors for lines, faces, features, and drawing and view borders
Crosshatch	Crosshatch pattern and its scale and angle
Detailing	The dimensioning standard, detailing options such as arrow, witness line, or balloon styles, and font size and style
Drawings	Drawing sheet size, template, scale, projection type, model edge display, and inferencing behavior when dragging detail items
Edges	Display of various edge types, tangent edge display, repainting and highlighting behavior
External References	Preferences for the way part, assembly, and drawing documents are opened and referenced
General	Optional behaviors when in a sketch or model, FeatureManager design tree behavior, angle increment and speed of view rotation, and behavior of overdefined dimensions
Grid/Units	Grid display, spacing, and snap behavior; unit types (inches, feet, millimeters, or centimeters); spin box increments
Import/Export	Settings for IGES, Parasolid, STL, DXF/DWG, and ACIS; trim curve accuracy; optional output as trimmed surfaces or 3D curves
Line Font	Style, and weight of edge lines for selected edge types
Material Properties	Material property for the current part (only available when a part document is active)
Performance	Display quality for shaded, wireframe, and transparent models
Planes	Default names for planes in part or assembly documents

- **SolidWorks online help provides assistance while you are working.**

Click  or the **Help** button 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.

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

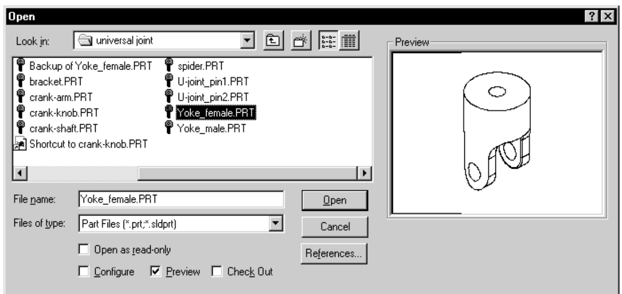
To open a *new* part, drawing, or assembly document:

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 4-8 for more information.



5. Select **Check Out** to allow other users to access the document at the same time. When you save the document, you receive a list of changes made by other, simultaneous users.

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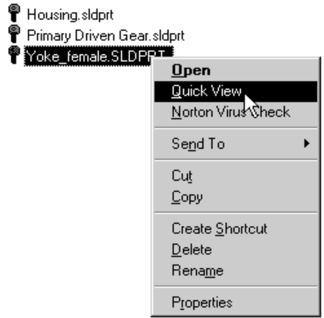
1 In this manual, “Click...” means point the cursor and press the *left* mouse button

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**.

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

Opening an Existing Part from Explorer

You can preview and open a part, drawing, or assembly document directly from the Microsoft Explorer.

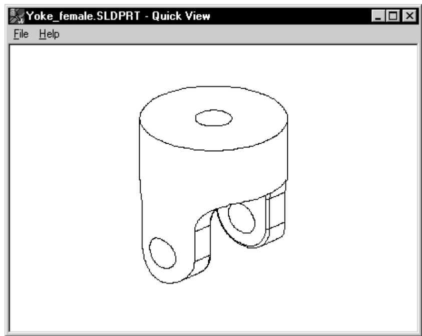


To view the part, drawing, or assembly without opening the document:

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.

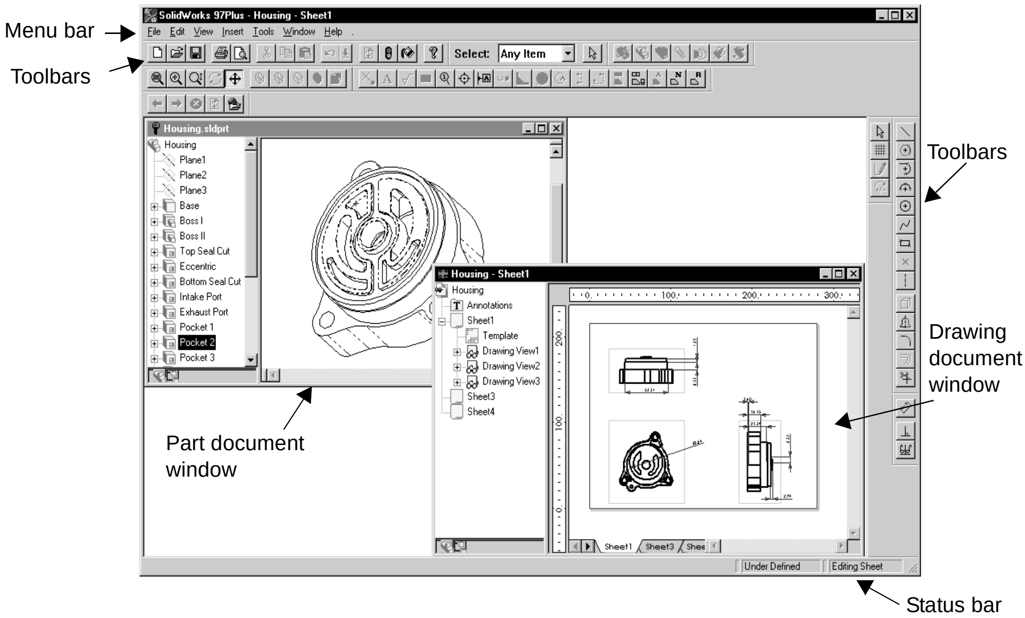


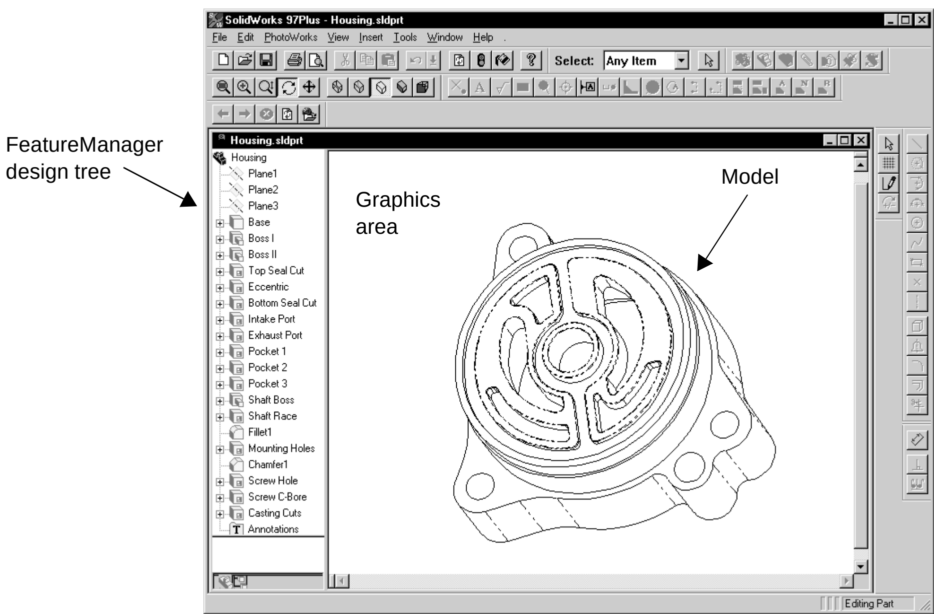
To open the part, drawing, or assembly from Explorer:

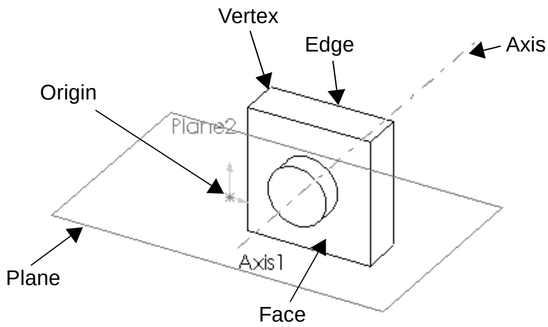
- ❑ Double-click on the name of the part, drawing, or assembly in Explorer.
- ❑ Right-mouse click on the name of the part, drawing, or assembly in Explorer and select **Open** from the right mouse menu.
- ❑ Drag and drop a part from Explorer to an open SolidWorks assembly window or an open, empty SolidWorks part window.
- ❑ Drag and drop an assembly from Explorer to an open SolidWorks assembly window.

SolidWorks Terms

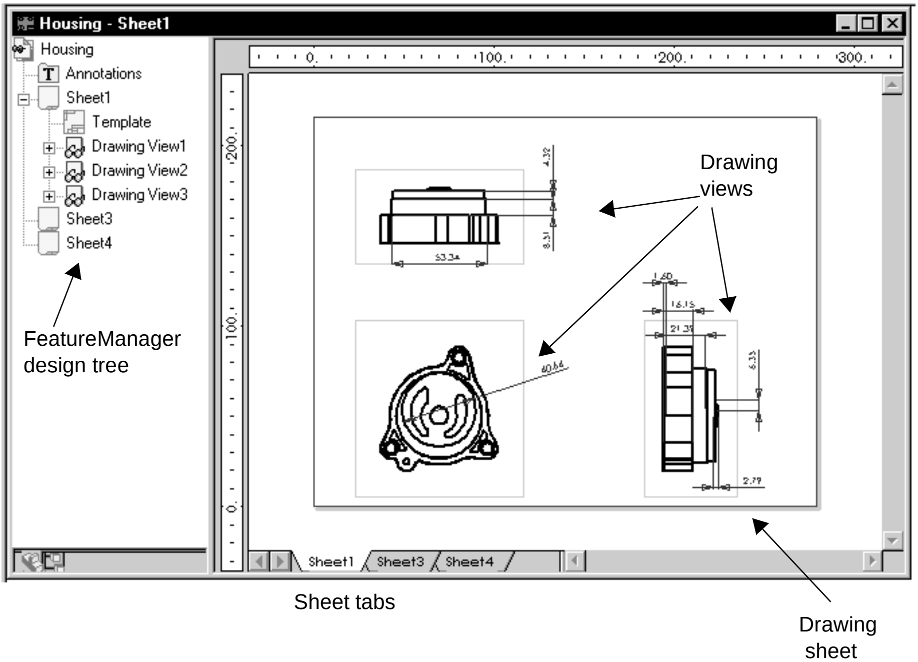
This section illustrates some common SolidWorks terms.







Vertex
Edge
Axis
Origin
Plane2
Plane
Axis1
Face



Housing - Sheet1
Housing
Annotations
Sheet1
Template
Drawing View1
Drawing View2
Drawing View3
Sheet3
Sheet4
FeatureManager design tree
Drawing views
Sheet1
Sheet3
Sheet4
Sheet tabs
Drawing sheet

Document Windows

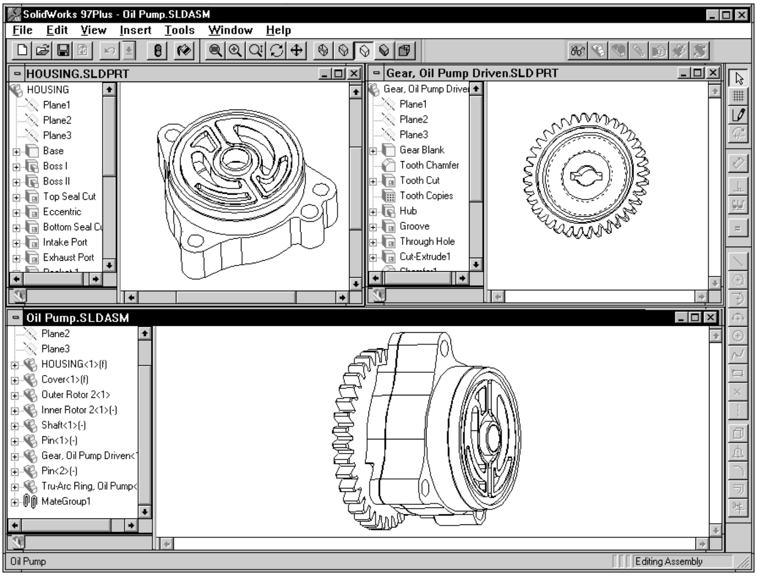
In the SolidWorks application, each part, assembly, and drawing is referred to as a document, and each document is displayed in a separate window. (Each drawing document can contain multiple drawing *sheets*, though.)

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.

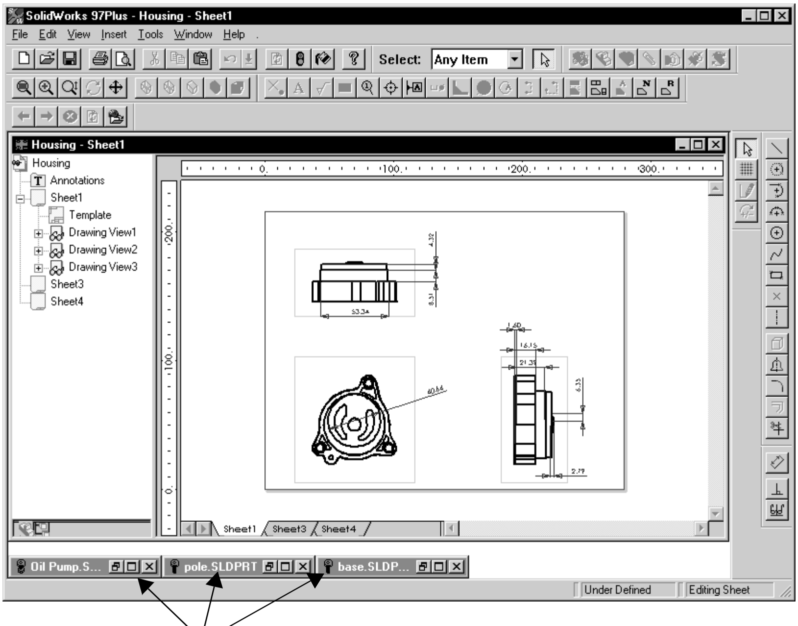
To arrange the windows, you can drag them by the title bar, and resize them by dragging a corner or border. Also, from the **Window** menu, you can select **Cascade**, **Tile Vertically**, or **Tile Horizontally**.

Part windows

Assembly window



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.



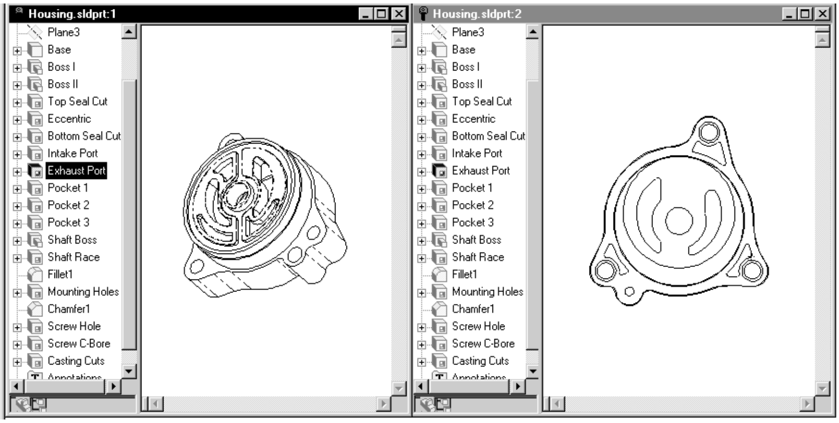
Document icons

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

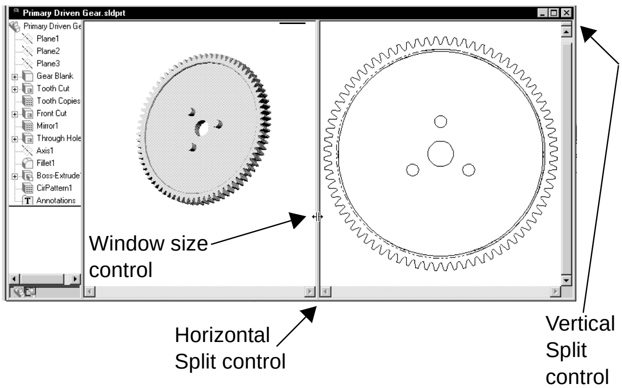
Multiple Views

You can open additional views of the same document.

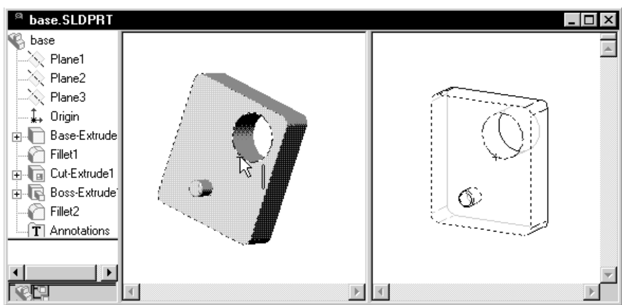
❑ Click **Window, New Window** to open another view in a new window.



❑ 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.



Selecting an item in one view will select 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.



Using Named Views

You can display a part or assembly using named views.

- ❑ You can use the view list (displayed with the **Orientation** command on the **View** menu) to switch quickly between named views. Double-click the name of the view you want to see.
- ❑ You can rotate the model or change the zoom, then give the current view a name and add it to the view list. Click **Add**, and give the view a meaningful name. Then click **OK**. The new name will be added to the **View Orientation** list where you can double-click on it when you want to return to this view.
- ❑ Because the **View Orientation** dialog box is a useful tool, you may want to keep it available while you work. Drag the dialog box to a convenient location on your screen and click the *push pin* icon. This keeps the list open and on top of all other windows.

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

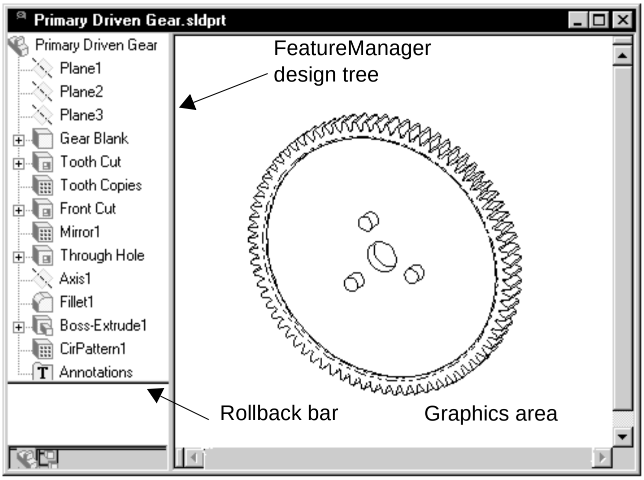
- ❑ You can return your model to a previous view by clicking the **Undo** button on the **View Orientation** dialog box. You can undo the last 10 view changes.

- ❑ You can change the orientation of all the standard views. Rotate or zoom a view, then click (not double-click) the name of the standard view you want to assign to the current orientation of the model. For example, click **Front** if you want the current view to become the front. Click **Update**. This updates all of the standard views so they are relative to this new view.
- ❑ Click the **Reset** button to return all standard views to their default settings.

The FeatureManager Design Tree

SolidWorks document windows have two panels:

- A *FeatureManager design tree*, which lists 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.
- Identify and change the order in which features are created.
- 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.
- Suppress or hide selected features.
- Temporarily roll the model or assembly back to an earlier state using the *rollback* bar.

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 overdefined; they are preceded by **(–)** if they are underdefined; they are preceded by **(?)** if the sketch could not be solved. For more information about overdefined and underdefined sketches, see **Fully Defining Sketches** on page 2-28. (There is *no* prefix if the sketch is fully defined.)
- Assembly components in the FeatureManager design tree are preceded by **(+)** if their position is overdefined; they are preceded by **(–)** if their position is underdefined; 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 overdefining 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.

Views and Options

❑ In an assembly, you can select alternate displays of the FeatureManager design tree by clicking **View, FeatureManager Tree** and choosing either:

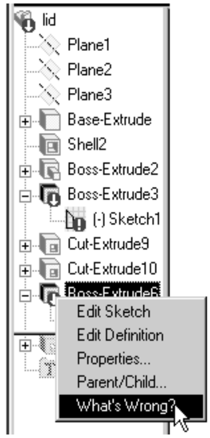
- **By Features,** to display all components, features, planes, and mate groups.
- **By Dependencies**, to display only the top level components and mate groups.

❑ 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-10.) 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. (See **General Options** on page A-10.)

- **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?

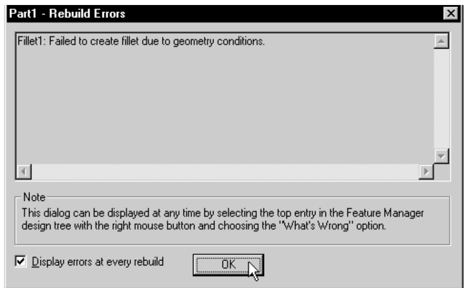


The SolidWorks 97Plus application offers a "What's Wrong" functionality, that allows you to view any regeneration errors of a part or assembly. Down arrows appear in the FeatureManager design tree next to the name of the part or assembly and the name of the failed feature. An exclamation mark (**!**) 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:

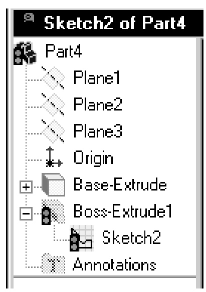
- ❑ Dangling Dimensions or Relations – dimensions or relations to an entity that no longer exists
- ❑ Features that cannot be regenerated, such as a fillet that is too large



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 **Rebuild Errors** 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.

The rebuild symbol also displays when you edit a sketch; when you exit the sketch the part rebuilds automatically.

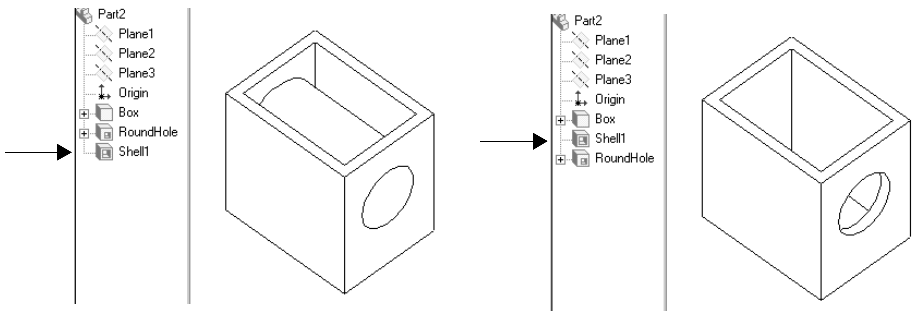
Drag and Drop

SolidWorks supports several drag and drop operations.

- **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.

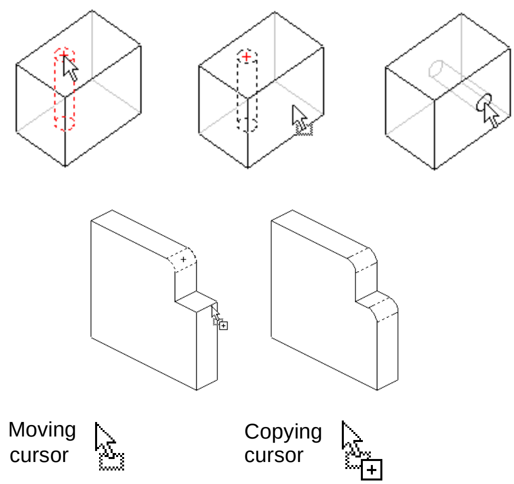
  In the following example, the **RoundHole** was cut before the **Shell** feature was added. Later, **Shell** was dragged and dropped before **RoundHole**.



- **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.

  Or, 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.



- **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.

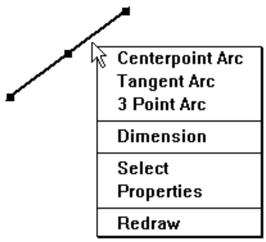
Note: When you drag into another document, the item is always *copied*, not moved.
When you drag within a document, the item is *moved* by default, unless you hold the **Ctrl** key while you drag.

Right-Mouse Menus

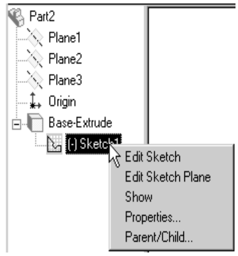
Whether you are working with a sketch, a model, an assembly, or a drawing, you have access to a wide variety of tools and commands from the right mouse menu. This gives you an efficient way to do your work without continually moving your cursor to the main pull-down menus or the toolbar buttons.

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, you can



- Right-click on a sketch entity and select another 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.



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 **Options, Edges Options** on page A-7.

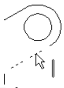
- **Selecting with the Selection Filter.** To make it easier to select specific items, you can set the **Selection Filter** to the kind of item that you want to select: **Faces**, **Edges**, **Axes**, **Planes**, or **Any Item**. With the filter set, the specified kinds of items are highlighted when you pass your cursor over them. This makes it easy for you to select only the items that you intend to select.

  Note: If it is not visible, you can show the **Selection Filter** by clicking **View, Toolbars** and choosing **Selection Filter** from the list.

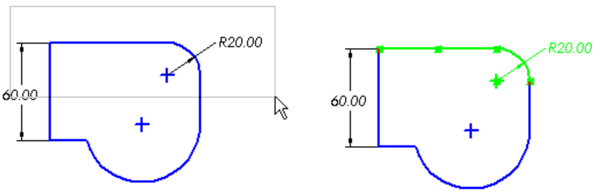
- **Face and Edge Selection Cursors.** To assist you in selecting items, the cursor symbol changes to indicate the kind of item that the cursor is currently pointing to. When you pass the cursor over a face, the cursor symbol is shaped like a flag; when it is over an edge, the symbol is a vertical bar.

  
  Face Selection

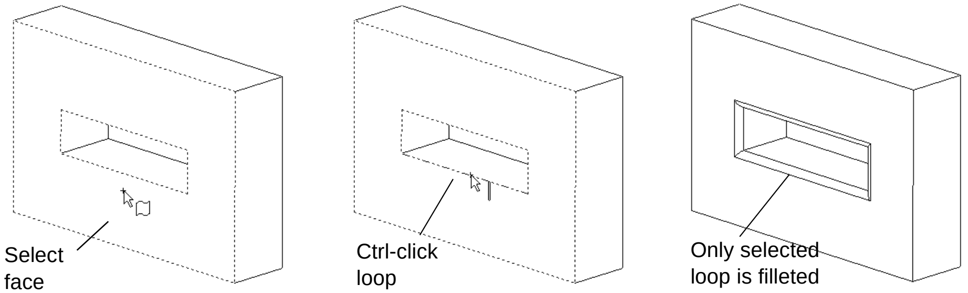
  
  Edge Selection

- **Selecting multiple items.** To select more than one item, hold the **Ctrl** key while you click on the items.

- **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.

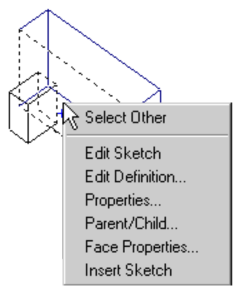
- **Selecting loops.** If you select a face, all *loops* (closed collections of connecting edges) on the face are selected, too. To select individual loops, click on the face, then hold down the **Ctrl** key and click on the loops you want to select. (This will deselect the face and leave the loops selected.)



- **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 or **Ctrl**-clicking in the model. To start over, right-mouse click the model and choose **Clear Selections**.

Selecting Hidden or Coincident Items

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

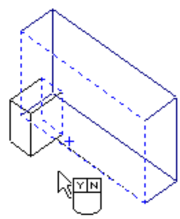


To select a hidden item:

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.

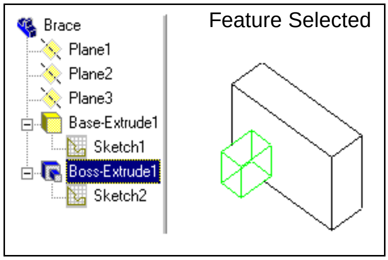


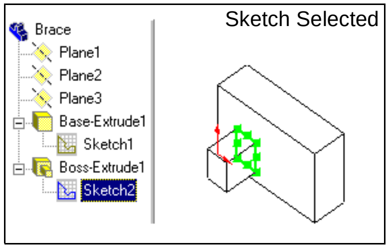
3. When the item you want is highlighted, click the *left* button (**Y**) to select that item.

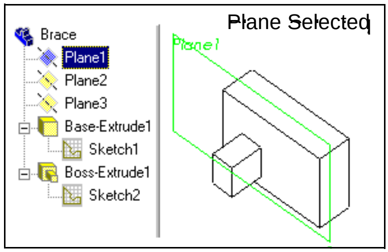
Selecting from the FeatureManager Design Tree

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

- ❑ You can select features, sketches, planes, and axes in the model by clicking on their names in the FeatureManager design tree.

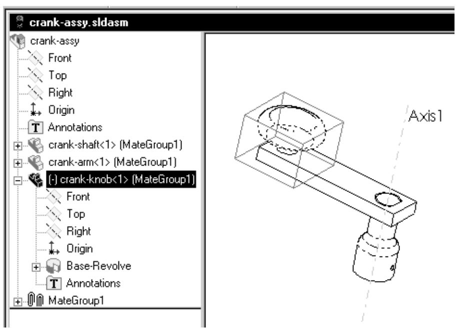






- ❑ You can select multiple consecutive items in the FeatureManager design tree. **Shift-**selection selects all consecutive items between the first and last selected items.

- ❑ You can select from the FeatureManager design tree in an assembly, as well. In this example, a component is selected and its bounding box is displayed

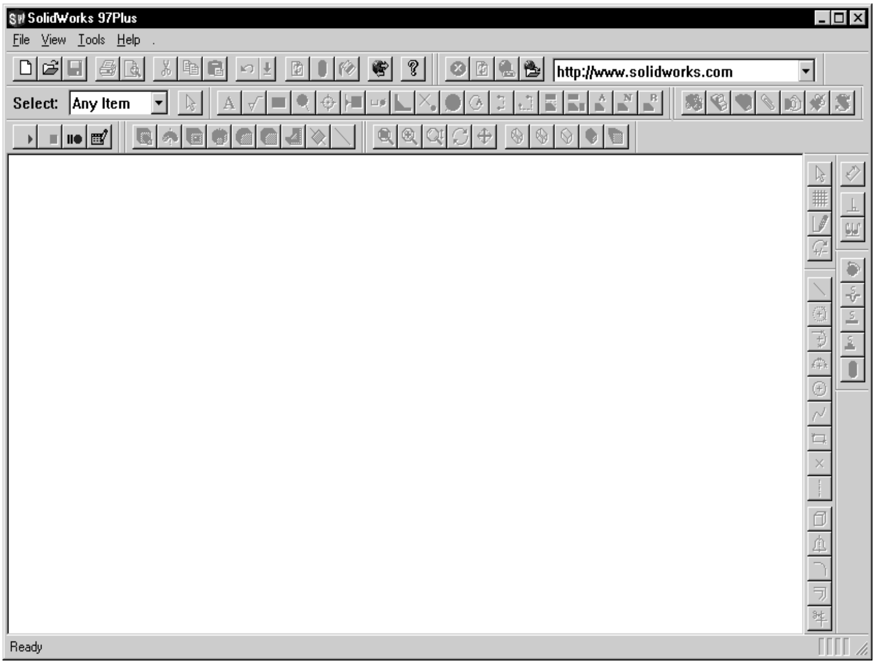


Note: You can make selections by clicking items in the graphics window or the FeatureManager design tree, whichever is more convenient.

Toolbars

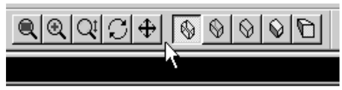
The toolbar buttons are shortcuts for frequently used commands. All the available toolbars are displayed in this illustration of the SolidWorks initial screen, but your SolidWorks window probably will not be arranged this way. You can customize your toolbar display in a way that is convenient for you.

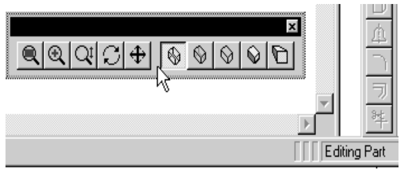
Some toolbars display automatically when you open a document. For example, when you open an assembly document, the assembly toolbar displays.



- ❑ Click **View, Toolbars** to
  - Display or hide individual toolbars or the selection filter
  - Display the toolbar icons in a larger size
  - Display or hide the tooltips that pop up when the cursor moves over an icon

Arrange and Customize Toobars

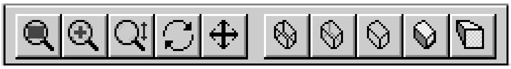


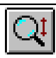
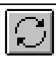


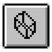
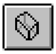
- ❑ Arrange the toolbars by clicking on the space *between* the buttons and dragging. If you drag a toolbar to an edge of the SolidWorks window, the toolbar docks to the edge automatically.
- ❑ If you drag a toolbar away from the window edge, it becomes a floating palette.
- ❑ To customize toolbars, you can
  - Drag buttons from one toolbar to another or to a new location on the same toolbar
  - Delete buttons by dragging them from the toolbar to the graphics area
  - Click **Tools**, **Customize** to select buttons to duplicate on more than one toolbar

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.)



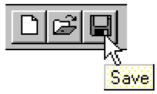
Icon	Description
 **Zoom to Fit**	Rescales the view so the entire part, drawing, or assembly is visible.
 **Zoom to Area**	Zooms in on a portion of the view that you select by creating a bounding box. 1. Place the cursor where you want the one corner of the box to be. 2 Drag the cursor diagonally to 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.
 **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. 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. You can 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.
	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.

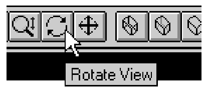
Note: For information about other SolidWorks toolbars, click **Help** to access online help. In the help **Contents**, select **SolidWorks Basics**, and click **Toolbars**.

Time Saving Tips

The following tips may help you save time.



- **Save your work often.** SolidWorks files are very compact and take very little time to save. Press **Ctrl+S** or click **Save** frequently.
- **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.

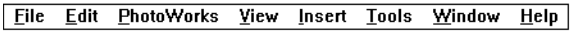
- **Use the modify spin box as a calculator.** Enter values and arithmetic symbols directly into the spin box to calculate the dimension.

- **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.

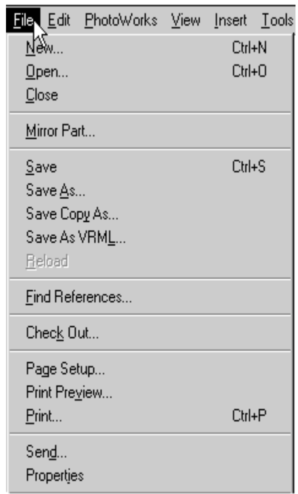
  Note: You can hide or display the status bar. Click **View, Status Bar**. A checkmark next to the menu item means the status bar is visible.

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.

Note: You can customize the keyboard shortcut keys to suit your style of working. Click **Tools, Customize**, and select the **Keyboard** tab. You can assign new shortcut keys, remove shortcut keys, or reset the shortcut keys to their original setting.

The following table lists the 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**

Rendering Models with the PhotoWorks Application

PhotoWorks is a photo-realistic rendering application, available through the SolidWorks 97Plus application, that lets you create realistic images directly from SolidWorks models.

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, and logos.

You can find detailed instructions on how to use the PhotoWorks application in *Learning to Use SolidWorks 97Plus*, Chapter 16.

SolidWorks API Development Environment

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, 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 (www.solidworks.com) under the Technical Support area.

2

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 “45-Minute Running Start” in the *Learning to Use SolidWorks 97Plus* tutorial. Creating the parts in the rest of the tutorial chapters will acquaint you with many sketching techniques, as well.

This chapter describes:

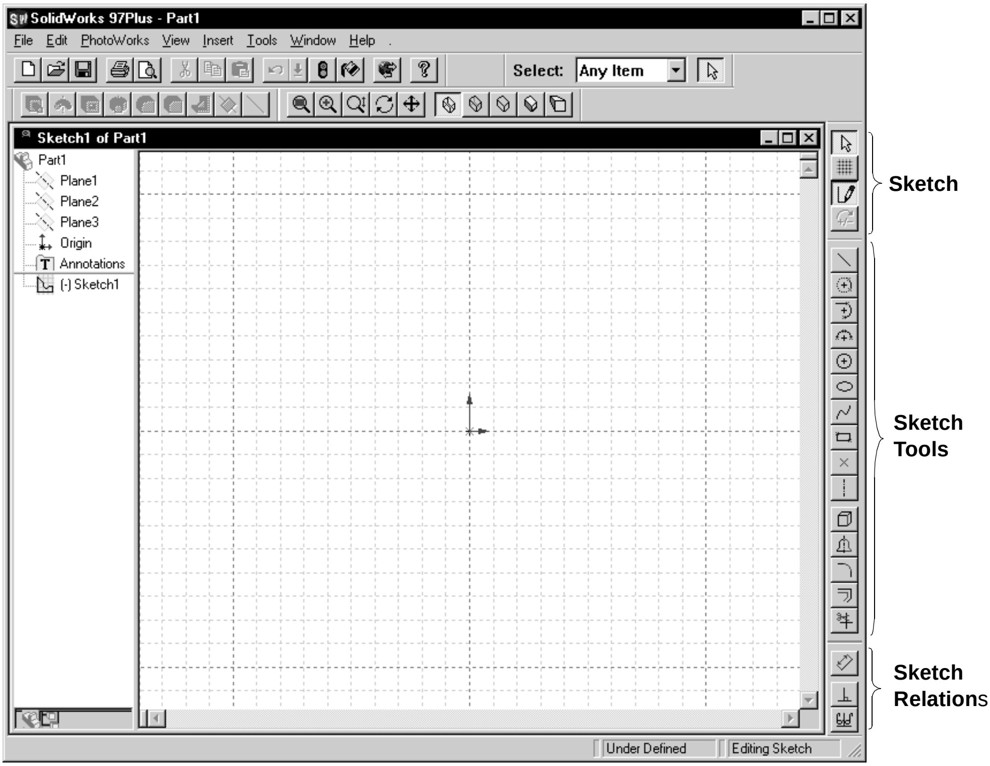
- Starting a new sketch
- Accessing and using the sketch tools
- Dimensioning the sketch
- Adding and deleting geometric relations
- Fully defining sketches
- Taking advantage of inferencing
- Using the right-mouse button
- Extruding from a sketch
- Sketching on the face of a part
- Editing and exiting a sketch
- Creating and using construction planes

Starting a New Sketch

To open a new sketch, you can either

- Click the **Sketch** tool  , 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.



When creating sketch entities or dimensions, you may find it more convenient if the tool you select remains active to create multiple entities with a single click. To specify this behavior, click **Tools, Options**, and select the **General** tab. If you check the **Single command per pick** checkbox, the sketch and dimension tools deselect after each use; if you uncheck this option, the tools remain active until you select a different tool.

For information about other options that you can select to customize the sketching functionality, refer to **General Options** on page A-10.

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 Entity** or **Offset Entity**
- 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 **Tools, Options**, and select the **Grid/Units** tab. See **The Sketch Grid** on page 2-31.

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 2-15.

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 for the area 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 an ellipse from a centerpoint, start point, and end point (menu item only).

Spline creates a spline curve.

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 2-9.

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 6-45.

For descriptions of creating and using sketch entities, see **Using the Sketch Entities** beginning on page 2-6.

To use sketch entities as construction geometry, see **Creating an Axis** on page 2-39.

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 in an active part or assembly document only.)

Mirror creates copies of sketch elements by are mirroring them around a centerline. When you create mirrored elements, a *symmetric* relation is applied between corresponding pairs of 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, 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 in an active part or assembly document only.)

Trim/Extend trims or extends a sketch element.

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 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.

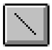
For descriptions of using the Sketch Tools, see **Using the Sketch Tools** beginning on page 2-11.

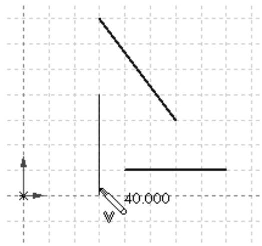
Using the Sketch Entities

This section discusses how to create and use sketch entities.

Line

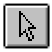
To sketch a line:

1 Click the **Line** tool. 

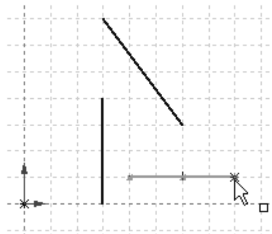


2 Press the left mouse button where you want to begin sketching the line and drag the cursor.

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. 

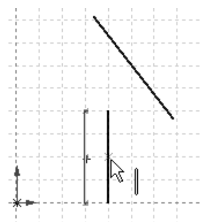
To change the length of the line:



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

2 Drag to lengthen or shorten the line.

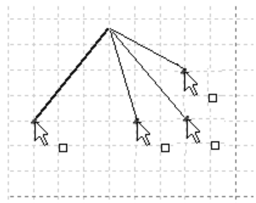
To move the line:



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:

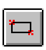


1 Point the cursor to an end point and press the left mouse button.

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

Rectangle

To create a rectangle:

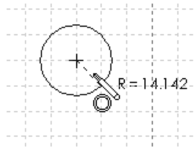
1 Click the **Rectangle** tool. 

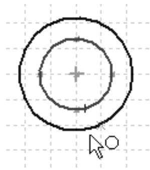
2 Place the cursor where you want one corner of the rectangle to appear.

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

Circle

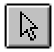
To sketch a circle:





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:

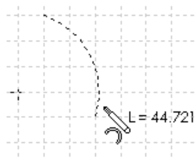
Point the cursor at the circle center and drag to another position.

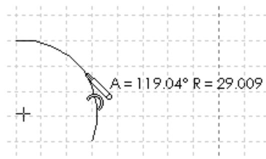
To change the radius:

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

Arcs

To sketch an arc through three points:





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

2 Point where you want the arc to start.

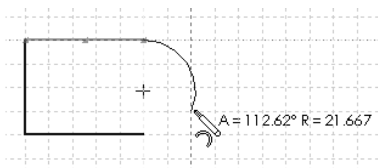
3 Press the mouse button and drag to where you want the arc to end.

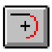
4 Release the mouse button.

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

6 Release the mouse button.

To sketch an arc, tangent to a sketch element:



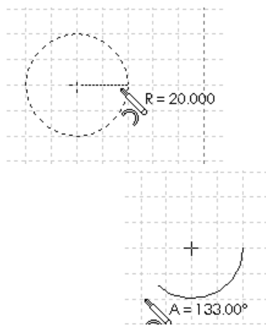
1 Click the **Tangent Arc** tool. 

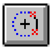
2 Press the mouse button over the end point of a line, arc, or spline.

3 Drag out the arc to the desired shape.

4 Release the mouse button.

To sketch an arc 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.

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.

Spline

To create a spline:

1. Click the **Spline** tool. 
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:

1 Click the **Select** tool.

2 Click on the spline; handles appear on the segment end points.

3 Drag the handles to reshape the spline.

Note: The default open spline is *proportional*; when you drag the endpoints, the spline retains its shape. To change the spline's shape, or to add relations or dimensions to interior points on the spline, remove the proportional property. Right-mouse click the spline and select **Properties**. Click the **Proportional** checkbox to remove the check.

Point

To create a sketch point (reference point):

1 Click the **Point** tool.

2 Click where you want to create the sketch point.

Centerline

To create a centerline:

1 Click the **Centerline** tool.

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

Creates text on a part as either an extruded boss or a cut.

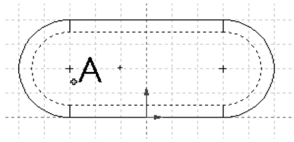
Note: The text aligns with the x-axis of the sketch plane and proceeds in the direction of the increasing x value.

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.

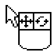
4 Type your text into the edit box.

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.

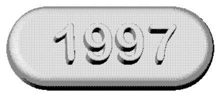
8 To change the orientation or location of the text before you extrude it,

a) Exit the sketch and click **Tools, Sketch Tools**, **Modify Sketch** or . The cursor changes to the shape of a mouse. The left mouse button moves the text, the right mouse button rotates the text. 

b) Use the mouse to position the text.

c) When you are finished, click **Tools, Sketch Tools**, **Modify Sketch** or again.

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 With the sketch closed, select the text sketch in the FeatureManager design tree.

2 Click **Tools, Sketch Tools**, **Modify Sketch** or . (See **Modify Sketch** on page 2-15.)

3 Using the mouse, position the text.

4 Click **Tools, Sketch Tools**, **Modify Sketch** or to exit.

5 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 2-21.

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**.

A relation is created between the new sketch curve(s) and the selected entity (entities), so the curve(s) will be updated if there are changes to the entity (entities).

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 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.

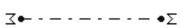
2 Hold down the **Ctrl** key and select the centerline and the items you want to mirror.

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**. M's 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.

To create a fillet in a sketch:

1. Click  or **Tools, Sketch Tools, Fillet**.
2. Select the sketch elements to fillet. (The elements must form a corner.)
3. Enter a fillet radius value.
4. Click **Apply**.

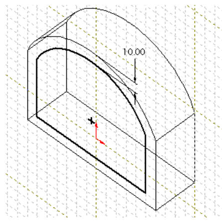
The Sketch Fillet dialog box remains open so you can continue creating fillets until you close the dialog box.

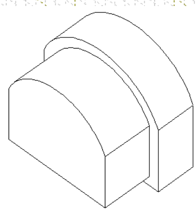
Offset Entities

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 2-21.

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  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/Extend

Use the **Trim/Extend** tool to:

- Trim a line, arc, ellipse, or circle to its intersection with a line, arc, ellipse, circle, spline, or centerline.
- Extend a line or arc to meet a line, arc, ellipse, circle, spline, or centerline.
- Delete the portion of a line that lies between two intersecting lines.

To trim a sketch element:

1 Click the **Trim/Extend** tool. 

2 Click the portion of the sketch element you want to remove.

To extend a sketch element:

1 Click the **Trim/Extend** tool. 

2 Click on an endpoint of a sketch element that you want to extend (a line, arc, or centerline).

3 Drag the endpoint to a second sketch element. When the second element is highlighted, release the mouse button; the first element extends to meet the second.

To extend one or two endpoints – a shortcut:

1 Click the **Trim/Extend** tool. 

2 Hold the **Ctrl** key and click the sketch element you want to extend.

The sketch element extends to intersect with the next sketch element, if one is available. It extends both ends of the sketch element if there are possible intersections at both ends.

If there are other possible intersections with additional sketch elements beyond the first intersection, repeatedly **Ctrl**-clicking the extending sketch element causes it to continue to grow until there are no more sketch elements available with which it can intersect.

Detach Segment On Drag

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

To detach a sketch segment:

Click **Tools, Sketch Tools, Detach Segment On Drag** and select the entity that you want to drag away from another sketch entity.

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

Note: **Detach Segment On Drag** does not delete any relations that the sketch entity may have, so it is possible that the entity may not drag or may not drag smoothly. If this occurs, you should delete any relations before you drag/detach the entity. (See **Display/Delete Relations** on page 2-26.)

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. In a part sketch, click **Tools, Sketch Tools, Override Dims on Drag**.

   When **Override Dims on Drag** has a check mark next to it, you can drag sketch entities and the dimensions will update automatically in the part, assembly and drawing.

2. 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, 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.

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.**

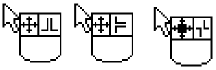
– or –

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

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

b) Press the left mouse button to move the sketch.

c) 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.



d) 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 external references. (The mouse cursor has a **?** on the left button.)

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:

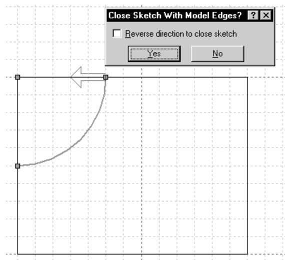
1 Open a sketch on a model face.

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

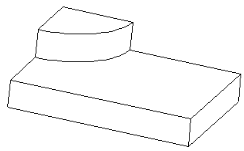
3 Click **Tools, Sketch Tools, Close Sketch to Model**.

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

Notice that an arrow points in the direction in which the profile will close. (Refer to the illustration.)



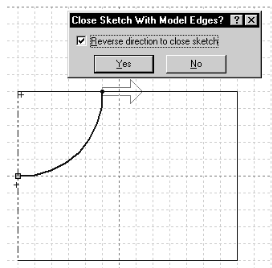
4 Click **Yes** when the arrow points in the correct direction; click **No** to exit the dialog without specifying a direction.



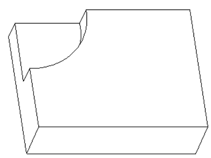
5 Click **Insert, Boss, Extrude** and specify the **End Conditions** in the **Extrude Feature** dialog box (as illustrated),

– or –

Click **Insert, Cut, Extrude** and specify the **End Conditions** in the **Extrude Feature** dialog box.



6 Click **OK** to make the extrusion.



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 2-18.

Add Relations creates geometric relations (like tangent or perpendicular) between sketch elements, or between sketch elements and planes, axes, edges, or vertices. See **Geometric Relations** on page 2-21 and **Add Relations Table** on page 2-24.

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.

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 2-27.

Constrain All sets constraints on the sketch entities of an unconstrained imported .DXF or .DWG drawing. See **Constrain All** on page 2-27.

Dimensioning Sketches

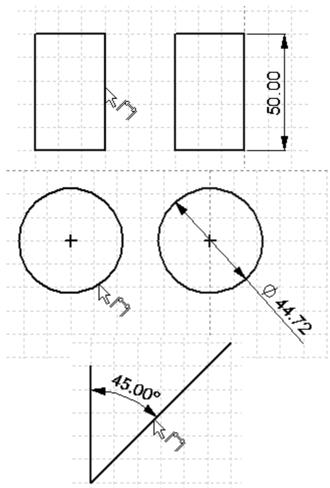
Usually, you dimension your sketch as you go along. However, it is not *necessary* to dimension sketches before you use them to create features. When you add or change dimensions in a sketch, the changes are reflected in the part and the drawing views of the part. When you add dimensions to a drawing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

To dimension a sketch:

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

Note: If you click the icon, , the default dimension is **Parallel**. Press the right mouse button to select either **Horizontal** or **Vertical,** whichever dimension is most appropriate for the sketch entity you are selecting.

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



Dimensioning with Equations

You can insert dimensions using mathematical relations between dimensions or parameters, using dimension names as variables. When using equations in an assembly, you can set equations between parts, between a part and a subassembly, with mating dimensions, and so on.

Note: Dimensions driven by equations cannot be changed by editing the dimension value in the model.

To add an equation:

1 Click **Tools**, **Equations**.

2 Click **Add**.

3 In the model or the FeatureManager design tree, double-click on the feature that contains the first dimension you want to use in the equation.

4 Drag the dimension to the equation to paste its name into the equation. (Dimension names are in the form ***dimension name @ feature*** or ***sketch name***.)

5 Complete the equation by typing or clicking on the calculator buttons, or by dragging other dimensions to paste their names.

Equations are solved left to right (i.e., the dimension on the left is driven by the value on the right), in the order in which they appear in the equation list.

6 Click **OK**. The equation will appear in the **Equations** window.

7 Click **OK**, then click  or **Edit**, **Rebuild** to update the model. (All equations are solved before the geometry is regenerated.)

Note: When using any trig function in an equation or dimension dialogue, the value of the angle is interpreted as radians. For example, sin(90) is evaluated as .89 (90radians) not 1.0 (90deg).

To edit equations:

1 Click **Tools**, **Equations**.

2 Click **Edit All**.

3 Edit the equations. Each equation must be on a separate line, and dimension names must be enclosed in quotes.

4 Click **OK** to close the **Edit Equations** window.

5 Click **OK** to close the **Equations** window.

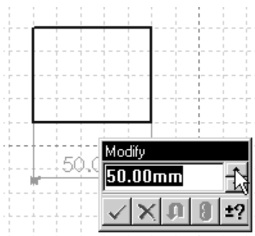
6 Click  or **Edit**, **Rebuild**.

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.)

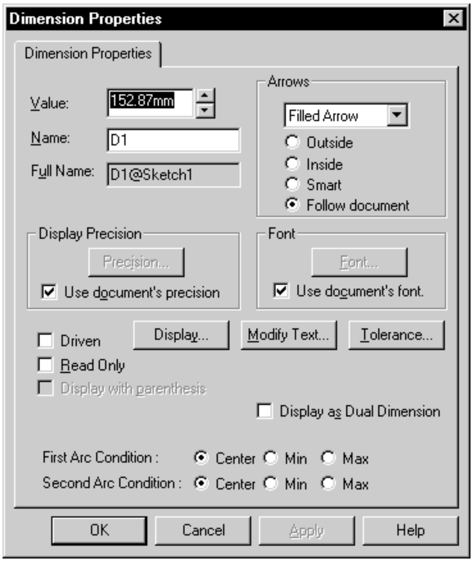
 Reset the spin box increment value.

Dimensioning Arcs and Circles

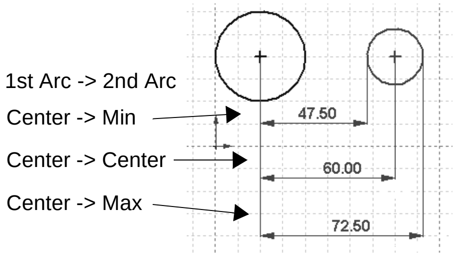
By default, when you dimension to an arc or circle, the dimension goes to the centerpoint of the arc or circle. You can edit the dimension so that it measures to the edge of an arc or circle.

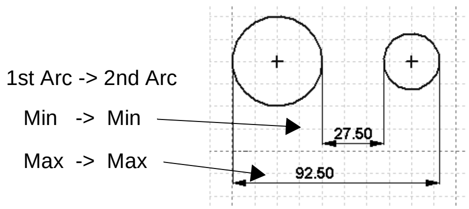
To change dimension options for arcs and circles:

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.





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 silhouette** option. Click **Tools, Options**, and select the **General** tab. In the **Sketch** block 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), drag your cursor over a cylindrical face until the cursor changes to include a small cylinder next to the pointing arrow. At this time, the cylinder's silhouette edge becomes visible as a dashed line.

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.

Geometric Relations

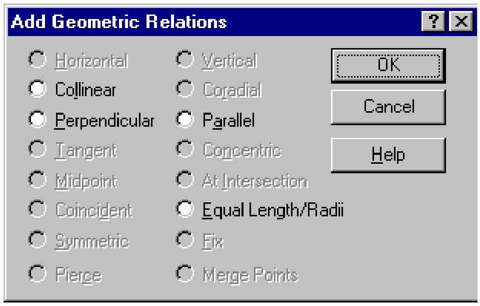
Add Relations  creates geometric relations (like tangent or perpendicular) between sketch elements, or between sketch elements and planes, axes, edges, or vertices.

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 2-21.

Geometric relations can be created two ways:

- ❑ Manually created by the user
- ❑ Automatically created by the SolidWorks software

Manually-Created Relations



To apply relations manually, first select the items (sketch elements, vertices, edges, and so forth) and then click , or click **Tools, Relations, Add**.

The **Add Geometric Relations** list box makes available only those relations that are appropriate for the items that you have selected; other relations are “grayed out.”

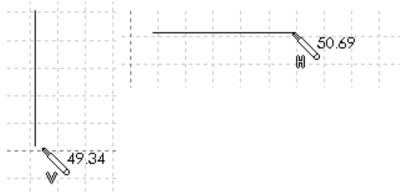
Note: For a complete discussion of all the sketch relations, refer to the **Add Relations Table** on page 2-24.

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.

You can turn **Automatic Relations** on or off by clicking **Tools, Sketch Tools, Automatic Relations**.

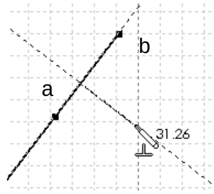
As you sketch, the cursor changes shape to indicate which relations can be created. Note that the cursor has a small **H** when the line is horizontal; the cursor has a **V** when the line is vertical. If the **Automatic Relations** option is on, those relations are created automatically.



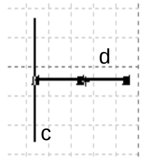
The following relations may also be created automatically: **At Intersection**, **Coincident**, **Midpoint**, **Tangent**, **Parallel**, and **Perpendicular**.

Examples of Automatic Relations:

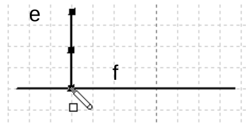
Automatic relations are formed between lines, and between points and lines.



- Line **a** has a *perpendicular* relation to line **b**, and a *coincident* relation to the endpoint of line **b**.
- Line **b** has a *perpendicular* relation to line **a**.



- Line **c** has a *vertical* relation, and a *midpoint* relation to the endpoint of line **d**.
- Line **d** has a *horizontal* relation.



- Line **e** has a *vertical* relation.
- Line **f** has a *horizontal* relation, and a *coincident* relation to the endpoint of line **e**.

Add Relations Table

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

To add this relation:	Select:	Result:
Horizontal or Vertical	One line or two points	The line becomes horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically.
Collinear	Two lines	The two items lie on the same infinite line.
Coradial	Two arcs	The two items share the same centerpoint and radius.
Perpendicular	Two lines	The two items are perpendicular to each other.
Parallel	Two lines	The two items remain parallel.
Tangent	An arc, spline, or ellipse and another arc, spline, ellipse, or line	The two items remain tangent.
Concentric	Two 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.
At 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 Length/ Radii	Two lines, or two 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	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.

*Definition 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.

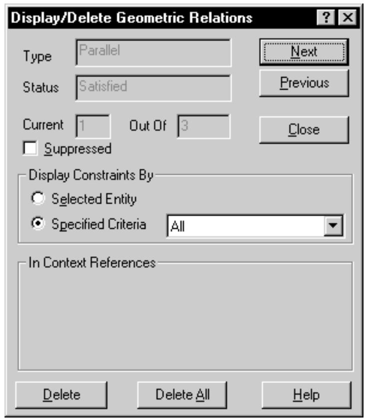
Point Can be a point in the sketch, or an external entity (origin, vertex, axis, or point in an external sketch) that projects as a point in the sketch.

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 “overdefined” or “not solved,” use the **Display/Delete** to examine and delete relations as necessary.

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 tells how many relations apply and allows you to view or delete the relations.

   The **Status** box informs you of any conflicting relations. (The status can be Dangling, Satisfied, Overdefined, or Not Solved.)

3 Click **Next** or **Previous** to view each of the relations in turn, if there is more than one.

4 You can click the **Suppress** check box to suppress or unsuppress the current relation.

5 Click **Delete** to delete the current relation; click **Delete All** to delete the relations to this entity.

You can specify search criteria for listing relations in a sketch by using the **Display Constraints By** box. For example, you can click **Specified Criteria**, and choose **Dangling** to list all the dangling relations in the sketch.

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.

Scan Equal

Click **Tools, Sketch Tools, Scan Equal** to scan a sketch for elements with equal lengths and/or radii. **Scan Equal** also provides a way to create equal length/radii relations between the sketch elements.

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 radii 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 Length/Radii** 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 2-26.
- *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 97Plus 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, you can set that option by clicking **Tools**, **Options**, and selecting 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 which 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, perhaps).

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

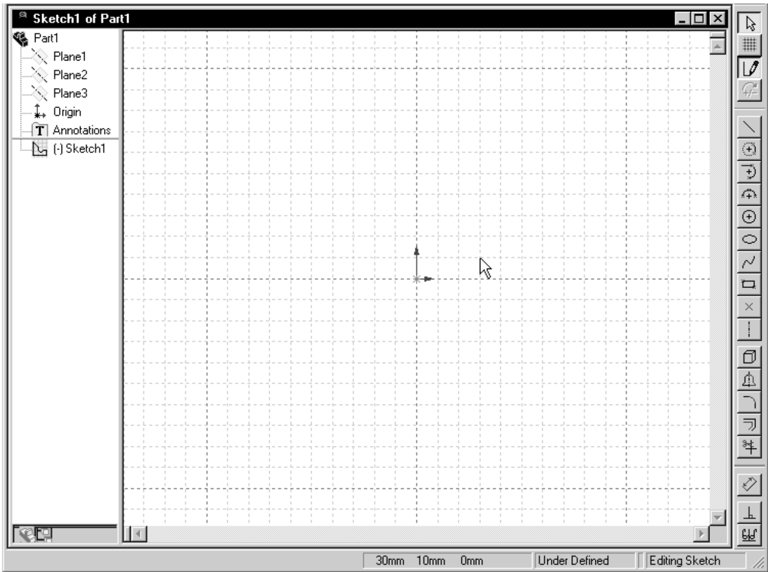
Over Defining – red. The dimension or relation is involved in over defining one or more entities.

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 2-28 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 2-37.

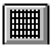
The Sketch Origin

The sketch origin is displayed in red in the active sketch. You can dimension to and add relations to 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 **Tools, Options.**
2. Select the **Grid/Units** tab.
3. 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:

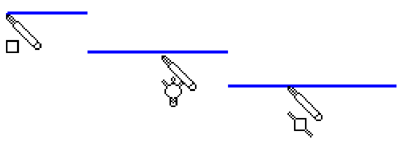
1. Click **Tools, Options**.
2. Click the **Grid/Units** tab.
3. Under the **Snap Behavior** heading, deselect **Snap to Points**.

For more information about additional ways to customize the sketch grid, such as changing the grid spacing, see **Grid/Units** on page A-13.

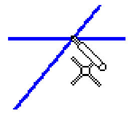
Inferencing Cursors and Lines

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.

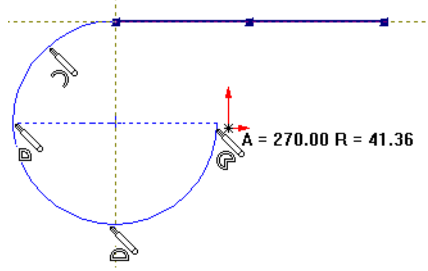


- ❑ When two lines intersect, the cursor changes to indicate the intersection.

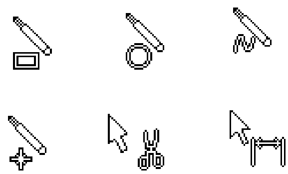


- ❑ When you sketch an arc, the cursor changes as it moves around the arc.

- ❑ The cursor provides information about dimensions as you sketch lines or arcs, such as the length, angle or radius of the sketch entity.

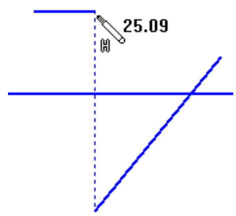


- ❑ When you select the various sketch or dimension tools, the cursor carries an appropriate symbol with it. Shown are the **Rectangle**, **Circle**, **Spline**, **Point**, **Trim/ Extend**, and **Dimension** cursors.

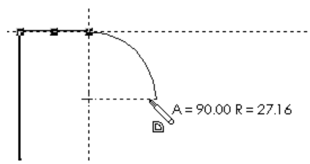


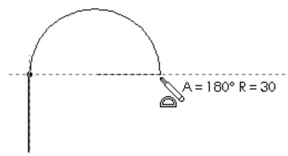
As you sketch, the SolidWorks software monitors your activity and provides *inferencing lines* to help you work more efficiently. While sketching, notice that dashed inferencing lines align your cursor with lines or points that you have already sketched and with existing model geometry.

- ❑ 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.



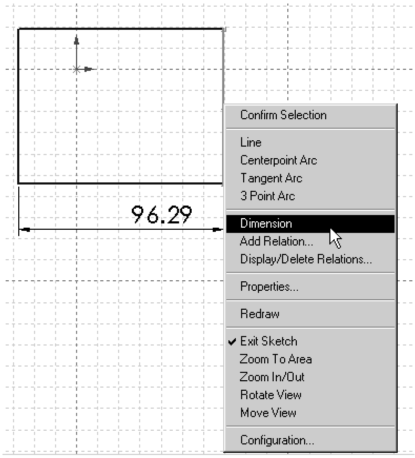
- ❑ When you sketch arcs, inferencing lines and dimensions guide you to create the size and shape arc that you want.





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 a line, press the right mouse button. The menu that comes up lets you select a different sketch tool or the **Dimension** tool without moving the cursor to the toolbar or main menu.

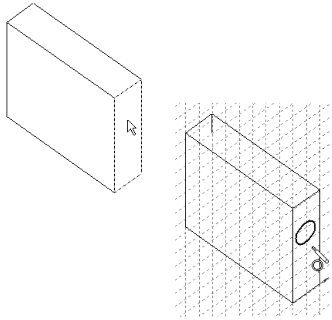
The right-mouse menu changes depending on the task that you are performing. Only those actions that are appropriate are listed as menu items.

Creating Features from Sketches

Sketches are the basis for many different types of features. See Chapter 3, *Creating Features*, for information about using sketches to create solid model geometry and reference geometry.

Creating a Sketch on the Face of a Part

To sketch on model faces:

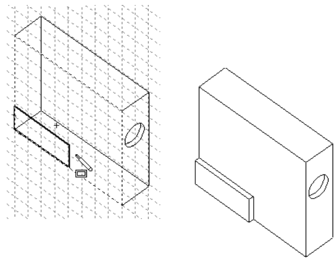


1 Click the model face on which you want to sketch.

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.

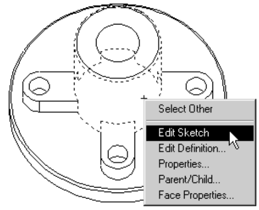


To sketch on a different face, you need to click the new face and open a new sketch. Note how the grid appears on the broad face of the base part in the this illustration.

Editing a Sketch

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

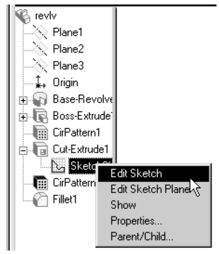
To open a sketch for editing using the right-mouse menu:



1 Right-mouse click a face on a part or right-mouse click the sketch in the FeatureManager design tree.

2 Click  in the Sketch toolbar or select **Edit Sketch** from the right-mouse menu.

The sketch used to create the selected feature opens for editing.



To edit a sketch:

1 Right-mouse click the sketch, or the feature created from the sketch, in the FeatureManager design tree and select **Edit Sketch**.

2 Edit the sketch in the graphics area.

3 Click  or right-mouse click and select **Exit Sketch**.

4 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 feature or a cut from your sketch.
- Click **Rebuild** when you are in a sketch.
- Click the **Sketch** tool  when in a sketch.
- Click **Insert, Sketch** from the main menu.
- Select **Exit Sketch** from the right-mouse menu.

Derived Sketch

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

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 on 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 sketch name in the FeatureManager design tree and select **Underive** from the menu. (After the link is broken, the derived sketch will no longer update 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.)

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 2-40. Additionally, you can insert a new plane and sketch on it.

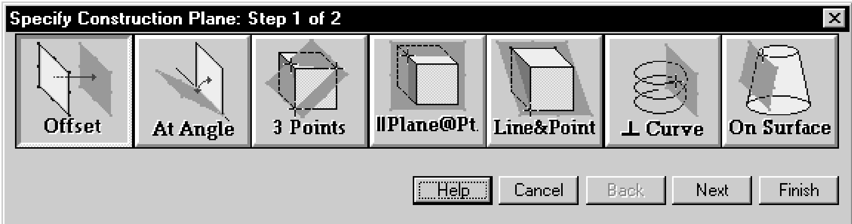
To add a plane:

1 Click **View, Planes**.

2 Click **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 or points.
2 Click **Finish**.

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

1. Select plane or planar face and a vertex or point.
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 or point.
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 will also need to select an edge on that surface.
4. Click **Other Solutions**, if necessary.
5. Click **Finish.**

Changing Construction Plane Names

You can create new names for Planes 1, 2, and 3 in the current document. Simply click two times 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 which 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 **Planes** tab.
3. Enter the new names in the boxes that correspond to the original plane names.
4. Click **OK**.

To return to the original names, click **Reset All**.

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 on the plane or on its name in the FeatureManager design tree.
2. Click **Hide** or **Show**.

Note: Individual planes always appear when you select them.

Edit Sketch Planes

You can change the model's sketch plane.

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 model face in the sketch.
3. Click **Rebuild** or click **Cancel** to exit without making a change.

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 (or construction axis) in a part.

You can hide or show all temporary axes by default.

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 on the axis or on its name in the FeatureManager design tree.

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

To create an axis in a part or assembly:

1 Click **Insert**, **Axis**.

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

- **One Axis.** Select **View, Temporary Axis**, and then select the axis.
- **Two Planes.** Select **View, Planes**, and then select two construction planes (while holding the **Ctrl** key).
- **Two Points.** Select two sketched points or two vertices.
- **One Line.** Select one line.
- **One Surface.** Select one cylindrical surface.

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.

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 an entity's property sheet:

1 Right-click on 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 select each sketched entity. In the **Common Properties** dialog, click **Properties** and click the **Construction Geometries** box.

In a drawing, you can convert sketch entities to construction geometry by selecting the entities and clicking **Tools, Sketch Tools, Construction Geometry**. For more information, see **2D Sketching** on page 6-45.

3

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:

- The Features Toolbar
- Base, Boss, and Cut
- Extrude, Revolve, Sweep, and Loft
- Fillet/Round, Chamfer, and Draft
- Hole - Simple and Hole Wizard
- Shell
- Rib
- Dome
- Pattern - Circular, Linear, and Mirror
- Curve
- Surface

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**. On the **Toolbars** tab, choose a category, and examine the available buttons. Drag the buttons you want into any toolbar, delete buttons you don’t 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

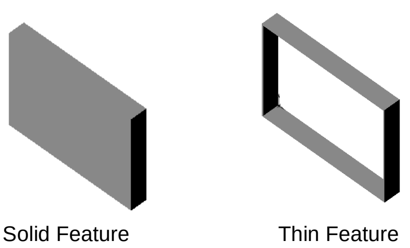
A *cut* is a feature which removes material from a part or an assembly.

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*.

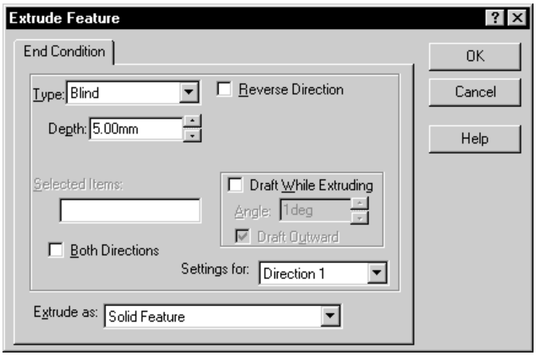
An extrude operation can either add material to a part (in a base or boss) or remove material from a part (in a cut or hole).



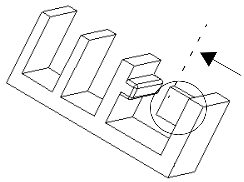
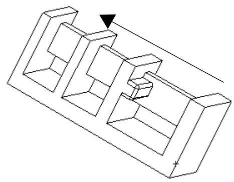
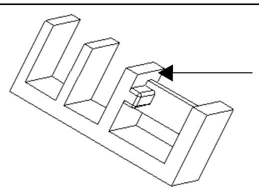
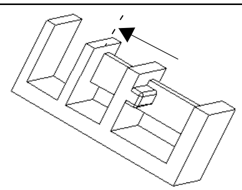
See **Surface** on page 3-43 for information about using **Extrude** to create a surface.

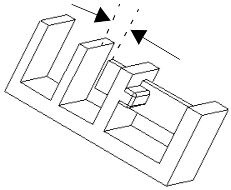
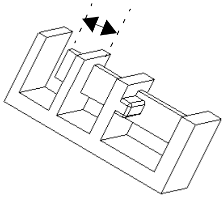
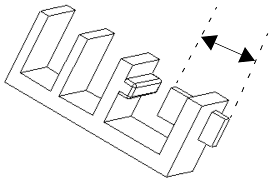
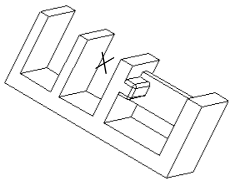
The **End Condition** tab of the **Extrude Feature** dialog lets you define the characteristics of extruded features created with these commands on the **Insert** menu:

- **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 surface, or set of surfaces, that intercepts the entire profile. The *next* surface must be on the same part.
Up to Surface		Extends the feature from the sketch plane to the selected surface.

Type	Example	Description
Offset from Surface		Extends the feature from the sketch plane to a specified distance from the selected surface.
Offset from Surface, Reversed		Extends the feature from the sketch plane to a specified distance *beyond* the selected 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

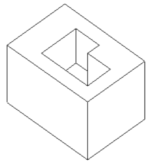
- **Type** determines how far the feature extends. See the preceding chart for examples.

  If you choose **Blind** or **Mid Plane**, you have to specify the **Depth**.

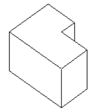
  If you choose **Offset from Surface**, you have to specify the **Offset**.

- **Depth** specifies the depth of the extrusion.
- **Reverse Direction** lets you extend the feature in the opposite direction from that shown in the preview.
- **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.

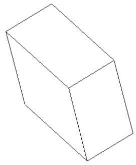


Default cut

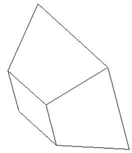


Flip side cut

- ❑ **Draft While Extruding** lets you add a 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.



No draft



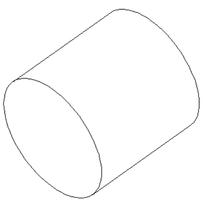
25^{o} draft angle – inward

- ❑ **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.

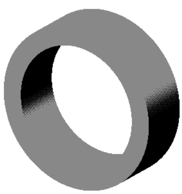
- ❑ **One Direction** or **Both Directions.**
  - **One Direction** extrudes the feature in one direction from the sketch plane. This is the default (**Both Directions** *unchecked*).
  - **Both Directions** extrudes the feature in both directions from the sketch plane. Specify all the settings for the 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.

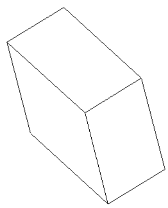
- ❑ **Extrude As:** a **Solid Feature** or a **Thin Feature**.
  - **Solid Feature** adds (or removes) solid volumes to the model.
  - **Thin Feature** adds (or removes) thin-walled volumes to the model. A **Thin Feature** base can also be used as a basis for a sheet metal part. See Chapter 8, *Sheet Metal*, for more information.



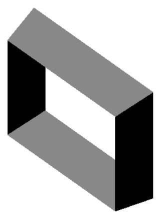
A circle extruded as a solid feature



A circle extruded as a thin feature



A rectangle extruded as a solid feature

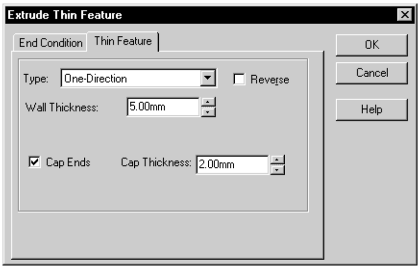


A rectangle extruded as a thin feature, with draft

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 Directions**.
  - **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 directions** 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.

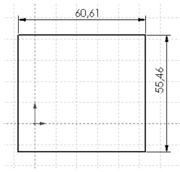
  This option covers (caps) the ends of the feature, creating a hollow part. If you check this option, you must also specify the **Cap Thickness**.
- ❑ If you create an *open profile* sketch, you can use the **Auto Fillet** option.

  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).

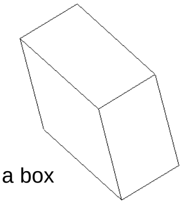
Extrude Base/Boss 

To extrude a solid or thin feature base (or boss):

1 With a sketch active, click one of the following:



A sketched rectangle...



...extruded to make a box

- **Insert**, **Base**, **Extrude**
- **Insert, Boss, Extrude**
- The **Extrude Base/Boss** button  on the Features toolbar

The **Extrude Feature** dialog box appears.

2 Select the **Type** of extrusion. (See the table on page 3-3 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 3-6.)

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 Round**, and specify the desired **Round Radius**.

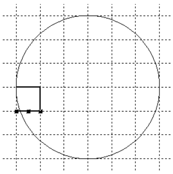
d) To cap the ends of a closed profile base feature, click **Cap Ends**, and specify the desired **Cap Thickness**.

10 When you are satisfied with the preview, click **OK**.

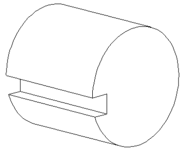
Extrude Cut 

To extrude a solid or thin feature cut:

This sketch for a cut on a cylinder face...



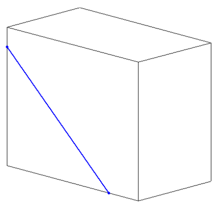
...results in this extruded cut with **Flip Side to Cut** selected.



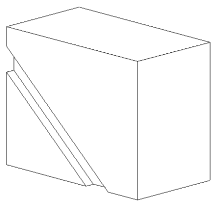
1. With a sketch active, click **Insert**, **Cut**, **Extrude** or click **Extrude Cut**  on the Features toolbar.

   The **Extrude Feature** dialog box appears.
2. Select the desired **Type**. (See the table on page 3-3 for more information.)
3. Specify the **Depth**.
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 3-6.)
    - **b)** Specify the **Wall Thickness**.
    - **c)** To extrude in the opposite direction, click **Reverse**.
11. When you are satisfied with the preview, click **OK**.



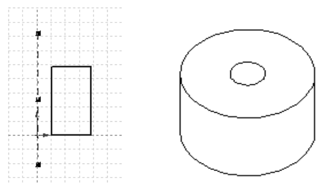
Thin Feature Cut from a sketched line



Revolve

Revolve creates a base, boss, or cut by revolving a sketch around a centerline. The default angle is 360^{o}. See **Surface** on page 3-43 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:

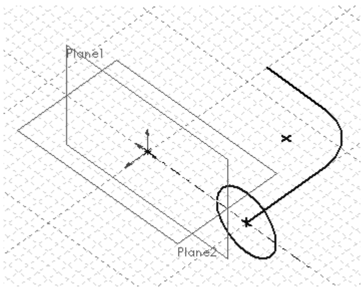
- The profile must be closed.
- The path may be open or closed.
- The path may be a set of sketched curves, a reference curve, or a set of model edges.
- The start point of the path must lie on the plane of the profile.
- Neither the section, the path, nor the resulting solid can be self-intersecting.

See **Surface** on page 3-43 for information about using **Sweep** to create a surface.

Simple Sweep

To create a simple sweep:

Sketch on multiple planes, following these general steps:

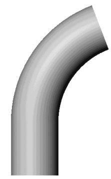


1. Sketch a *closed, non-intersecting* profile on one plane.
2. Create the path that the profile will follow. Use a sketch, existing model edge(s) or reference curve(s).
3. Click one of the following:
   - **Insert, Base, Sweep**
   - **Insert, Boss, Sweep**
   - **Insert, Cut, Sweep**

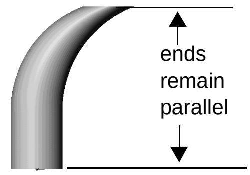
   The **Sweep** dialog box appears. For a simple sweep, you only need to use the **Sweep** tab in the dialog box.
4. Click the **Sweep Section** box, then select the section profile either in the model or in the FeatureManager design tree.
5. Click the **Sweep Path** box, then select the sketch, edge, 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.



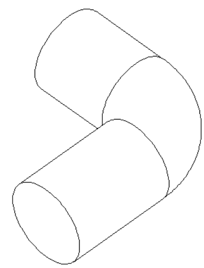
Follow Path



Keep Normal Constant

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**.

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 97Plus*, Chapters 4 and 12.

Sweep with Guide Curves

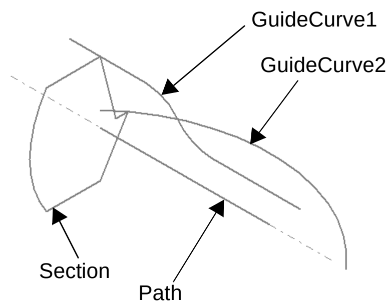
You can use guide curves to control the intermediate profiles as the sketch is swept along the path.

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).



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 profile *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. Therefore the profile must come *after* the path and guide curves in the FeatureManager design tree.

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 2-36 for more information.

As you sketch the section, pay special attention to the use of **Horizontal** or **Vertical** relations. Because of the way the section changes as it sweeps along the path, you may need to adjust the **Orientation/Twist Control** (see Step 8) to achieve the desired effect.

4 Create relations in the profile sketch to the guide curve(s). Use **Pierce** relations where the guide curve intersects the profile at a vertex or a user-defined sketch point. It is important that the section be constrained to the curves, and not vice-versa.

5 Click **Insert, Base/Boss/Cut, Sweep**.

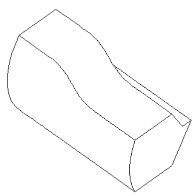
The **Sweep** dialog box appears. For a sweep with guide curves, you need to use both the **Sweep** tab and the **Advanced** tab.

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.

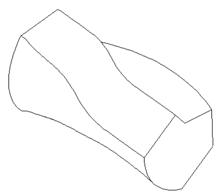
7 Click the **Sweep Path** box, then select the path sketch, curve, or model edge.

8 Under **Orientation/Twist Control**, there are two additional choices besides **Follow Path** and **Keep Normal Constant** (see page 3-11):

- **Follow Path and 1st Guide Curve -** The angle between the section and the path remains the same along the length of the path, and the twist is based on a vector between the path and the *first* guide curve.
- **Follow 1st and 2nd Guide Curve -** The angle between the section and the path remains the same along the length of the path, and the twist is based on a vector between the *first* and *second* guide curves.



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:

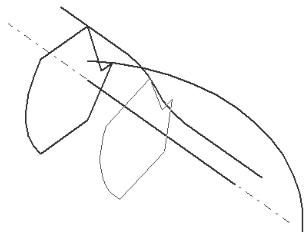
- **Maintain Tangency -** if the sweep section has tangent segments, checking this option causes the corresponding surfaces in the resulting sweep to be tangent. Faces that can be represented as a plane, cylinder, or cone are maintained. Other adjacent faces are merged, and the sections are approximated. 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 Click **OK.**

To view the modifications in the profile as it moves along the path:

You can examine the intermediate profiles created along the sweep path, even in the case of a sweep that is not solved successfully. 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.
  - If the path and the guide curves are offset from one another, the sweep is as long as the shortest overlapping distance between them all.
- Guide curves may meet at a common point, which is the apex of the swept surface.
- If a profile is symmetrical, symmetry is maintained even as the section varies due to the guide curve.
- If a guide curve has no **Pierce** relation with a sketch point, no edge on the sweep will follow the guide curve.
- 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 3-37 for information about creating reference curves.

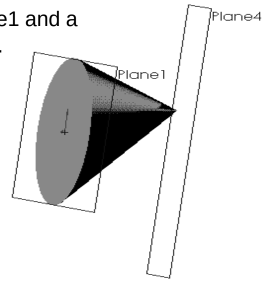
Loft

Loft creates a feature by making transitions between cross-sections. A loft can be a base, boss, or cut.

You may sketch two or more profiles, or at least one profile and one point. Only the first and/or last profile can be a point.

See **Surface** on page 3-43 for information about using **Loft** to create a surface.

A circle on Plane1 and a point on Plane4.



Simple Loft

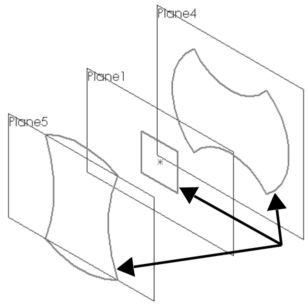
To create a simple loft:

1 Set up the planes needed for the profiles. Use existing faces and construction planes, or create new construction planes. Planes do not have to be parallel.

2 Sketch the profiles, or profile and point.

3 Click one of the following:

- **Insert, Base, Loft**
- **Insert, Boss, Loft**
- **Insert, Cut, Loft**



Click corresponding points

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.

5 If the preview curve looks wrong:

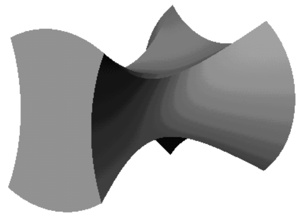
- Use the **Up** or **Down** buttons to rearrange the sketches if you have selected them in the wrong order.
- If the preview curve indicates that the wrong vertices will be connected, click the profile once to deselect it, then click again to select a different point on the profile.
- To clear all selections 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 **OK**.

For examples of creating simple loft features, refer to the tutorial, *Learning to Use SolidWorks 97Plus*, Chapter 7.



Loft with Guide Curves

You can create a *guide curve loft* by sketching two or more profiles and one or more guide curves to connect the profiles.

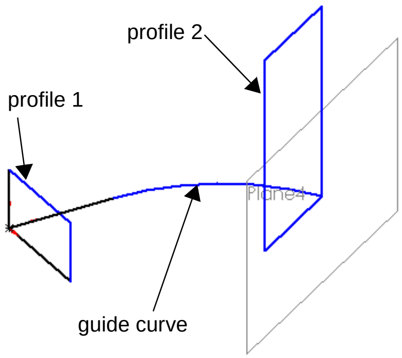
To make a loft with guide curves:

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.



It is recommended that these relations be added to the profile sketches.

4 Click **Insert, Base, Loft.**

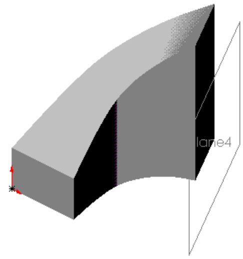
5 Click the **Profiles** box, then select the profile sketches in order in the FeatureManager design tree or the graphics window.

6 Click the **Guide Curve(s)** box, then select the guide curve sketch(es).

7 Examine the path preview, and use the **Up** or **Down** buttons if necessary to adjust the order.

8 Click **Close Along Loft Direction**, **Advanced Smoothing**, and **Maintain Tangency** if needed. See **Simple Loft** on page 3-15 for more information about these options.

9 Click **OK.**



Notes on Creating Lofts with Guide Curves:

- ❑ Guide curves must pass through the sketched profiles, either at the vertices or at user-defined sketch points on the profile.
- ❑ There is no limit on the number of guide curves you may use.
- ❑ Guide curves can intersect at a sketch point or the apex of the lofted surface.
- ❑ 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 3-37 for information about creating reference curves.
- ❑ If a simple loft fails or twists, it is recommended that you 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 3-41.
- ❑ Guide curves can be longer than the resulting loft. Only that portion of the curve that is needed will be used.
- ❑ You can 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.

Fillet/Round 

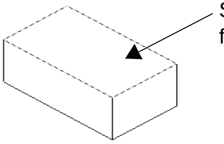
Fillet/Round creates a rounded internal or external face on the part. You can fillet all edges of a face, selected sets of faces, selected edges, or edge loops.

In general, it is best to follow these rules when making fillets:

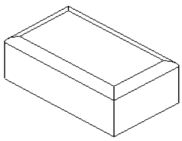
- ❑ Add larger fillets before smaller ones. When several fillets converge at a vertex, create the larger fillets first.
- ❑ Add drafts before fillets. If you are creating a molded or cast part with many filleted edges and drafted surfaces, in most cases you should add the draft features before the fillets.
- ❑ 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:



Before and after creating a fillet feature.



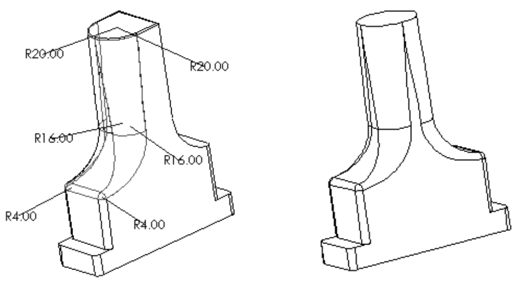
1. Click **Fillet**  on the Features toolbar, or **Insert**, **Features, Fillet/Round**.
2. Select the faces and edges that you want to fillet or round.
3. Specify the fillet **Radius**.
4. Select **Fillet Type** of **Constant Radius**.
5. Verify that the correct number of edges and faces are shown in the **Items to Fillet, Edge Fillet Items** list.
6. Select **Propagate Along Tangent Edges** if necessary. This extends the fillet to all edges that are tangent to the selected edges.
7. Select the **Overflow Type**. See **Overflow Type** on page 3-20 for more information.
8. Click **OK**.

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. Select **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**.



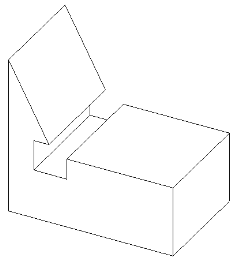
To change the radii of the fillets after they are inserted, double-click the fillet feature, then double-click and modify the dimension displayed on each vertex.

For an example of creating variable radius fillets, refer to the tutorial, *Learning to Use SolidWorks 97Plus*, Chapter 13.

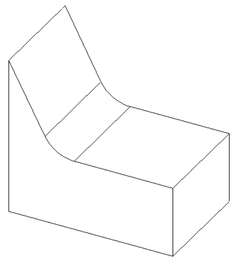
Face Blend

You can blend non-adjacent faces with a **Face Blend** fillet.

To insert a face blend fillet:



1. Select **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**.
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 which 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**.

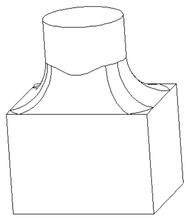


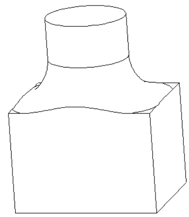
Overflow Type

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

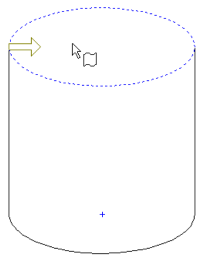

Keep Surface

Chamfer

Chamfer creates a beveled edge on the selected edges and/or faces.

1 **To create a chamfer:**

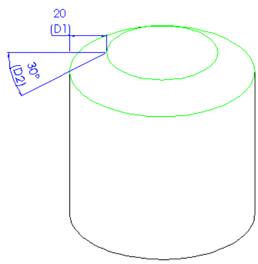
2 Click **Chamfer** on the **Features** toolbar, or **Insert, Features, Chamfer.**



3 Select the faces and/or edges to chamfer. Specify the **Distance** and the **Angle** of the chamfer. The distance is measured in the direction the arrow points.

4 Verify that the list in **Items to Chamfer** is correct.

An arrow appears on the part to indicate the direction of the chamfer.



5 Verify that the arrow is pointing in the desired direction. If necessary, click **Flip Direction.**

6 Click **OK.**

Draft 

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 **Insert, Features, Draft**.

2 In the **Type of Draft** box, select **Neutral Plane**.

3 Set the **Draft Angle.**

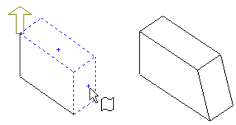
4 Click in the **Neutral Plane** box, and select the neutral plane in the graphics area.

5 If you want the draft to slant in the opposite direction, click **Reverse Direction**.

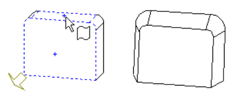
6 Click in the **Faces to Draft** box, and select the faces to draft in the graphics area.

7 Choose the **Face Propagation** type that describes how you want the draft to propagate across additional faces.

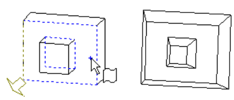
- **None**. Only the selected face is drafted.



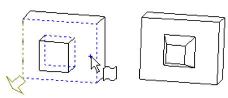
- **Along Tangent.** Extend the draft to all faces that are tangent to the selected face. (The faces meet with filleted corners.)



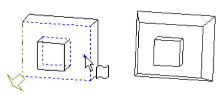
- **All Faces.** Draft all faces next to the neutral plane and extruded from the neutral plane.



- **Inner Faces.** Draft all faces extruded from the neutral plane.



- **Outer Faces.** Draft all faces next to the neutral plane.



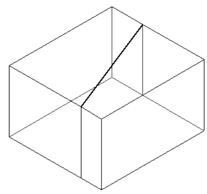
8 When you are satisfied with the results, click **OK**.

Parting Line

The parting line option lets you draft surfaces around a parting line. The parting line can be non-planar.

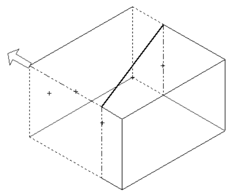
To draft on a parting line, you may first divide the faces to be drafted by inserting a **Split Line** (see page 3-39), or you may use an existing model edge. Then you specify the *direction of pull*, that is, which side of the parting line material is removed from.

To insert a draft angle in an existing part using parting line:

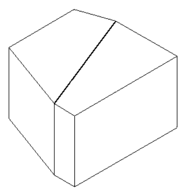


1. Sketch the part to be drafted, and desired parting line(s).
2. Insert a split line curve as described in **Split Line** on page 3-39.
3. Click **Insert, Features, Draft.**
4. In the **Type of Draft** box, select **Parting Line**.
5. Specify the **Draft Angle.**
6. Click the **Direction of Pull** box, 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 necessary.



7. Click the **Parting Lines** box, and select the parting lines in the graphics area.
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 (faces that meet with fillets or rounds.)



9. 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.

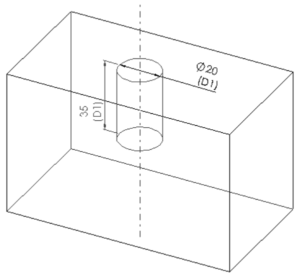
- **Simple -** Places a circular hole of the depth you specify.
- **Wizard -** Creates holes with complex profiles, such as **Counterbore** or **Countersunk**.

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.

To insert a Simple hole:

1. Select a planar face on which to create the hole.
2. Click **Insert, Features, Hole**, **Simple**.
3. Select the **Type**, and specify the **Depth** or **Offset**, if necessary.

   Note: For information about selecting **Type**, see **Specifying End Conditions** on page 3-4.



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. To create a hole in both directions, select the **Both Directions** checkbox. Indicate settings for **Direction 1** and **Direction 2**.
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 the necessary dimensions to position the hole.
   - **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.

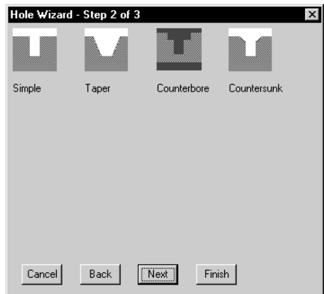
2 Click **Insert, Features, Hole, Wizard**.

3 Select the **Type** and specify the **Depth** if necessary.

4 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.

5 Click **Next.**

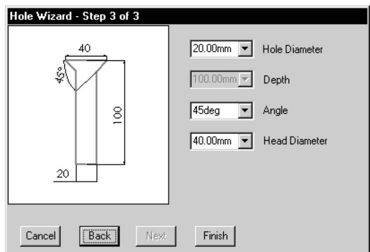
The **Step 2 of 3** dialog box appears.



6 Select the hole type by clicking its diagram.

7 Click **Next.**

The **Step 3 of 3** dialog appears, where you can change dimensions, if needed. The appearance of this dialog box depends on the style of hole that you choose. (The dialog box shown here is for a **Countersunk** hole.)



8 Modify the dimensions as necessary.

9 Click **Finish.**

10 To locate the hole:

a) Expand the hole feature in the FeatureManager design tree by double-clicking its name, or by clicking the plus sign beside its name.

b) Right-click the first sketch under the hole name in the FeatureManager design tree and select **Edit Sketch**. (The sketch consists of a single point at the center of the hole.)

c) Add the necessary dimensions to position the point in the sketch.

d) Exit the sketch or click **Rebuild**.

Shell 

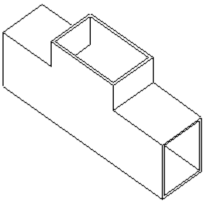
Shell hollows out the part, leaving open the faces you select, and thin walls on the remaining faces.

To insert a shell:

1 Click **Insert**, **Features, Shell** or click **Shell**  on the Features toolbar.

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.

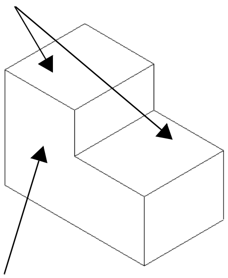


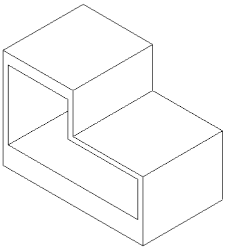
3 Specify the wall **Thickness**.

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 **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.

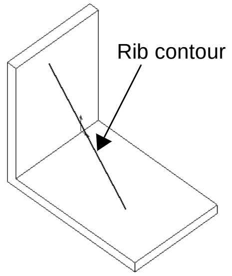
6 Choose whether to offset the shell inward or outward.

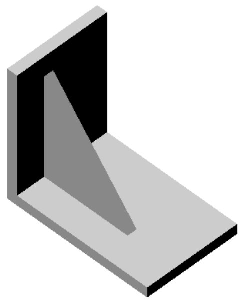
7 Click **OK**.

Rib 

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**  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.

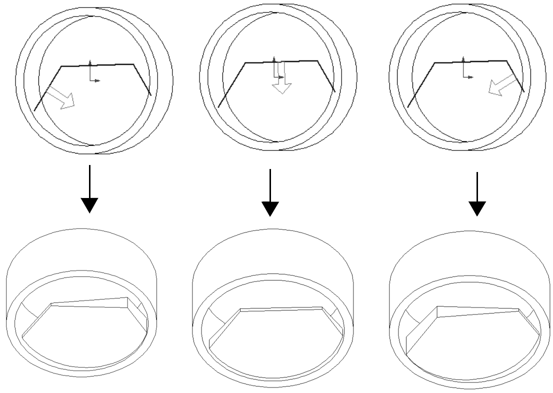
   If you chose **Single Side**, examine the preview and select **Reverse** if necessary.
4. Enter the **Thickness** of the rib and click **Next.**
5. Note the direction of the arrow in the preview. If necessary select **Flip Side of material** to reverse the direction in which material is added.
6. To add draft, select **Enable Draft** and enter the **Angle**. Select **Draft Outward** if necessary.
7. Click **Finish**.

To insert a rib with multiple draft angles:

1. Sketch the part and contour lines of the rib.
2. Click **Rib**  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 model face that has a circular, elliptical, or four-sided boundary. A four-sided boundary need not be rectangular, and each side may consist of a single segment or a set of tangent segments.

To create a dome on a planar face:

1 Click **Insert, Features, Dome**.

2 Select a planar **Dome Face** in the graphics area.

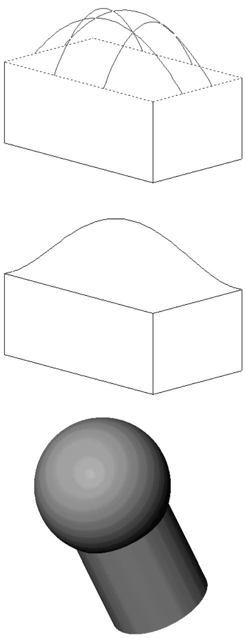
3 Specify the **Height**, and observe the preview. The height is measured from the centroid of the selected face.

4 Click **Reverse Direction** to create a concave dome (default is convex).

5 If a circular or elliptical face is selected, you can click **Do Elliptical Surface.** This creates a dome whose shape is a half ellipsoid, with a height equal to one of the ellipsoid radii.

6 Click **OK**.

Note: A dome on a circular face may be larger in diameter than the selected face.



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.

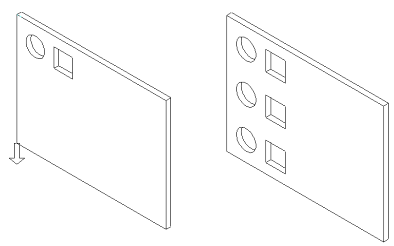
For information about using a pattern of components in an assembly, see **Adding a Component Pattern** on page 5-9.

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. In the FeatureManager design tree, select the feature(s) to repeat.
3. Select **Insert**, **Pattern/Mirror**, **Linear 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.

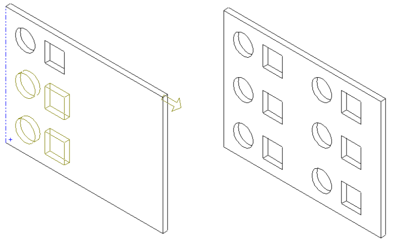
Spacing is the distance from a point on one feature to the corresponding point on the adjacent copy (*not* edge-to-edge spacing).

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 3-33 for more information about using **Vary Sketch**.

8 Click **OK** to create the pattern in one direction, or continue with Step 9 to create the pattern in two directions.



9 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**.

10 Set the values for **Spacing** and **Total Instances** of the pattern in the **Second Direction** and examine the preview.

11 Click **OK**.

Circular Pattern

You use a circular pattern to create multiple copies of a feature or features in rotation about an axis.

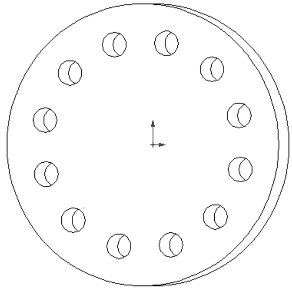
To create a circular pattern:

1 Create 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).

2 Select the axis or edge, hold the **Ctrl** key and select the feature(s) to pattern.

3 Click **Insert, Pattern/Mirror, Circular Pattern**.



4 Select the **Reverse Direction** checkbox if you want to create the pattern in a counter-clockwise direction. Otherwise, the pattern goes in a clockwise direction.

5 Specify the **Spacing** in degrees and the **Total Instances** of the feature.

As you modify the values, a preview of the resulting pattern is displayed.

6 Click **Vary Sketch** if you want the pattern to change as it is repeated. (Refer to **Controlling and Modifying Patterns** on page 3-33 for more information on using **Vary Sketch**.)

7 Click **OK**.

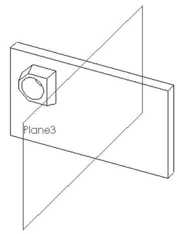
Mirror Feature

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):

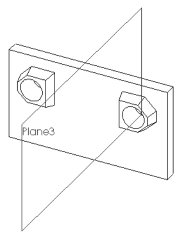
1 Select the feature(s) to mirror in the model or in the FeatureManager design tree.

2 Hold the **Ctrl** key and select the plane about which to mirror the feature(s).



3 Click **Insert, Pattern/Mirror, Mirror Feature**.

A preview of the mirrored feature(s) is displayed.



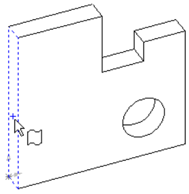
4 Click **OK**.

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 All

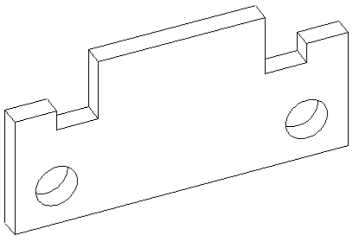
To create a part that is symmetrical about a planar face, you can use **Mirror All.** You build one half of the part, then mirror the entire model all at once. Any changes you make to the original half are reflected in the other half.

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.



Mirror Pattern

To create a mirrored copy of a pattern:

1. Select a 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.
2. Hold down the **Ctrl** key and select a plane about which to mirror the pattern.
3. Click **Insert, Pattern/Mirror, Mirror Feature.**

   A preview of the mirrored pattern is displayed.
4. Click **OK**.

Pattern of Patterns

To create a pattern of patterns:

1. 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.
2. Select **Insert**, **Pattern/Mirror**, then either **Linear** or **Circular** pattern.
3. Proceed as described in **Linear Pattern** on page 3-29 or **Circular Pattern** on page 3-30.

Controlling and Modifying Patterns

You can control and modify feature patterns in the following ways:

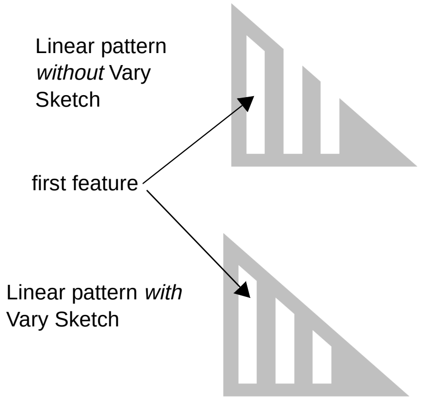
- ❑ Use the **Vary Sketch** option to adjust the profile of the patterned feature based on its dimensions and relations to other features.
- ❑ Use **Delete Instance** to remove individual instances of the patterned features.
- ❑ Use a mathematical **Equation** to calculate values that define the pattern.

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 hole 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.



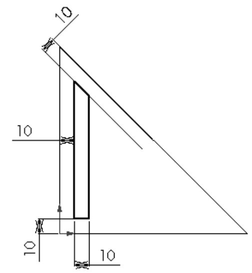
To create a variable pattern:

1 Create a sketch for the first feature in the pattern, observing the following recommendations:

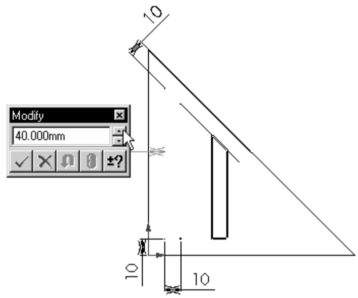
- The feature sketch must be constrained to the boundary that defines the variation of the pattern instances. For example, in the illustrated pattern, the angled top edge of the first feature is parallel and dimensioned to the angled edge on the base part.
- The feature sketch should be *fully defined*.

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 **Reverse Direction** if the arrow points in the wrong direction.

8 Click the **Vary Sketch** checkbox.

9 Specify the values for the **Spacing** and **Total Instances** of the pattern.

10 Click **OK** to create the pattern.

Delete Instance

To delete an instance of a feature from a pattern, or an entire pattern:

1 Select a face on the instance of the feature you want to delete.

2 Press the **Delete** key.

3 Indicate whether to **Delete Pattern Instances** (selected instances of the pattern) or **Delete Pattern Feature** (the entire pattern).

Equations

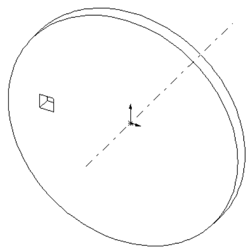
You can use equations to define mathematical relations between parameters or dimensions. For example, you can define a ratio between dimensions that causes a boss to be half as high as the base of a part.

In the example that follows, you use an equation to calculate the spacing angle used in a circular pattern. The equation divides 360 degrees by the number of total instances of the feature in the pattern. This way, the features are uniformly spaced around the circle, regardless of the number of instances.

To use an equation to control a circular pattern:

1 Create a base feature and a feature to pattern.

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

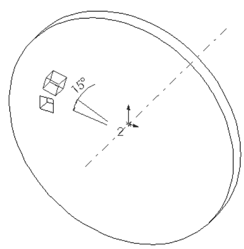


3 Select the temporary axis that passes through the center of the part.

4 Hold the **Ctrl** key and select the feature to pattern in the FeatureManager design tree or in the model.

5 Click **Insert**, **Pattern/Mirror**, **Circular Pattern**.

6 Leave the default values in the **Spacing** and **Total Instances** boxes. It is not important what numbers you use at this time.



7 Click **OK**.

8 In the FeatureManager design tree, double-click the circular pattern feature.

Two values appear in the model: **Total Instances** and **Spacing** angle.

9 Click **Tools, Equations**, then click **Add** in the **Equations** dialog box.

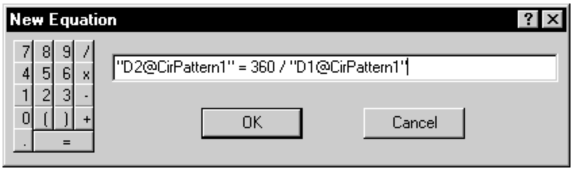
10 Click the **Spacing** angle dimension in the model.

Its full name (from the **Dimension Properties** sheet) is entered in the text field of the **New Equation** dialog box.

11 Using the calculator keypad in the dialog box, click the equals sign (**=**) and click **360 /**. (Or you can type “=360/” on the keyboard.)

12 Click the **Total Instances** value in the model.

Its full name is added to the end of the equation.



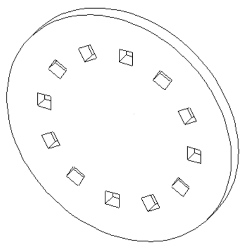
13 Click **OK** to complete the equation, and click **OK** again to close the **Equations** dialog box.

14 Change the number of features in the pattern:

a) Double-click the **Total Instances** value in the model.

b) Set the value in the spin box to the number of features that you want.

15 Click **Rebuild**  in the **Modify** dialog, or close the dialog and click **Edit, Rebuild**.



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 to a model face, or from sketched lines on intersecting planes
- **Helix -** by specifying pitch, revolutions and height
- **Split Line -** for 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

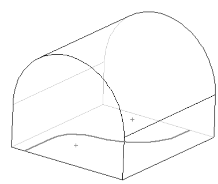
You can then use the curves to create solid model features. For example, you can use a curve as the path or guide curve for a sweep feature, as guide curve for a loft feature, as a parting line for a draft feature, etc.

Projected Curve

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

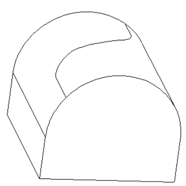
To project a curve onto a face:

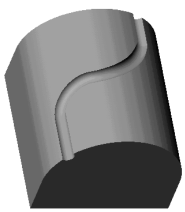
1. Create a sketch containing a single open or closed curve (made up of lines, arcs, or splines) on a plane or model face.



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.



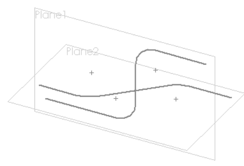


Projected curve used as sweep path

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

To create a projected curve using sketches on intersecting planes:

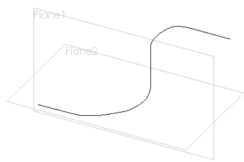
1 Create a sketch on each of two intersecting planes, closing each sketch when you are done.



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.

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 a definition from the **Defined by** scroll box. To define a helix, you specify two values, and the third is calculated automatically.

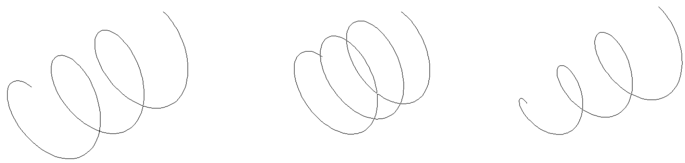
- **Pitch and Revolutions**
- **Height and Revolutions**
- **Height and Pitch**

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 necessary, click **Taper Helix**, specify a taper **Angle,** and check **Taper Outward** if desired.

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**).



Smaller pitch Tapered

9 Click **OK**.

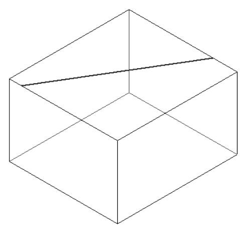
Split Line

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 and modified individually. **Split Line** is used with the **Parting Line** option in the **Draft** feature (see page 3-22).

To project a split line onto a planar face:

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

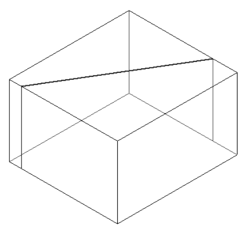
2 Exit the sketch.



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.

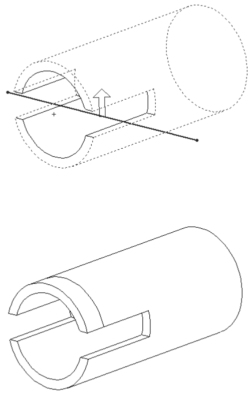
7 Click **Finish.**

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 sketch plane of the **Sketch to Project** lies in the area of a cut.

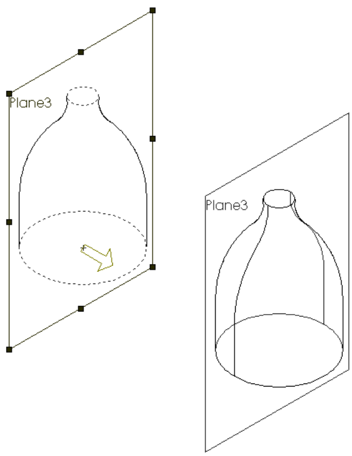
- Click **Single Direction** to project the curve in one direction only.
- Click **Reverse Direction** if the preview indicates that the curve projects the wrong way.

If you do not click **Single Direction** here, 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, Curve, 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 **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.

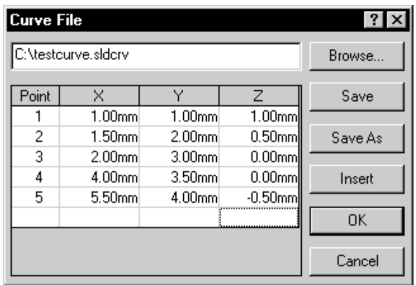
4. Click **OK**.

Curve Through Free Points

You can create a 3D spline from a set of points. There are two ways to generate a point list for this purpose:

- You can create a curve file 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 one point at a time:



1. Click **Insert, Reference Geometry, Curve Through Free Points.**

2. In the **Curve File** dialog box, double-click any cell in the first row to activate the cell.

3. Enter the coordinates for **X**, **Y**, and **Z** (double-click to activate each cell). Notice that the value in the **Point** column automatically is set to 1. The numbers in the **Point** column define the order in which the points will be connected.

4. Double-click in the next row, and enter the coordinates. Notice the preview of the curve in the graphics area.

5 Repeat Step 4 to continue adding as many points as needed. The larger the number of points, the greater your control of the resulting curve.

6 If necessary, you can insert a new point between existing points. Either click the point number or drag the cursor across the cells to select the row where you want the new point to occur, then click **Insert**. A new row is added, and all the points below the selected row move down.

7 If desired, 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**.

To create a curve from a file:

1 Create a point file using a text editor, spreadsheet, or other utility. The format of the file is 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:

- Extrude, revolve, sweep, or loft from sketches
- Offset from existing geometry
- Import from IGES (see Chapter 11, *Importing and Exporting Files*, for information about importing and using surfaces)

You can then use the surfaces to create features. For example, you can:

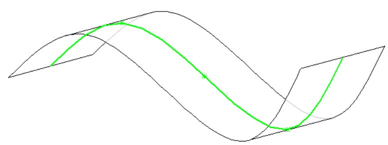
- Create a solid or cut feature by thickening a surface.
- Extrude a solid or cut feature with the end condition **Up to Surface** or **Offset from Surface**.
- Cut the model with a reference surface or by thickening a surface.

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, or offset from existing model geometry. A surface may have either an open or closed profile.

To extrude a surface:

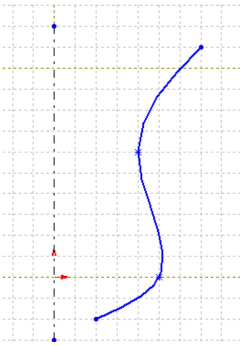
1. Sketch the profile of the surface.
2. Click **Insert, Reference Geometry, Extruded Surface**.
3. In the **Extrude Feature** dialog box, on the **End Condition** tab, choose the **Type**, and specify the **Depth**.
4. Examine the preview. If the offset is in the wrong direction, click **Reverse Direction**.
5. Click **OK**.

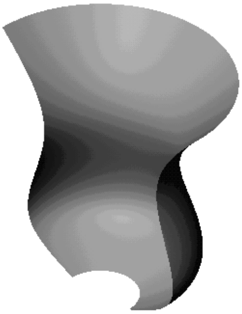


Extruded surface (MidPlane type) from a sketched spline

See **Extrude** on page 3-2 for more information about extruded features.

To revolve a surface:





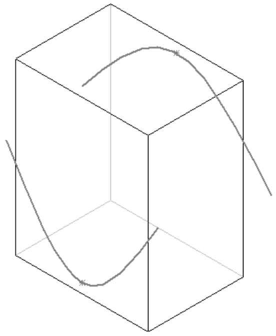
1. Sketch a profile and a centerline around which to revolve the profile.
2. Click **Insert, Reference Geometry, Revolved Surface**.
3. If the profile is open, you will be asked if you would you like the sketch to be automatically closed.

   Click **Yes** to create a closed profile, or **No** to leave the profile open.
4. Enter the value for **Degrees**, and click **Reverse Direction** if necessary.
5. Click **OK**.

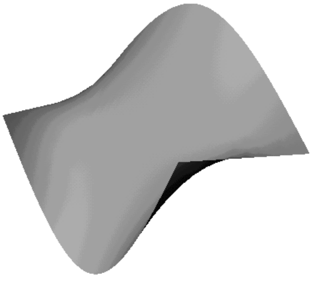
See **Revolve** on page 3-9 for more information about revolved features.

To loft a surface:

1. Sketch the guide curves (if any are needed) and the profiles for the lofted surface.
2. Click **Insert, Reference Geometry, Lofted Surface**.
3. 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.
4. If you are using guide curves, click the **Guide Curves** box in the **Loft Surface** dialog, then select the guide curves.
5. Choose from the options **Maintain Tangency**, **Close along Loft Direction**, and **Advanced Smoothing** as desired.
6. Click **OK**.



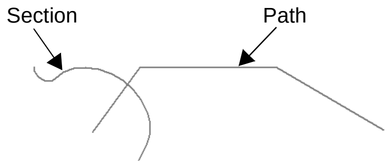
Profiles sketched on model faces



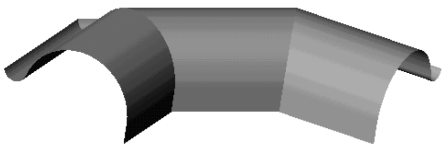
See **Loft** on page 3-15 for more information about lofted features.

To sweep a surface:

1. Sketch the guide curves (if any are needed) and path for the swept surface on model faces, or create planes as needed for sketching.



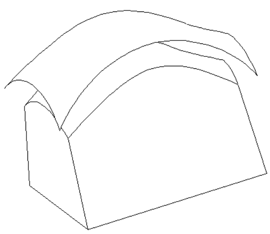
2. Sketch the section of the surface, and create coincident/pierce relations between the guide curve(s) and the section. If the path is made up of more than one segment, the section must lie on a plane which is normal to the path.
3. Click **Insert, Reference Geometry, Swept Surface**.
4. Click the **Sweep Section** box, then click the profile sketch.
5. Click the **Sweep Path** box, then click the path sketch, curve, or model edge for the path.
6. If you are using guide curves, click the **Advanced** tab. Click the **Guide Curves** box, then click the sketch, curve, or model edge for the guide curve(s).
7. Select options as desired under **Orientation/Twist Control**, **Advanced Smoothing**, and **Maintain Tangency.**
8. Click **OK**.



See **Sweep** on page 3-10 for more information about 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**.



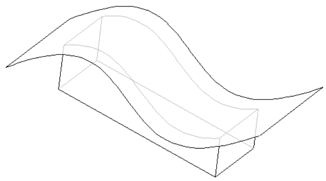
Offset Surface from a lofted model surface

Using Surfaces to Create Features

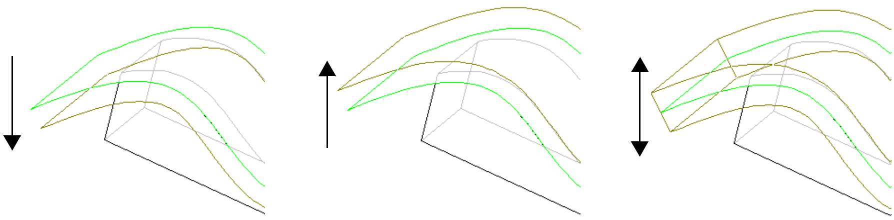
A reference surface has no thickness. You can thicken a surface to create model geometry, or to cut into existing geometry. You can also use a zero-thickness surface to cut through a model, similar to cutting with a parting line.

To create a solid feature by thickening a surface:

1. Create a surface as described in the previous section, or import an IGES surface.



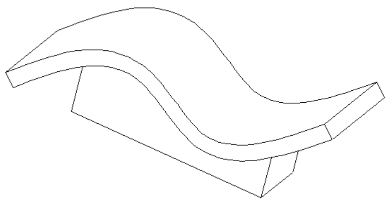
2. Click **Insert**, **Base**, **Thicken** (if there are no features in the part) or **Insert, Boss, Thicken**.
3. Select the surface.
4. In the **Thicken Feature** dialog, specify the desired **Thickness** for the feature.
5. Specify the side(s) you want to thicken. Examine the preview to see the effect.



Thicken Side 1 Thicken Side 2 Thicken Both Sides

If you are creating a boss, the thickened surface *must* intersect the existing part.

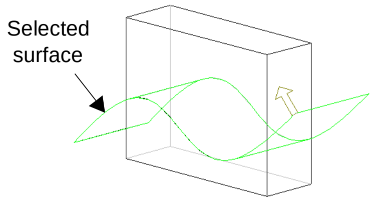
6. Click **OK**.



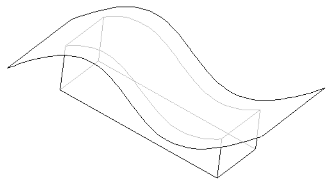
To cut a part with a surface:

1. Create a reference surface in the part model, or import one.
2. Select the surface.
3. Click **Insert, Cut, With Surface**.

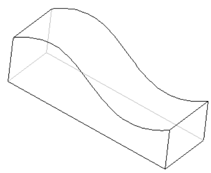
   Note the preview arrow, indicating which side of the part will be cut away.



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**.



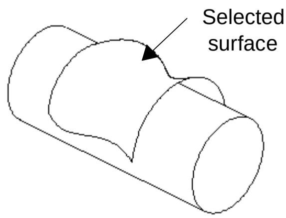
6. To hide the surface if desired, right-click the **RefSurface** feature in the FeatureManager design tree and select **Hide**.

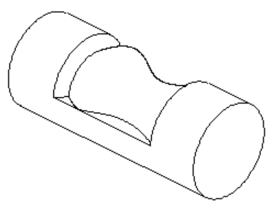


Note: You can also cut a part with a surface by using the **Up To Surface** option when extruding a cut.

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**.





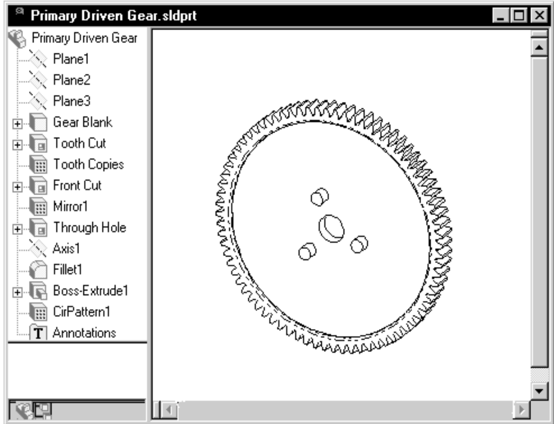
4

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:

- Using the FeatureManager design tree
- Understanding Parent/Child relationships
- Suppressing and unsuppressing part features
- Working with part configurations
- Using a design table
- Annotating parts
- Applying lighting direction and intensity to the model

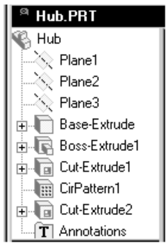
The FeatureManager design tree on the left side of part windows provides an outline view of the part. The design tree makes it easy to see how the part was constructed.

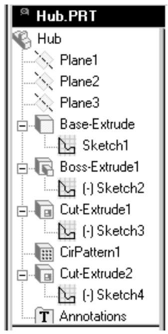


In a part, features are listed in the order in which they are regenerated. You can change that order by dragging a feature and dropping it earlier or later on the list.

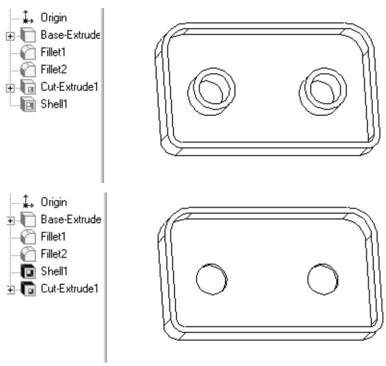
In the FeatureManager design tree of an active part, you can:

- ❑ Click on the small ⊞ sign to the left of an item's icon to expand the item and display its contents.
- ❑ Right-click a feature so that you can edit its definition, sketch, or properties.
- ❑ Double-click to expose a feature's dimensions.
- ❑ Click twice slowly (click, *pause*, click), and type in the new text to change the name of an item in the FeatureManager design tree.





- ❑ Select a feature to revert to when using **Rollback, Suppress, Unsuppress**, or **Unsuppress with Dependents**.
- ❑ View parent/child relationships by right-clicking a feature in the list, then clicking **Parent/Child**.
- ❑ Drag and drop items in the FeatureManager design tree list to reorder them. This changes the order in which features are reconstructed when the model is rebuilt.



Edit Definition

You can edit the definition of solid features on a part.

To edit the definition of a feature:

1 Right-mouse click the feature in the FeatureManager design tree and select **Edit Definition**.

Depending on the selected feature type, the appropriate dialog box appears.

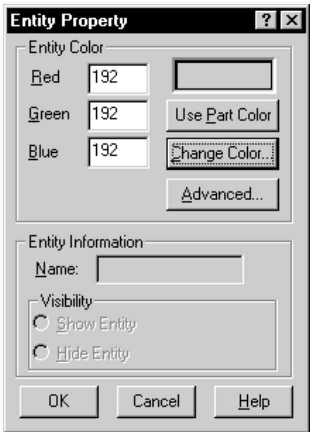
2 Edit the definition in the dialog box by specifying new values or options.

3 Accept your changes by clicking **Apply**, or **OK**; discard your changes by clicking **Cancel**.

Feature Properties

You can view and edit the properties of features, dimensions, faces, edges, sketches, axes, and planes. The dialog that appears corresponds to the type of item that is selected when you choose **Properties** from the right mouse menu.

To view properties:



1 In the FeatureManager design tree or in the graphics area, right-mouse click a feature, dimension, or item.

2 Select **Properties** from the menu; the **Properties** dialog appears for the type of item you selected.

Depending on the item selected, you may be able to edit the properties. In some cases, they are read-only.

To change feature colors:

1 Right-mouse click the feature in the FeatureManager design tree and select **Properties.**

2 In the **Feature Properties** dialog, click **Color**.

3 In the **Entity Property** dialog, click the **Change Color** button.

4 Select a color from the color palette or define a custom color.

5 Click **Advanced** to change advanced lighting values such as ambient, shininess, and transparency.

6 Click **OK**.

To change the color of a feature face:

1 Right-mouse click the feature face and select **Properties.**

2 In the **Entity Property** dialog, click the **Change Color** button.

3 Select a color from the color palette or define a custom color.

4 Click **Advanced** to change advanced lighting values such as ambient, shininess, and transparency.

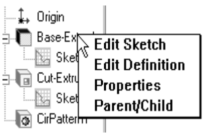
5 Click **OK**.

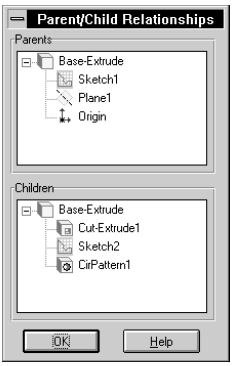
Parent/Child Relationships

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 extrusion 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, select the feature whose relationships you want to see.

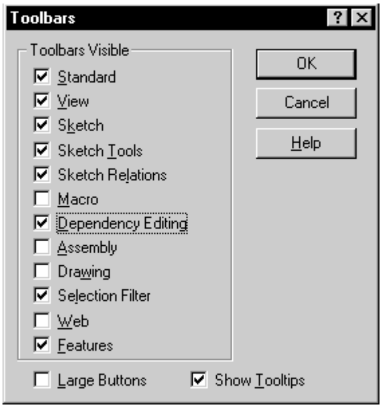
2 Click the right mouse button.

3 Select **Parent/Child** from the menu.

Dependency Editing

You can suppress a feature to work on the model with the selected feature temporarily omitted from the model. Features that depend on the selected feature are also suppressed.

The tools that suppress and unsuppress features are on the **Dependency Editing Toolbar**.



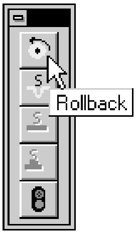
To display the Dependency Editing Toolbar:

1. Select **View**, **Toolbars**.
2. Click the **Dependency Editing** check box in the **Toolbars** dialog box.

Rollback

Reverts the model to the state it was in before the selected feature was created. Use this command to change a part at an earlier point of its development.

To revert to an earlier state:



1. Select a feature in the FeatureManager design tree or in the graphics display area.
2. Select the **Rollback** tool or click **Edit, Rollback**. The model reverts to the state it was in before the selected feature was created.
3. Add new features or edit existing features while the model is in the rolled-back state.
4. When you are finished, regenerate the model by clicking the **Rebuild** tool.

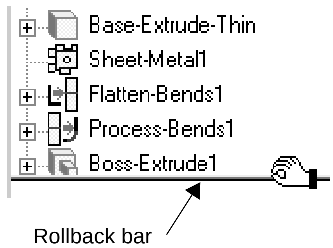
Using the Rollback Bar

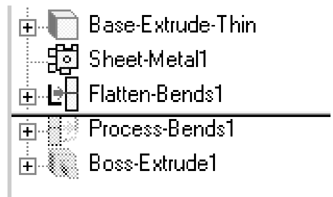
To rollback by dragging the rollback bar:

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.)





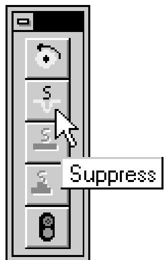
Note: You do not have to select a feature before you move the rollback bar. Note also that the rolled back icons are grey.

Suppress/Unsuppress

Suppresses a feature so you can work on the model with the selected feature temporarily omitted from the model. Features that depend on the selected feature are also suppressed. To suppress a feature:

1. Select the feature(s) in the FeatureManager tree.
2. Select the **Suppress** tool or click **Edit, Suppress**.

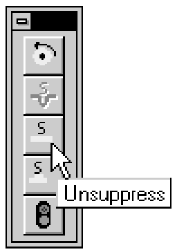
   The feature disappears from the model view and is grayed out on the FeatureManager design tree.



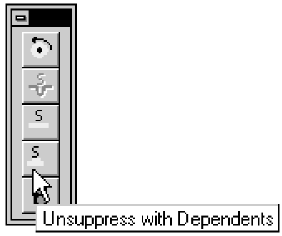
To unsuppress a previously-suppressed feature:

1. Select the suppressed feature on the FeatureManager tree.
2. Select the **Unsuppress** tool or click **Edit, Unsuppress**.

   Note: While a feature is suppressed, the only way you can select the feature to unsuppress it is through the FeatureManager design tree.



To unsuppress a previously-suppressed feature and its dependents:



1 Select the suppressed feature on the FeatureManager design tree.

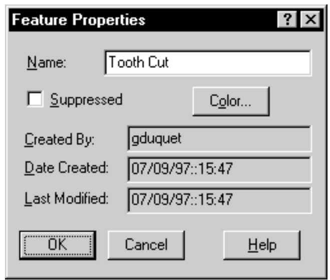
2 Select the **Unsuppress with Dependents** tool or click **Edit, Unsuppress with Dependents**.

To suppress the display of a feature using Properties:

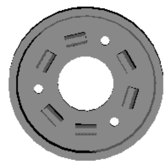
1 In the FeatureManager design tree, right mouse click on the desired feature.

2 From the right mouse menu, select **Properties**.

3 In the **Feature Properties** dialog box, click the **Suppressed** check box so that it contains an **check mark**, and then click **OK**.



The selected feature disappears from the display. In the FeatureManager design tree, the icon for the feature appears grayed out, indicating that the display of the feature is suppressed.



To redisplay the feature, repeat this procedure and click the **Suppressed** check box so that it does not contain an **check mark**.

Working with Part Configurations

Part configurations allow you to quickly and easily look at a part with certain features suppressed or with different feature parameter values. You can

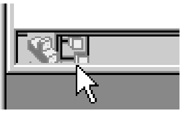
- Create a configuration
- View a configuration
- Open a configuration
- Change a configuration name or description
- Delete a configuration

Creating a Part Configuration

You create a part configuration that has suppressed features, and give the configuration a unique name. You can then view the part with the suppressed features by selecting the named configuration.

To create a part configuration:

1. Click the Configuration icon at the bottom of the FeatureManager design tree to change to the Configuration Manager view.

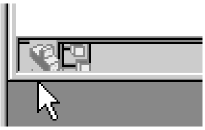


2. In the Configuration Manager tree, right-mouse click the part name and select **Add Configuration**. The **Add Configuration** dialog box appears.

3. Enter a **Configuration Name**, and add **Comments** if desired.
4. Click **OK**.

   The new configuration name appears in the tree.

5. To return to the FeatureManager design tree view, click the FeatureManager icon.



6. Suppress and/or hide components for the desired configuration.

Viewing a Part Configuration

You can switch between the various part configurations that you have created. You may need to look at a part without certain features. Or you may want to use a simplified version of a part in an assembly.

To switch to a different configuration:

1. Click the **Configuration** icon  to change to the Configuration Manager.
2. Right-mouse click the name of the configuration you want to view and select **Show Configuration**.

   The named configuration becomes the active configuration, and the display updates to reflect any differences in hidden or suppressed items.

Editing a Configuration

To edit a part configuration:

1. View the desired configuration.
2. Change to the FeatureManager view, and change the suppression and/or visibility as needed.

Deleting a Configuration

To delete a part configuration:

1. In the Configuration Manger, 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 <u>does not</u> delete any features. You cannot delete the configuration in use.

Changing the Properties of a Part Configuration

To edit the configuration properties:

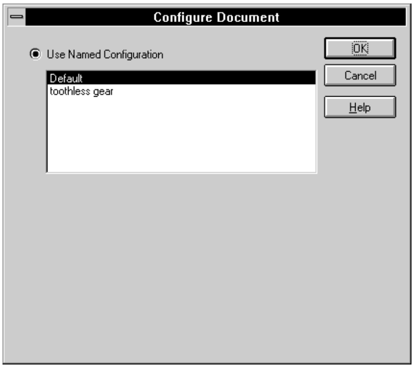
1. Right-click the configuration name, and select **Properties**. The **Configuration Properties** dialog box appears.
2. Edit the name, and comments as desired.
3. Click **OK**.

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 Part Configuration

When opening a part file, you can specify a configuration.

To open a configuration of a part:



1 Click **File**, **Open**.
2 In the **Open** dialog box, set the list of file types to **Part Files** or **All Files**.
3 Select a part, click the **Configure** check box so that it contains a **check mark,** and click **OK**.
4 Select the desired configuration.
5 Click **OK**.

   The part opens in the selected configuration.

Design Table

A design table allows you to build multiple configurations of parts by driving dimension values from cells 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.

Creating a Design Table

To use design tables, you must have Microsoft Excel installed on your computer.

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, as described below.

2 In the first *row*, enter the names of the dimensions or the features that you want to control.

   Dimension names are in the form ***Dimension name @ Feature*** or ***Sketch name***, or just the ***Feature name.*** For example, the default name for the depth of the first extrusion in a part is **D1@Base-Extrude1**.

To see a dimension's name, right-click on the dimension, then click **Properties**. You can also use the **Properties** command to assign more meaningful names to dimensions. Dimension names are case sensitive, so the name in the spreadsheet must match the name in the part exactly.

3 In the first *column*, enter the names of the design configurations that you want to create.

4 Fill in the dimension values in each column. For example:

	D1@Base-Extrude	D2@Cut1-Extrude
Housing, rev 1	45	88
Housing, rev 2	48	94

Note: You can unsuppress a feature by putting the feature name in the first row and **yes** in the appropriate cell. You can suppress a feature by leaving the related cell *blank*. For example, the following will suppress the feature **Cut1-Extrude** in the version named **rev 2**:

	D1@Base-Extrude	Cut1-Extrude
Housing, rev 1	45	yes
Housing, rev 2	48	

5 Save the spreadsheet.

6 Open the part where you want to use the design table.

7 Click **Insert, Design Table**.

8 Locate the spreadsheet file and click **OK**.

The spreadsheet appears in the part document, and the Excel menus and toolbars also appear.

9 Edit the table, if necessary. When you are finished editing, click anywhere outside the table. The SolidWorks menus and toolbars reappear.

To select one of the design table configurations, click the configuration icon at the bottom of the window. Right-click a configuration and select **Show Configuration**.

Note: The various design table instances can be associated to drawing views. Different design table instances can be displayed simultaneously in different drawing views.

Edit a Design Table

You can make changes to an existing design table.

To edit a design table:

1. Click **Edit, Design Table**. The table appears, and the Excel menus and toolbars also appear.
2. Edit the table. You can change the dimension values in the cells, add new rows to add design variations, or add new columns to control additional dimensions.
3. Click anywhere outside of the design table to close it.

Delete a Design Table

You can delete a design table in the currently active document.

To delete a design table:

1. Click **Edit, Delete Design Table**.
2. Confirm the deletion and click **OK**.

Note: Deleting the design table does not delete the configurations created by it.

Annotations

You can add annotations to your model to further clarify the information the part document provides. The following annotation types are available: notes, reference dimensions, weld symbols, surface finish symbols, datum feature symbols, datum targets, and geometric tolerances. See **Annotations** on page 6-34 of the *Drawing and Detailing* chapter for more information about creating and using the various symbols.

Lighting

You can adjust the direction, intensity, and color of light in the shaded view of a model.

To adjust the lighting in a shaded view:

1. Click **View**, **Lighting**.

   The **Light Sources** dialog opens with an **Ambient** light page and a **Directional** light page already in place. Additionally, you may select more **Directional Light** and **Spot Light** sources.

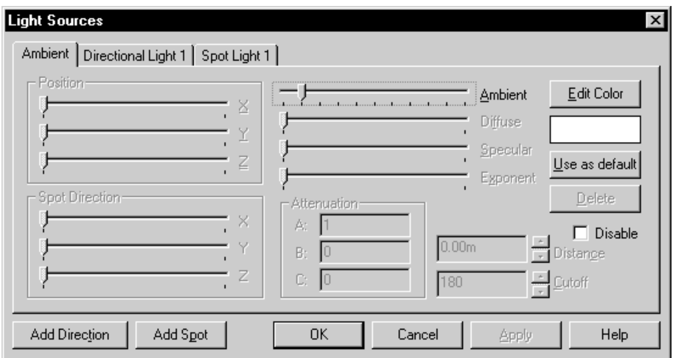
2. To add light sources, click the **Add Direction** or **Add Spot** buttons located at the bottom of the dialog box. (You may use a maximum of eight light sources in addition to the ambient light source.)

   For each new light source, a tabbed page is created on which you can define the characteristics of that light source. Move between the tabbed pages by clicking their tabs.

3. On the dialog page for each light source, select the color, direction, intensity, and other characteristics of the light source, as described below.

4. Click **Apply** to see the changes as you make them. (It is best to experiment with the values until you get the results that you want.)

5. Click **OK** to save the changes; click **Cancel** to end the session without saving the changes.



Light Sources Dialog Box

There are three light source options:

- **Ambient Light.** Light that is scattered so greatly that it seems to come 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. The light travels from one 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 and focused light that emits in a cone-shaped beam that is brightest at its center.

The light source characteristics and command buttons vary depending on the kind of light source you are working on.

Light source characteristics:

Position

- Adjust the slider bars to specify the amount of directional light striking the model surface from positions **X**, **Y**, and **Z**.

Move the slider to the right to increase the effect of each light source characteristic:

- **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 tends to bounce off the surface in a particular direction. Specularity indicates that the surface is shiny. A dull surface has no specularity; a glossy surface has high specularity.

Spot Light characteristics:

Additional characteristics are unique to the **Spot Light**:

- **Spot Position.** Sets the direction from which the light arrives. Adjust the slider bars to specify the amount of spot light striking the model surface from directions **X**, **Y**, and **Z**.
- **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 your model. Use the scroll arrows to increase or decrease the distance. (This value is represented by **d** in the attenuation factor equation.)
- **Cutoff**. The property that restricts the shape of the light cone. The maximum value is 180; reduce the value to reduce the spread of the light cone.

Command Buttons:

Edit Color. Defines the color of the light striking the model surface. Select a color from the color palette or click **Define Custom Color** to create a custom color.

Use as default. Preserves the **Direction**, **Ambient**, **Diffuse**, and **Specular** settings as the default settings for new models. (Not available for **Spot Light**.)

Delete. Removes the current light source page from the dialog box. (Not available for **Ambient**.)

Disable. Click this checkbox to disable the light options and characteristics of the current page.

Sending Part Documents

You can send the current part, assembly, or drawing document to another system using electronic mail.

To mail a document to another computer:

1. Click **File, Send**.
2. Enter your mail password, the mailing address, and any other information requested by your mail application.

To mail an assembly or drawing document to another computer:

1. With an assembly or drawing document active, click **File, Send**.
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 assembly or 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**. (The paper clip icon is removed.)
- Click a removed filename and click **Attach** to replace the file. (The paper clip icon is replaced.)
- To return all the removed files, click **Attach All**.

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.

5

Working with Assemblies

You can build complex assemblies consisting of many parts and subassemblies. This chapter describes the following assembly functions:

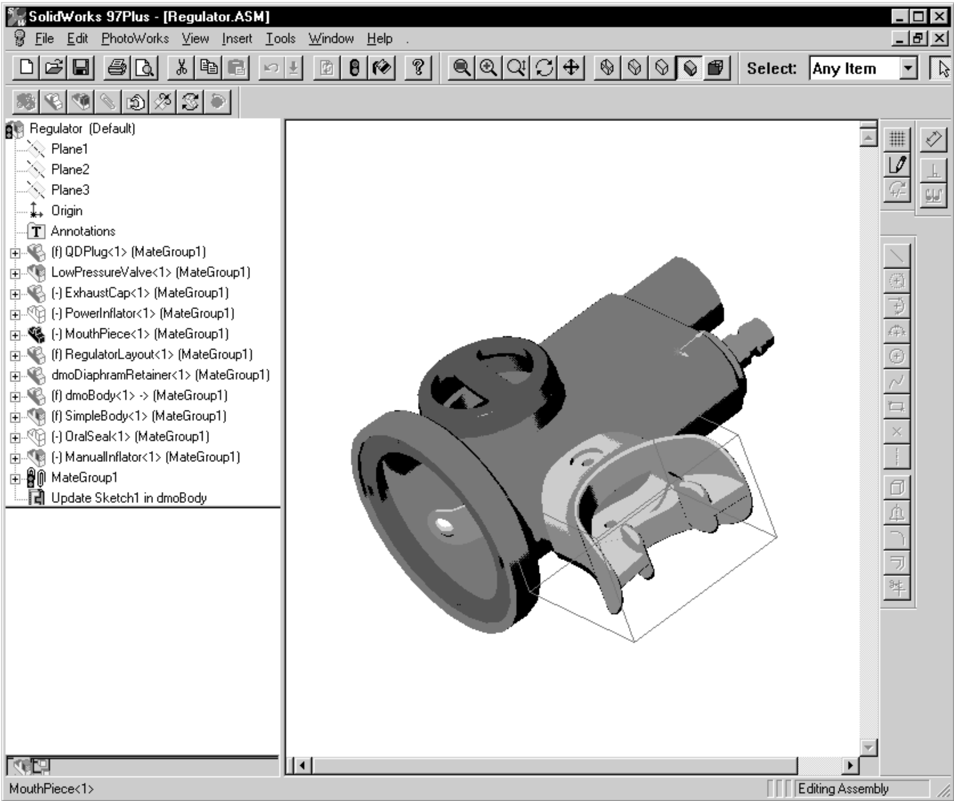
- Understanding assemblies
- Creating an assembly
- Assembly mating
- External references
- Exploding an assembly view
- Working with parts in an assembly
- Design methodologies
- Working with assembly features
- Creating molds
- Joining part volumes
- Simplifying large assemblies
- Using assembly configurations
- Opening assemblies
- Customizing the appearance of an assembly
- Mailing an assembly file

Understanding Assemblies

When a part is inserted into an assembly, it is referred to as a *component*. An assembly can be a *subassembly* 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, Subassemblies and Parts
- MateGroups and mating relations
- Component patterns
- Assembly planes, axes, sketches, and features
- 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 subassembly or component to view its detail by clicking the ⊞ beside the component or subassembly name.

You can use the same part multiple times within an assembly. Each component or subassembly has the suffix <***n***>. For each occurrence of the component in the assembly, the number ***n*** is incremented.

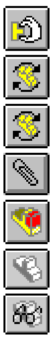
In the FeatureManager design tree, a part or subassembly name may have a prefix, providing information about the state of its relationships to other components. The prefixes are:

- **(–)** 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 Component

Show Component

For information about moving and rotating components in an assembly, see **Positioning Components in an Assembly** on page 5-7.

For information about mating components, see **Assembly Mating** on page 5-12.

For information about editing a part in an assembly, see **Working with Parts in an Assembly** on page 5-23.

For information about hiding and showing components, see **Simplifying Large Assemblies** on page 5-36.

Viewing the Assembly Hierarchy

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, subassemblies, assembly features, and mating relationships. The image of the assembly, including all its components (other than any suppressed 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 of an assembly:

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

2 Select **Show Hierarchy Only**.

The FeatureManager design tree displays only the parts and subassemblies, 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:

Right-mouse click the assembly name in the FeatureManager design tree, and select **View Dependencies**, *or*

Click **View, FeatureManager Tree, By Dependencies**.

To view an assembly by features:

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. You can:

- Add and delete components
- Position components
- Create component patterns
- Measure distances between components
- Check for interference between components

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.

You can add components to a new or existing assembly in any of these ways:

- Click **Insert, Component** and choose an existing part file.
- Drag and drop a part from the FeatureManager design tree of an open part window into the assembly window.
- Drag and drop a part from the Windows Explorer into the assembly window.
- Drag and drop a component from the FeatureManager design tree of the current assembly into the assembly window to add another instance of the component.

To add components by clicking Insert, Component:

1. With an assembly open, click **Insert, Component, From File**. The **Open** dialog box appears.
2. Browse to the directory that contains the component (or subassembly) you want to insert into the assembly.
3. Double-click the component (or subassembly) name (or click the component name, then click **Open**). The cursor changes to a cross ✢.
4. Position the cursor in the area of the assembly window where you want to place the component and click the left mouse button.

To add components by dragging and dropping from an open part window:

1. Open an assembly and open the part that you want to insert into the assembly.
2. Click **Window, Tile Horizontally** or **Tile Vertically.**
3. At the top of the FeatureManager design tree area of the part window, click the part icon or name.
4. Keeping the mouse button depressed, drag the part icon into the assembly window, then release the mouse button.

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 part.
3. In the **Explorer** window, click the part icon.
4. Drag the part icon into the assembly window and release the mouse button.

To add another component instance by dragging and dropping:

1. In the current assembly, click a component in the FeatureManager design tree.
2. Hold the left mouse button and drag the component into the graphics area.
3. Release the mouse button.

   A copy of the component is added to the assembly, and the instance number suffix is incremented.

Deleting a Component from an Assembly

To delete a component from an assembly:

1. Click the desired component in the display or in the FeatureManager design tree to select it.
2. Press the **Delete** key, or click **Edit**, **Delete**.
3. Click **Yes** at the prompt to confirm the deletion.

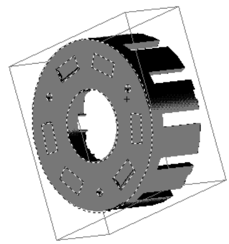
   The component and all its dependent items are removed.

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. The first component placed in an assembly is fixed by default; however, you can unfix it at any time.

As you add mating relationships, you can move the components within the unconstrained degrees of freedom, visualizing the mechanism's behavior.

When you select a component by its name in the FeatureManager design tree, the component is enclosed in a *bounding box* that indicates which component is selected. This indicator is helpful when you are working with a large and complex assembly.



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.

To fix a component:

1. In the FeatureManager design tree, right-mouse click the component or subassembly icon.
2. Select **Fix**.

   In the FeatureManager design tree, the prefix **(f)** appears next to the name of the fixed part or assembly.

To float (unfix) a component:

1. In the FeatureManager design tree, right-mouse click the component or subassembly icon.
2. Select **Float**.

   In the FeatureManager design tree, the prefix changes to reflect the status of the component before its position was fixed.

Rotating a Component

There are two ways you can rotate a component: freely around its centerpoint, or around an axis. Only components which 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 are maintained.

To rotate a component freely:

1. Click the desired component in the display or the FeatureManager design tree.
2. Click **Tools**, **Component**, **Rotate** or  on the Assembly toolbar.
3. Click and drag to rotate the component.

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 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.

Moving a Component

When you move a component, it retains any mating relationships it has with other components. You cannot move a component whose position is fixed or fully defined.

To move a component:

1. Click the desired component in the display or the FeatureManager design tree.
2. Click **Tools**, **Component**, **Move** or  on the Assembly toolbar.
3. Click and drag the component to move it.

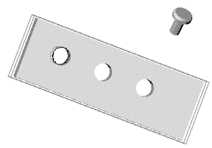
Note: Clicking **Pan**  on the View toolbar scrolls the view of the *entire assembly.*

Adding a Component Pattern

You can 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.

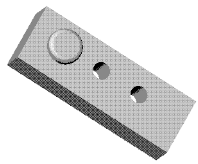
You can also define a pattern for placing components in an assembly in much the same way as you define a feature pattern in a part.

To use an existing pattern to place a pattern of components in an assembly:



1 Insert into your assembly a component that has a pattern feature.

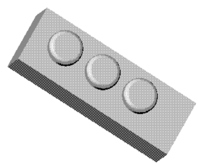
2 Insert into your assembly a seed component that you can mate with the pattern of the other part.



3 Mate the seed component to the original feature in the pattern. (See **Assembly Mating** on page 5-12 for more information.)

4 Click **Insert, Component Pattern**.

5 In the **Pattern Type** dialog, click **Use an existing feature pattern** and click **Next.**



6 In the **Derived Component Pattern** dialog box, click the **Seed Component(s)** box and select the seed component from the FeatureManager design tree.

7 Click the **Pattern Feature** box, then click the pattern feature in the FeatureManager design tree.

8 Click **Finish**.

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 Click **Define your own pattern**, choose **Linear** or **Circular**, and click **Next**.

4 Click the seed component.

5 Click the **Along Edge/Dim** box, click a model edge or dimension to indicate the pattern direction, and click **Reverse Direction** if necessary.

6 Specify the **Spacing** and **Instances**, and click **OK**.

Measuring Distances Between Selected Items

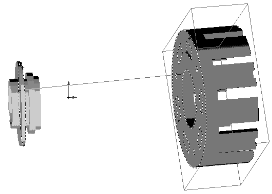
You may want to separate selected features of components in an assembly by specific distances, or measure sizes and clearances.

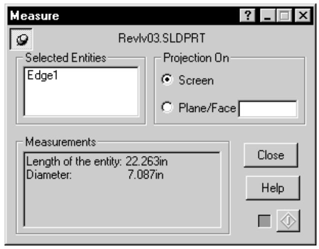
To determine the distance between surfaces, edges, or vertices of components:

1. Hold the **Ctrl** key and click the entities to measure. (You can either pre-select entities or select them after the dialog box is open.)
2. Click **Tools, Measure**.
3. To keep the dialog in place while you are working, click the push pin icon.

   To turn the **Measure** function on and off, click the switch in the corner of the dialog box.

   While the **Measure** function is **ON**, the cursor is a ruler with which you can select entities to measure.





4. 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. (When **Plane/Face** is selected the ruler icon has a small plane attached.)

   A line appears between the selected items, and appropriate distance values are displayed in the **Measurements** box.

   New measurements update dynamically when you change selections.
5. Click **Close** to close the dialog box.

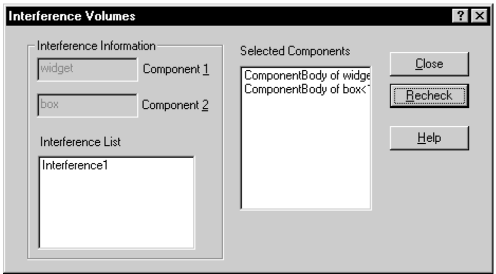
Adding Annotations to an Assembly

You can add annotations to an assembly. The following annotation types are available: notes, reference dimensions, weld symbols, surface finish symbols, datum feature symbols, datum targets, and geometric tolerances. See **Annotations** on page 6-34 of the *Drawing and Detailing* chapter for more information about creating and using the various symbols.

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 Either hold the **Ctrl** key and select two or more components in the assembly, or pick an assembly (top level or sub-assembly).

2 Click **Tools**, **Interference Detection**.

If there is interference,

- The **Interference Volumes** dialog box contains a list of interference occurrences (**Interference1**, **Interference2,** and so on). When you click an item in the **Interference list**, the related interference volume is highlighted in the graphics display area.
- 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 component.
- The names of the components that interfere with each other appear in the **Component 1** and **Component 2** boxes.

3 With the dialog box still open, you can reselect other components to check for interference. Right-click in the graphics area and select **Clear Selections**, select components for checking, then click **Recheck**.

4 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.

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.

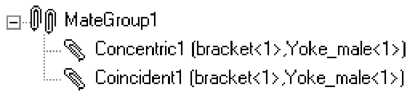
	Plane	**Cylinder**	**Line**	**Cone**	**Point**
Plane (planar face or plane)	coincident distance parallel perpendicular angle				
Cylinder (cylindrical face)	tangent	tangent concentric			
Line (linear edge, axis, or sketch line)	coincident parallel perpendicular distance	coincident tangent concentric	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, a mategroup is automatically created. This mategroup (which appears with a default name of **MateGroup1**) is shown on the FeatureManager design tree and has 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 as a sub-branch of 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 on these two parts, you will have one mategroup (**MateGroup1**) and two mates that are subordinate to it, **Concentric1 (*partnames*)** and **Coincident1 (*partnames*)**.

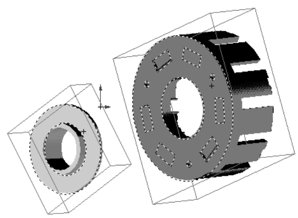


Assemblies can have multiple mategroups. Additional mategroups are automatically added when needed. This occurs when performing certain operations, such as adding a mate to a time-dependent feature in the assembly. See **Time-Dependent Features** on page 5-28 for more information.

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

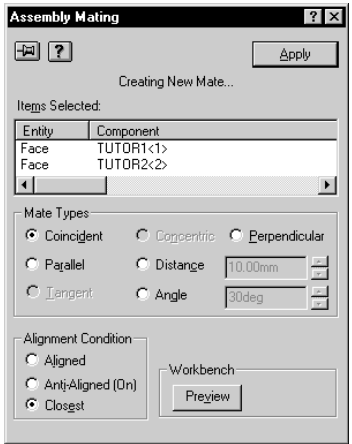
To mate components in an assembly:



1 Select the desired faces, edges, vertices, or reference planes on the components.

   Note: Set the **Selection Filter** to the desired entity type to make selection of the desired entities easier.

2 Click **Mate** on the Assembly toolbar or select **Insert**, **Mate**.



3 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.

4 Verify the contents of the **Items Selected** box. The selected entity types and component names are listed in this area.

5 Select the desired **Mate Type**. (Only the mating types that are valid for the selected features are available.)

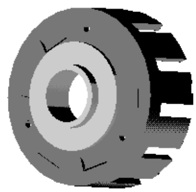
6 Select an **Alignment Condition**.

- **Anti-aligned** means that the selected faces are on *opposite* sides of a plane parallel to them.
- **Aligned** means that the selected faces are on the *same* side of a plane parallel to them.
- **Closest** means that the selected faces may be either aligned or anti-aligned, depending on which condition can be satisfied with the least part movement.

7 In the **Workbench** area of the dialog box, click **Preview** to see what the assembly will look like with the mate performed.

8 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.

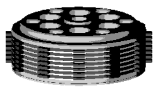
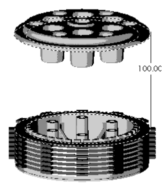
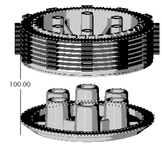
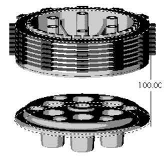
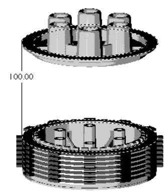
9 Click **Apply** when you are satisfied with the mate. When you close the **Assembly Mating** dialog box, the cursor changes from the paperclip mating cursor back to the select cursor.



Modifying a Mating Relationship

You can change these characteristics of a mating relationship:

- **Distance** between the mated components for distance mates
- **Angle** between mated components for angle mates
- **Alignment (aligned** or **anti-aligned)** on mates which imply a direction
- **Dimension direction** to flip (reverse) the direction in which a distance is measured

Offset Distance	Component Alignment	Flip Dimension To Other Side	Example
0	anti-aligned	—	
100mm	anti-aligned	—	
100mm	aligned	—	
100mm	anti-aligned	X	
100mm	aligned	X	

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. Click **Apply** to make the change.

Note: To change only the dimension value of a distance mate, double-click the relationship in the FeatureManager design tree, then double-click the dimension to edit.

To make a major modification to a mating relationship:

If you want to make a major modification to a mating relationship (such as making a mate with a third, different component) you can perform this modification without deleting the mate.

1. In the FeatureManager design tree, expand the MateGroup to view the mate you want to change.
2. Right-click the mate and select **Edit Definition**. The **Assembly Mating** dialog box appears.
3. In the **Entity** column of the **Items Selected** area, double-click the entity of the component mate you want to change.

   The system prompts: **You may now reselect another entity to replace this**.

4. Click **OK**.
5. In the assembly window, select the replacement entity. The new **Entity** and **Component** name appear in the **Assembly Mating** dialog box.
6. Click **Preview** to preview the new mate.
7. If the mate is satisfactory, click **Apply**. If the mate is not satisfactory, **Undo** and repeat the operation starting at Step 3.

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 to select it.
2. Press the **Delete** key, or click **Edit**, **Delete**.
3. Click **Yes** at the prompt to confirm the deletion.

External References

An *external reference* or *dependency* is created when a component, assembly, or drawing refers to another component, assembly, or drawing. If the referenced components change, the affected features change also. Components with references have an arrow next to their names in the FeatureManager design tree.

In an assembly, you can create an in-context feature on one component which 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 **Working with Parts in an Assembly** on page 5-23 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 used to create the cavity. If you change the design part, the cavity changes to match it. See **Creating Molds** on page 5-32.

You can change or break external references on features by editing the features and changing or deleting the relationship.

To set external reference status:

You can set the external reference status of a part to be read-only. If a part is set to read-only, other SolidWorks users can use this part in their designs but cannot change it.

1. Click **Tools, Options**.
2. Select the **External References** tab.
3. Click the check box **Open referenced documents with read-only access.**

To search a document folder list for external references:

It is important to know whether or not a SolidWorks document is an external reference in another document. You can search for external references in a document folder list as follows:

1 Click **Tools, Options**.

2 Select the **External References** tab.

3 Click the check box **Search document folder list for external references**.

4 Click **Add**. The **Choose Directory** dialog box appears.

5 Browse to the desired directory.

6 Click **OK**.

Note: A component whose absorbed components have an external reference has the suffix **->** next to the component name if the external reference is in-context. If the external reference is out-of-context, the suffix is **->?**.

Concurrent Document Access - Write Access Notification

If you try to open a document that is currently opened by another SolidWorks user, you are informed that the document is in use by that person. You are asked if you want to make your own copy of the document. Click **Yes** to make a copy, or **No** to cancel the operation.

File Reload

Click **File, Reload** to:

- Reload a read-only file for modification
- Reload a file opened for modification
- Replace the contents of the open document and all active references to another file

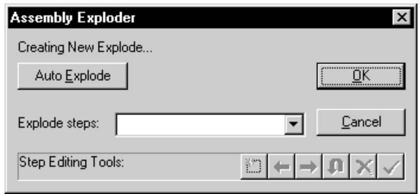
Exploding an Assembly View

Sometimes it is 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 with which it is created.

Creating an Exploded View

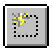
To explode an assembly:

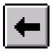


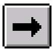
1 Click **Insert, Exploded View.**

The **Assembly Explode** 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 to current step

 **Delete -** delete current explode step

 **Apply -** apply/update current explode step

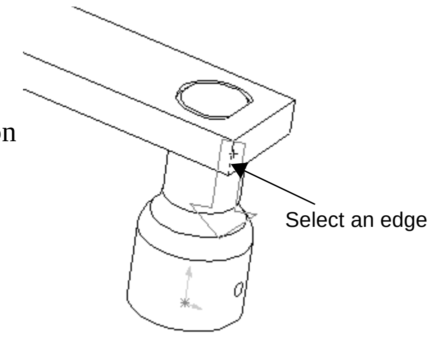
2 Click **New** . The **Assembly Exploder** dialog box expands.

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**.



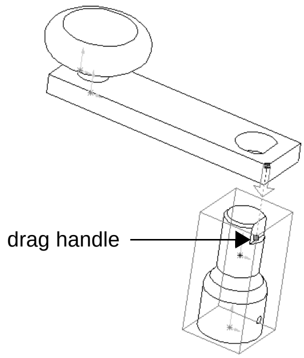
Note: If you are working on a complex assembly, it may be easier to work in shaded view mode.

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.

5 Click **Apply** ☑.

The component explodes from the assembly. Note the green drag handle.



6 Drag the green handle to position the component as desired.

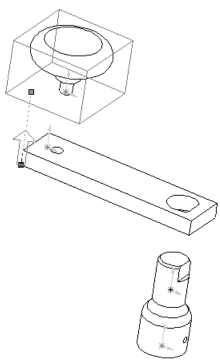
7 Click **Apply** ☑ to confirm this step.

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**.

If you want to add more explode steps to the exploded view, click **New**, repeat Steps 3 through 7 until all the steps are complete, then click **OK** to finish the exploded view.

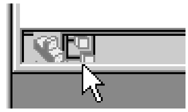
Remember to click **Apply** after defining each step.



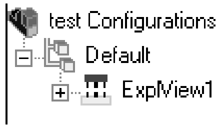
9 To collapse the assembly, click **Edit, Rebuild,** or right-click anywhere and select **Collapse**.

Exploding and Collapsing an Exploded View

To explode and collapse an exploded view:



1. Click the **Configuration** icon at the bottom of the window. The Configuration Manager tree is displayed.



2. Click the plus sign **+** beside the desired configuration.
3. Double-click the **ExplView** feature. The assembly explodes.
4. To collapse the exploded view, either click **Edit, Rebuild** or 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 enclosed in bounding boxes.
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 Configuration view to FeatureManager view by clicking the FeatureManager icon at the bottom of the window.
3. Right-click the component to reposition, and select **Show Explode Steps**.
4. Proceed as described above.

Using AutoExplode

The AutoExplode feature 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.

   Note: You can edit an **AutoExplode** view in the same manner as any exploded view.

Working with Parts in an Assembly

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
- ❑ Verify which part files are used in the assembly
- ❑ Copy the part files to a new directory
- ❑ Replace a part

Creating a Part in an Assembly

You may want to design a part while in an assembly so that you can use the geometry of the assembly while designing the part. The new component 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 cursor appears with a box next to it.
3. Select a plane or face on which to position the new part. Plane1 of the new part is mated coincident to 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.
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.

The new part is fully positioned by an **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 5-17.

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 or any part. Features may use any end condition, such as **Up to Surface**, on another component (See Chapter 3, *Creating Features,* for details on end conditions.)

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 for example, ***partname*** **in** ***assembly*****.asm**. Note that the message in the status bar indicates that you are now editing the *part* document even though the entire *assembly* is visible. The assembly component that you are editing changes to pink, and the others turn gray.

Note: You can change the colors. Click **Tools**, **Options**, select the **Color** tab, and click **View System Defaults**. Select **Edit Part in Assembly** (or **Non Edit Parts in Assembly**) from the **System** list box. Click **Edit** and change the color as desired.

2 Make necessary changes to the part.

3 When you finish editing the part, right-click the assembly name in the FeatureManager design tree or anywhere in the graphics area, and select **Edit Assembly:*****assemblyname*** 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 window, and select **Open <*filename*>**.

The part file opens in a separate window.

2 Edit the part as desired.

When you return to the assembly window, the assembly automatically updates to reflect the edits.

To open an 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 to which the feature has a reference opens in a separate window.

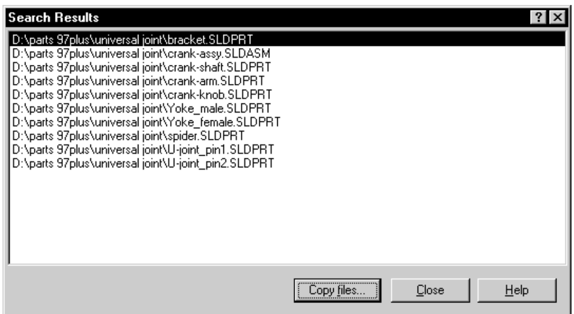
Verifying Which Part Files Are Used in an Assembly

You can check which part and subassembly files an assembly uses. This is useful if you have several versions of a part file and you want to make sure that you have used the correct one in the assembly.

To list the part files:

In an assembly, click **File**, **Find References**.

The **Search Results** dialog box appears with a list of the parts used in the assembly, including the full path names.



Copying the Part Files in an Assembly to a New Directory

You can copy all part and subassembly files used in an assembly to a new directory. This is useful if you want to modify parts and preserve the original parts without change.

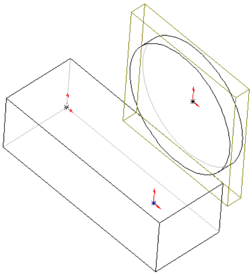
To copy the part files:

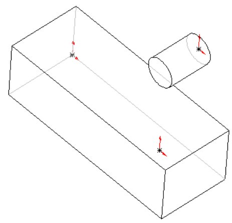
1. In the **Search Results** dialog box, click **Copy Files**.
2. In the **Choose Directory** dialog box, browse to the destination directory.
3. Click **OK**. The assembly file and all the referenced files are copied to the new directory.

Replacing a Part in an Assembly

You can replace a part in an assembly with another part. If the parts are similar, any mates used in the original part are applied to the replacement part.

To replace a part in an assembly:





1. In the FeatureManager design tree or the graphics area, right-click the part you want to replace, and select **Component Properties**.
2. In the **Component Properties** dialog box, click **Browse**.
3. In the **Open** dialog box, browse to the replacement part file. If desired, click the **Preview** checkbox to view the part in the **Preview** window.
4. Click **Open**.
5. Click **OK** in the **Component Properties** dialog box.

The new part replaces the original part, and the previous mating relationships are maintained if possible.

Saving an Assembly and Its Parts

When you save an assembly, you save the assembly and all referenced parts which have been changed. If you do not want to modify an original part that you have modified and used in an assembly, copy the part (**Save As**) with a different name and use the copy in your assembly.

Design Methodologies

You can create an assembly using bottom-up design, top-down design, or a combination of both methods.

Bottom Up Design

Bottom up design is the traditional method. In bottom up design, you insert components into an assembly and perform various operations on them. For example, you take two parts that have already been created, insert them into an assembly, and mate the parts 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.

Top Down Design

Top down design is different because you start your work in the assembly. You define points, known locations, etc. and then design the components around them. You start with a layout, define size, planes, etc. and design the parts referencing these definitions.

As an example of top-down design, you can create a part and build a fixture based on this part. Because you built the fixture based on references to the dimensions of the part, if you change a dimension of the part, the dimensions of the fixture automatically change and the new, changed fixture is correct for the modified part.

Another example of top-down design is an enclosure which expands or contracts automatically as the size of its contents changes.

Additional information on top-down design may be found in the section **Using an Assembly Layout Sketch** on page 7-15.

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.

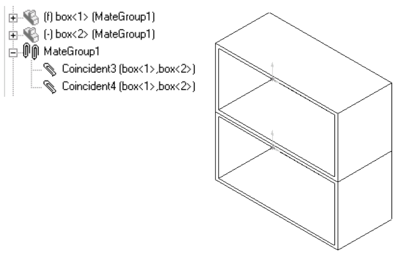
Time-dependent features include:

- Assembly features (assembly cuts and holes)
- 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**.

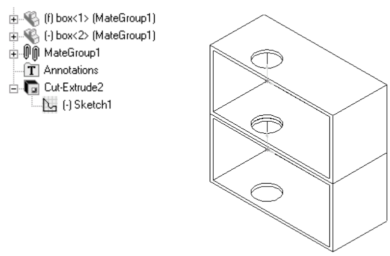
Mates are solved together as a system by default. This system is called a *mategroup*. Each assembly has a default mategroup, **MateGroup1**. Within a mategroup, the order in which mates are added does not matter; all the mates are solved at the same time. In a bottom-up design, only one mate group is required. All mates added between components are placed in **MateGroup1**.

In this example, the top and bottom faces of the components are mated, as is one side face of each component.

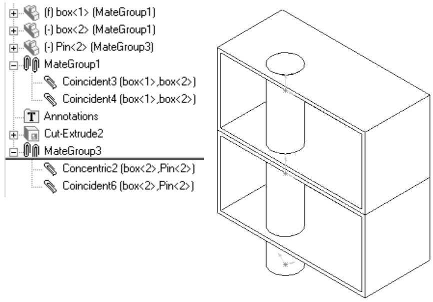


Then a hole is cut through both components. This hole is cut as an *assembly feature*, not as a modification to the original parts.

If you then mate new components to the results of this time-dependent feature, a new mate group is required for that mate. This is because the new component can only be positioned after the time-dependent feature has been updated.



Continuing this example, a pin is positioned in the assembly hole.



The position of a component in an assembly is fully defined in the first mate group which contains mates to it. If it is referenced in later mate groups, it is treated as a fixed component. In this example, the top is fixed in MateGroup2 while the pin is not (the pin moves to the top).

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 relative to one another in **MateGroup1** and you want to complete the positioning by adding a mate which references a time-dependent feature which is dependent on **MateGroup1**. This could be done by mating the bottom face of the pin coincident to the top face of the upper part before adding the assembly hole, then adding the assembly hole, then adding the concentric mate between the pin and the assembly hole. In this case, the coincident mate must be 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, SolidWorks automatically detects that the coincident mate can be moved forward and it will move the mate when the concentric mate is performed. However, in more complicated cases you may need to determine which mates to move forward. You can do this by deleting the mates which position the component you wish to move from the earlier mate group, then adding the mate to the time-dependent feature (which creates the additional mate group), then adding more mates to fully position it.

When positioning a component to geometry which depends on a time-dependent feature, it is good design practice to mate the time-dependent feature first, then add additional mates to complete the positioning.

Working with Assembly Features

While in an assembly, you can create features that exist *in the assembly only* and can affect one or more components. You determine which parts you want the feature to affect by setting the scope. In general, when you want to change a single part in an assembly, it is better to edit the part in context than to use an assembly feature.

Setting the Scope

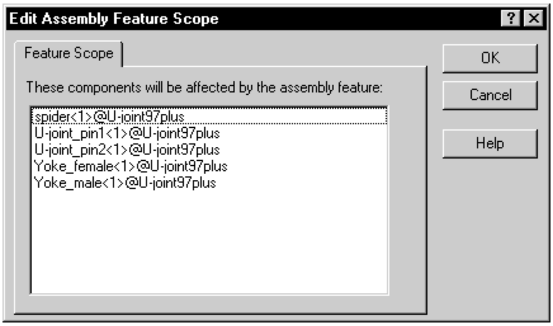
When creating an assembly feature, you must first determine which parts you want to be affected by the feature. By setting the scope, you can selectively choose the parts you want the feature to impact. For example, a hole cut through the assembly would only go through the parts you select.

The default setting is that assembly features affect *all* parts in the assembly.

To set the scope:

1 In an assembly, select **Edit**, **Feature Scope**.

2 Select the components that this assembly feature 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.

3 Click **OK**.

Note: To remove a component from the list when you have the **Edit Assembly Feature Scope** dialog box open, press **Ctrl** and click the component again in the graphics area or in the FeatureManager design tree.

Creating an Assembly Feature

Assembly features affect the assembly only; the part files are not affected. You must set the scope before you can create an assembly feature. The **Assembly Feature** menu has two selections: **Cut** or **Hole**.

Before adding the assembly feature, position the components, and set the feature scope to the desired parts as described on the previous page.

To create an assembly feature cut:

1. Open a sketch on a face or plane, and sketch a profile of the cut. (To make a revolve, you must also sketch a centerline.)
2. Click **Insert**, **Assembly Feature**, **Cut** and select either **Extrude** or **Revolve**.
3. Set the **Depth**, **Type**, and other options as needed in the **Extrude Feature** dialog box.
4. Click **OK**.

To create an assembly feature hole:

1. Click 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

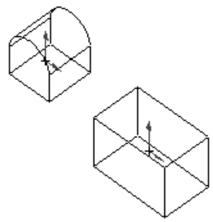
A mold consists of a base containing a cavity with the shape of the desired part. When you create a mold you must account for the *scaling factor*, that is the amount the material in the mold shrinks or expands as it solidifies. The scaling factor depends on the type of material used and the shape of the mold, and is expressed as a percentage (+/- 20%) of the linear size (not volume) of the cavity part.

SolidWorks sizes the cavity by the specified scaling factor using the following formula:

$$\textit{cavity size} = \textit{part size} * (1 + \textit{scaling factor})$$

To create a mold cavity:

1. Create the design part you want to mold.

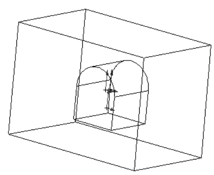


2. Create the mold base.

   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.

4. Position the design part *inside* the mold base, using mating relationships as needed.



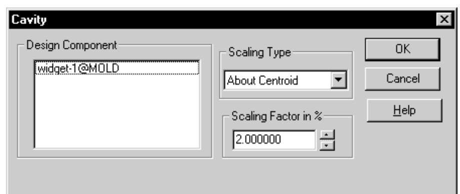
5. In the assembly window, right-click the mold base in the graphics area or in the FeatureManager design tree and select **Edit Part**,

   - or -

   Click the mold base and click **Edit Part** .

6. In the FeatureManager design tree, select the design part.

7. Select **Insert**, **Features, Cavity**.



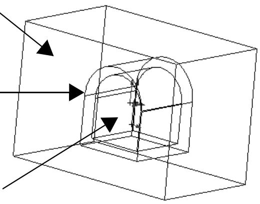
8. In the **Cavity** dialog box, enter the **Scaling Factor in %** (to a maximum of 20%).

   A positive value expands the cavity, a negative value shrinks the cavity.

9. Select the point about which scaling occurs in the **Scaling Type** field.

10 Click **OK**.

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.

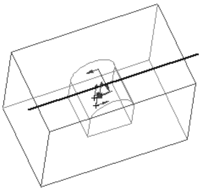


To cut the mold base to make two mold halves:

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 which has another part as its first feature).

2 Click to select a face on the mold base.

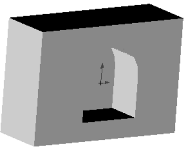


3 Open a sketch and sketch a line that bisects the mold base as desired, that is, a line representing the parting plane.

4 Click **Insert**, **Cut**, **Extrude**. Leave the **Flip Side to Cut** check box unchecked in the **Extrude Cut Feature** dialog box.

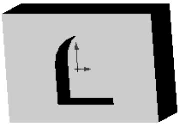
5 Click **OK**.

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, repeat this process, reversing the direction of the cut by checking the **Flip Side to Cut** check box in the **Extrude Cut Feature** dialog box.



An additional mold example is found in Chapter 14, *Creating a Mold*, in *Learning to Use SolidWorks 97Plus Tutorial*.

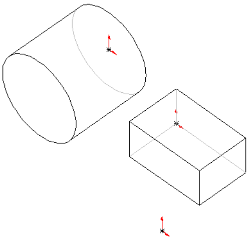
Creating Drafts for Molds

You may wish to add draft angles to parts when creating a mold. Refer to **Split Line** on page 3-39 and **Pattern/Mirror** on page 3-29 for information about splitting faces and adding draft angles.

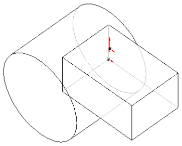
Joining Parts

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.

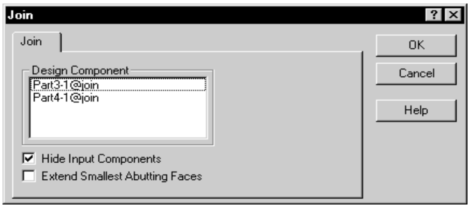
To join parts:



1. Create the parts that you want to join.
2. Create an assembly containing the parts.
3. Position the parts as desired in the assembly. The parts may either touch each other or intrude into one another.
4. **Save** the assembly but do not close the window.
5. Click **Insert, Component, New** to insert a new part into the assembly, and **Save** it.
6. Click a face on one of the components.



7. Click **Insert, Features, Join**. The **Join** dialog box appears.



8. In the FeatureManager design tree, click the components you want to join.
9. Check **Extend Smallest Abutting Faces** if you want to join parts that just touch each other.

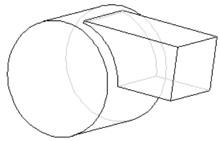
   The system finds all abutting faces that touch but do not intrude into each other's space. It then finds the affected face with the smallest area, extends it into the other component, and fills in the resultant gap.
10. Click **Hide Input Components** to improve graphics rendering.

    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 grey outline.

11 Click **OK**.

12 **Save** the part.

13 Click **Rebuild** to view the newly joined parts.



14 To exit from **Edit Part** mode, right-click anywhere and select **Edit Assembly.**

To identify which parts are used to create a joined part:

In the FeatureManager design tree, right-click the **Join** feature of the joined part and select **Edit Definition**.

- or -

Select the joined part, and click **File, Find 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, **Ctrl-**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. Any changes made to the original parts are reflected in the joined part.

Simplifying Large Assemblies

Large assemblies can be complex, consisting of many components. Some assemblies have subassemblies consisting of many parts, making it difficult to work with the internal parts. You can simplify a complex assembly by suppressing the use of components or turning off their display. This allows you to view underlying components of the assembly.

The simplified view of the assembly is called a *configuration*. You can assign a name to the configuration so you can switch between configurations as desired. For more information about configurations, see **Using Assembly Configurations** on page 5-39.

Assemblies can also contain simplified versions of components, that is, parts with suppressed features.

Suppressing vs. Hiding

You can view an assembly with certain components removed from the view. This allows you to focus on the components needed for the task at hand. You can simplify an assembly by either *suppressing* or *hiding* components.

Suppressing components

Suppressing a component *removes* it from the active assembly configuration.

A suppressed component is removed from memory, so rebuild speed and display performance are improved. Because of the reduced complexity, evaluation occurs faster. Since the component is no longer an active part of the assembly, its mates are not solved, it is not considered in mass properties, it is not included in the bill of materials (BOM), and it is not considered in a global interference check.

Suppressing of components should be used with care as it can change the result of a rebuild.

Hiding components

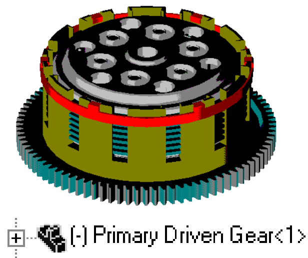
Hiding a component turns off *only the visibility* of the component in the active assembly configuration.

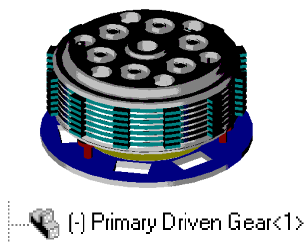
Because hiding the component affects the display only, it does not affect the rebuild or evaluation speed. Display performance improves however. Since the component is still an active part of the assembly, its mates are solved, and it is considered in mass properties, bill of materials, and global interference checking.

Suppressing Components

Large, complex assemblies contain a great amount of data. When you specify the use of only the components on which you want to work, the assembly displays faster and you make more efficient use of your system resources. However, suppressing components also suppresses mates and assembly features related to those components. Suppression, in cases like this, can change the regeneration behavior of the assembly. Also, models with suppressed components can sometimes result in conflicts when the components are unsuppressed. Therefore, use suppression carefully when modeling.

To suppress a part or subassembly:





1. In the FeatureManager design tree, right-click the desired part or subassembly, and select **Component Properties**.
2. In the **Component Properties** dialog box, select the **Suppress** check box, and click **OK**.

   The selected part or subassembly is removed from the assembly. In the FeatureManager design tree, the icon appears grayed out, indicating that the application is not using the part.
3. To unsuppress the part or subassembly, repeat this procedure, *deselecting* the **Suppress** check box.

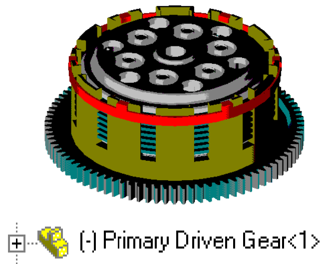
   Note: If you rebuild the assembly, the application generates it without the parts you suppressed.

You can also use the **Suppress**  and **Unsuppress**  icons on the Dependency Editing toolbar for this purpose.

Hiding Components

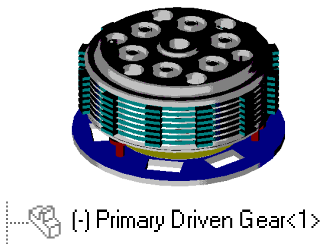
You may want to turn off the display of certain components, so you can see the underlying components and improve the display performance. However, you may still want the component immediately available. You can do this by *hiding* specified parts. Hiding components does not affect the regeneration of the assembly.

To hide a part or subassembly:



1. In the FeatureManager design tree, right-click the desired part or subassembly, and select **Component Properties**.
2. In the **Component Properties** dialog box, deselect the **Show Model** check box, and click **OK**.

   The selected part or subassembly is removed from the display. In the FeatureManager design tree, the icon for the part appears as an outline, indicating that the display of the part or subassembly is turned off.



3. To redisplay the part, repeat this procedure, *selecting* the **Show Model** check box.

   Note: If you rebuild the assembly, the application still uses the undisplayed parts to generate the assembly.

You can also use the **Hide**  and **Show**  icons on the Assembly toolbar for this purpose.

Using Assembly Configurations

Configurations allow you to build an assembly with various combinations of components suppressed or hidden. You can have several configurations of an assembly and can quickly switch between them.

This section describes how to:

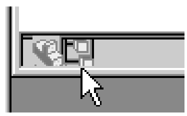
- Create an assembly configuration
- View an assembly configuration
- Change the properties of an assembly configuration
- Edit an assembly configuration
- Delete an assembly configuration

Creating an Assembly Configuration

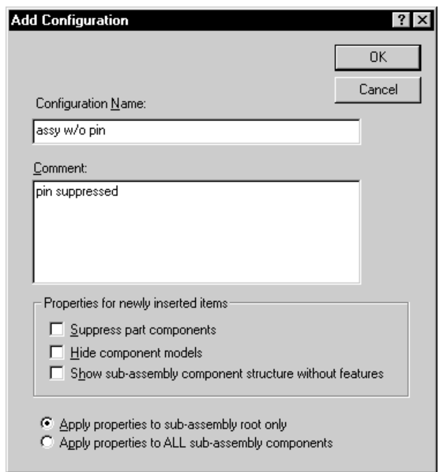
You create a configuration that has suppressed parts or subassemblies, and name the configuration. You can then view the assembly with the suppressed parts by selecting the named configuration.

To create a configuration:

1. Click the **Configuration** icon at the lower left of the window to change to the Configuration Manager view.



2. In the Configuration Manager tree, right-mouse click the assembly name and select **Add Configuration**. The **Add Configuration** dialog box appears.



3. Enter a **Configuration Name**, and add **Comments** if desired.

4 Select **Properties for newly inserted items** as desired:

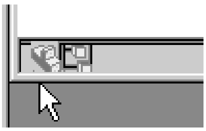
- Click **Suppress part components** or **Hide component models** if you want newly added parts to be added in the suppressed or hidden state.
- Click **Show sub-assembly component structure without features** to simplify the detail for all added sub-assemblies in the FeatureManager design tree.

5 Select the way properties are applied:

- Click **Apply properties to sub-assembly root only** to apply the properties selected in Step 4 to only the top subassembly components. All subassembly child components will inherit the properties specified in the configuration of the subassembly.
- Click **Apply properties to ALL sub-assembly components** to apply the properties selected in Step 4 to all added components. For example, if you want all components of newly added subassemblies to be suppressed, regardless of the properties specified in the subassembly configuration, choose this option.

6 Click **OK**.

The new configuration name appears in the tree.



7 To return to the FeatureManager design tree view, click the FeatureManager icon.

8 Suppress and/or hide components for the desired configuration.

9 **Save** the assembly. The new named configuration is created.

Viewing an Assembly Configuration

To switch to a different configuration:

1 Click the **Configuration** icon  to change to the Configuration Manager.

2 Right-mouse click the name of the configuration you want to view and select **Show Configuration**.

The named configuration becomes the active configuration, and the display updates to reflect any differences in hidden or suppressed items.

Changing the Properties of an Assembly Configuration

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/or properties as desired.
3. 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**.

Editing an Assembly Configuration

To edit an assembly configuration:

1. View the desired configuration.
2. Change to the FeatureManager view, and change the suppression and/or visibility as needed.

Deleting an Assembly Configuration

To delete a configuration:

1. In the Configuration Manger, 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.

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 any items into the system default features or into the components.

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.
- If you rollback an assembly pattern, the components generated by that pattern are rolled back.
- You cannot rollback to a component, that is, after some of the components are added, or before others. This is because the components are not time dependent. You should use hide or suppress for this purpose. See **Simplifying Large Assemblies** on page 5-36 for more information.
- If you rollback to layout features before the components, all the components are rolled back as well.

Opening Assemblies

You open an assembly file in much the same way you open a file in other Windows applications. When you open an assembly, the application accesses the information in all the part files that the assembly references. It then builds the assembly using this information.

You can open a simplified version of the assembly, an *assembly configuration*, displaying only the parts you want to see. You can also use simplified versions of parts, or *part configurations*, when opening an assembly. This allows you to focus on specific part features and see how they fit in an assembly.

This section describes how to open:

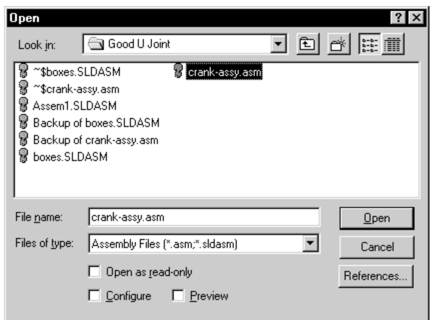
- ❑ The last saved configuration
- ❑ A named assembly configuration
- ❑ The assembly structure only

Note: Whenever you open an assembly, if a part or subassembly 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.

Opening the Last Saved Configuration

To open the last saved 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 the desired assembly, and click **Open**.



Opening a Named Configuration

You can open a named assembly configuration, that is, an assembly showing only the specific components or subassemblies you want to work on. Refer to the section, **Using Assembly Configurations** on page 5-39 for information on configurations.

You can also open an assembly using the simplified version of each part, that is, the parts configurations in which you have suppressed certain features. See **Suppress/Unsuppress** on page 4-6 for information about suppressing the display of a feature.

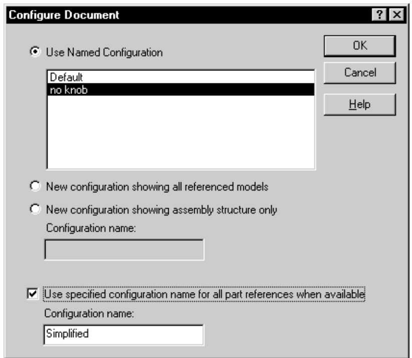
To open a named configuration of an assembly:

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 the simplified version of the parts, click **Use specified configuration name for all part references when available**, and enter a part configuration name.

Note: If the referenced configuration 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 5-40 for information.

Opening the Assembly Structure Only

By opening only an assembly structure, you can immediately view the FeatureManager design tree of an assembly without waiting for the components to be loaded. The image of the assembly does not appear. You can then display the parts you want to work with by unsuppressing only the ones you want. Refer to the section, **Simplifying Large Assemblies** on page 5-36 for information.

To open the structure of an assembly:

1. Click **File**, **Open**.
2. In the **Open** dialog box, set **Files of Type** to **Assembly Files** or **All Files.**
3. Select an assembly and click **Configure**.
4. Click **Open**.
5. In the **Configure Document** dialog box, select **New configuration showing assembly structure only** and enter a name.
6. Click **OK**.

   The FeatureManager design tree for the assembly appears; the image of the assembly does not appear.

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** 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, including the referenced part file changes.

To change the color of a part instance in an assembly:

1 Right-click the component instance in the graphics area or in the FeatureManager design tree, and select **Edit Properties.**

2 Specify RGB values, and click **OK**,

- *or* -

Click **Color**, click **Change Color**, choose a color from the palette, and click **OK**.

3 Click **Use Assembly** or **Use Component** to choose between the original part color and the component instance color used in the assembly.

4 Click **Advanced** to set other properties for the selected component, such as **Transparency** and **Shininess**.

5 Click **OK**.

The changes you make to the properties of a component affect only the selected instance of the component.

Mailing an Assembly File

You can send assembly files (**.sldasm** files) by electronic mail. You can choose to send the assembly itself or you can send the assembly and all referenced documents.

Note: 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**, all are included. You can then selectively delete part files from the mailing.

- Select the part files you do not want to send and click **Remove**.
- Select and click **Attach** to reattach them.
- Click **Attach All** to reattach all unselected part files.

3 Click **OK**. Your mail utility dialog box appears.

4 Enter any information required by your mail utility.

6

Drawing and Detailing

The SolidWorks 97Plus application allows you to build and detail complex parts and assemblies. Parts, assemblies, and drawings are linked documents; any changes that you make to the part or assembly change the drawing document.

You can insert dimensions and notations from the model or add reference dimensions directly to the drawing. Dimensions are always consistent with the model and change as the model changes.

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, your 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. Notes, symbols, drafted curves, centerlines, center marks, geometric tolerances, and other notations augment the drawing.

This chapter describes:

- ❑ Selecting drawing, detailing, and page set-up preferences
- ❑ Creating drawings
- ❑ Creating views
- ❑ Dimensioning drawings
- ❑ Annotating drawings
- ❑ 2D sketching

Detailing and Page Setup Preferences

SolidWorks provides a variety of options to customize your drawings to detailing standards and to conform to the requirements of your printer or plotter.

To customize your SolidWorks 97Plus drawings, you can:

- Select drawing and detailing options by clicking **Tools, Options** and then reviewing the options available on the **Detailing**, **Drawings**, **Crosshatch**, and **Line Font** tabs. See Appendix A for information about the SolidWorks 97Plus options.
- Set the page properties to meet your printer or plotter requirements. Click **File**, **Page Setup** to set up
  - Paper margins and orientation
  - Custom headers and footers
  - Line thickness
  - Scale of printed drawing

  For more information, refer to **Page Setup** on page 6-7.
- 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 6-4.)
  - Change the sheet size and orientation (See **Modifying the Sheet Setup** on page 6-6.)
- Customize line fonts. (See **Line Weight Preferences** on page 6-9.)

Creating a Drawing

Drawings consist of one or more views generated from a model or assembly. Parametric dimensions that originate in the model or assembly can be inserted in the appropriate drawing views. 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.

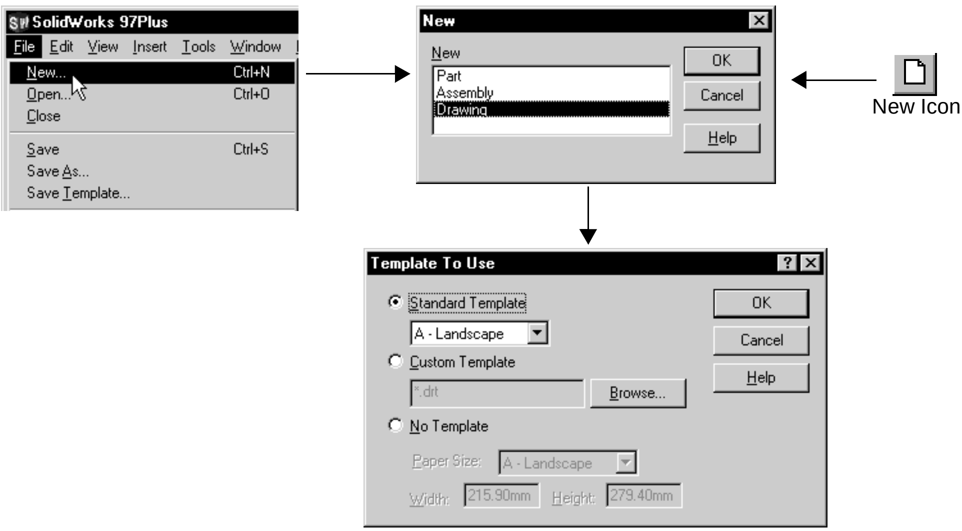
2. Select **Drawing** and click **OK**.

   The **Template To Use** dialog box appears.

3. Click one of the following radio buttons to select a template:

   - **Standard Template.** Select a standard sheet size template (for example, **C-Landscape**) from the pull-down list.
   - **Custom Template.** Enter the name of an already-defined template, or click the **Browse** button to find a template on your system or the network.
   - **No Template**. Select a blank, standard sheet from the **Paper Size** pull-down list, or choose the **User Defined** option from this list. If you choose **User Defined**, you can specify the paper size in the **Width** and **Height** boxes.

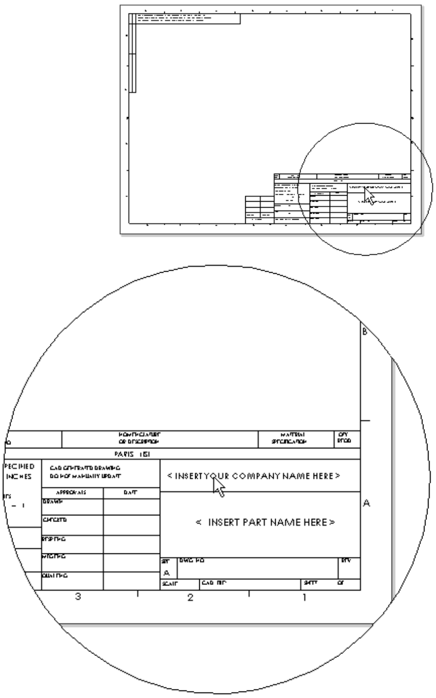
4. Click **OK**.



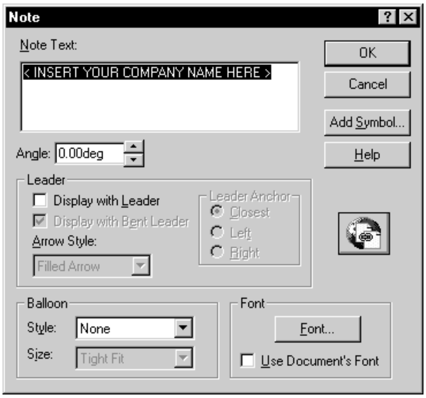
Customizing a Drawing Template

You can customize drawing templates to match your company's accepted format.

To edit the drawing template:



1. Right-mouse click anywhere on the drawing sheet and select **Edit Template** from the menu.
2. Double-click on the text that you want to replace.
3. Enter new text in the **Note** dialog box.
4. Click **Font** to select a new font style or size, if necessary.

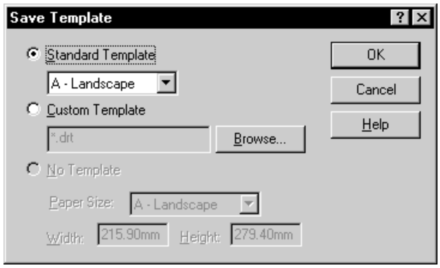


5. Click **OK** to accept the new font and new text.

   Additionally, you can 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 **Insert**, **Annotations**, **Note** or A. Complete steps 3, 4, and 5, and then drag the text to the correct location.

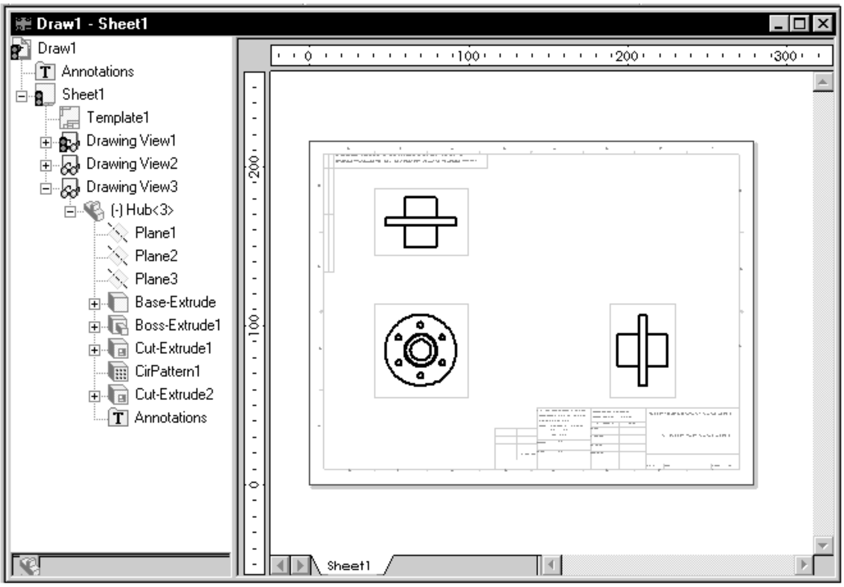
6 To save the template, click **File**, **Save Template** and enter a name. (Template files have the **.drt** extension.)



7 To end editing the template and return to working with an individual sheet, click **Edit, Sheet**, or click the right mouse button and select **Edit Sheet**.

The Drawing Window

The drawing window provides a FeatureManager design tree that is similar to the design tree in the part and assembly windows. The FeatureManager design tree for drawings also consists of a hierarchical list of items pertaining to the drawing. A ⊞ symbol beside an item's icon indicates that it contains associated items, such as sketches. Click the ⊞ to expand the item and display its contents.



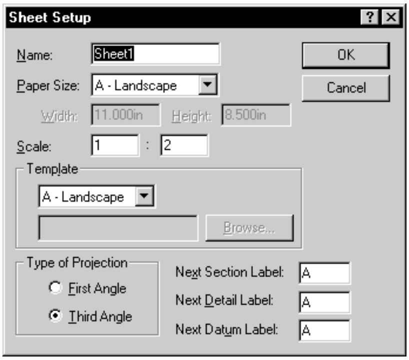
Modifying the Sheet Setup

You can set up the sheet details when you start a drawing, or later. You can also modify existing sheet details.

To specify sheet details:

1 Click the sheet icon in the FeatureManager design tree. If the drawing is composed of several sheets, select the sheet icon that corresponds to the sheet you want to modify.

2 Make the appropriate entries on the **Sheet Setup** dialog box:



- Enter a title in the **Name** box.
- Select a **Paper Size** from the drop-down list, or select **User Defined** to specify a custom paper size.
- If you selected **User Defined**, specify a **Height** and **Width** for the paper.
- Specify a drawing **Scale**; the default is 1:2.
- From the **Template** box, select a standard drawing template or select **Custom** or **None**.
- If you selected **Custom**, use the **Browse** button to locate and use a custom template.
- Select the **Type of Projection**, either **First Angle** or the default, **Third Angle**.
- Specify the letter of the alphabet that will be used for the next **Section Label**, **Detail Label**, and **Datum Label**. The default letter is **A**. Successive labels are in alphabetical order.

3 Click **OK**.

Multiple Sheets

You can add sheets to the drawing at any time.

To add sheets:

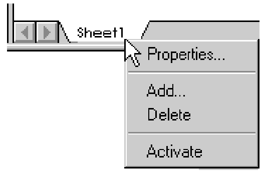
1 Click **Insert**, **Sheet**.

The **Sheet Setup** dialog box appears, with the default name of the next new sheet in sequence, for example, Sheet 3.

2 Set the parameters.

3 Click **OK**.

You can also right-mouse click on the sheet tab and select **Add** from the menu.



Right-mouse click

To delete a sheet:

1 Right-mouse click the sheet tab.

2 Select **Delete**.

3 Click **Yes** in the **Confirm Delete** dialog box.

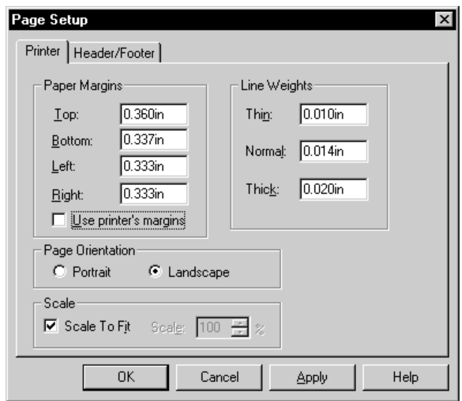
You can also click on the sheet and press the **Delete** key. Click **Yes** in the **Confirm Delete** dialog box.

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:



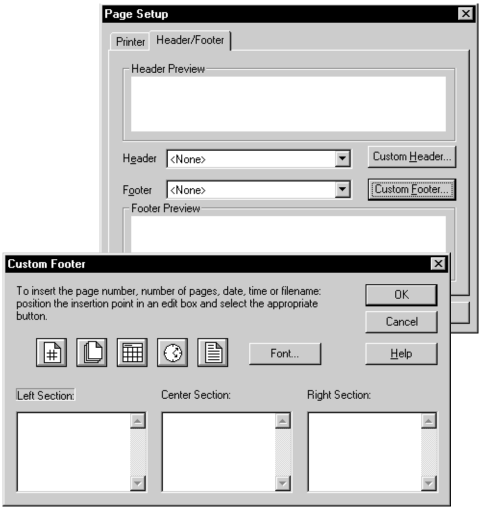
1 Click **File, Page Setup**.

2 From 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:

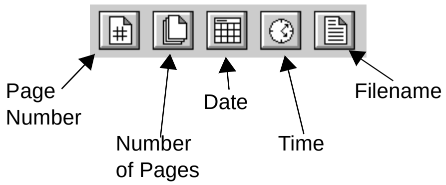


1 Click **File, Page Setup**.

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.

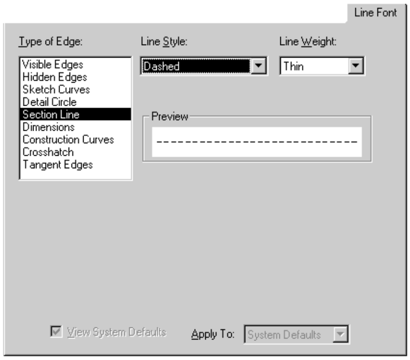
6 Click **OK** to close the dialog box.

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

Line Weight Preferences

You can specify the style and weight of edge lines for selected kinds of edges.

To specify the default line fonts:



1. Click **Tools, Options,** and click the **Line Font** tab.
2. From the **Type of Edge:** list, select an edge type from a list of line types: **Detail Circle, Sketch Curves**, **Dimensions**, **Section Line**, **Crosshatch**, and so on.
3. From the **Line Style:** scroll list, choose a line style to apply to the previously selected edge type. You can choose line styles such as **Solid, Dashed, Phantom**, and so on.
4. From the **Line Weight:** scroll list, choose a line weight to apply. Choose from **Normal, Thin,** or **Thick** lines.

   The **Preview** box displays the selected line.
5. Select the scope of the application in the **Apply To:** box.
   - **Active Document** – the document that you are currently working on
   - **System Defaults** – all new documents that are created
   - **All Possible** – the current document as well as all new documents
6. Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Reset All** to return to the installed system defaults.

Crosshatch Preferences

To specify a default crosshatch pattern:

1. Click **Tools, Options** and click the **Crosshatch** tab.
2. Select a crosshatch pattern from the **Pattern** list.
3. If necessary, set the **Scale** and **Angle** for the crosshatch pattern.

   A preview of the pattern is displayed in the **Type** box.
4. Select the scope of the application in the **Apply To:** box.
   - **Active Document** – the document that you are currently working on
   - **System Defaults** – all new documents that are created
   - **All Possible** – the current document as well as all new documents
5. Click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog box; click **Reset All** to return to the installed system defaults.

To make an immediate change to the crosshatch pattern for a surface, component, or view:

1. Right mouse click on a crosshatched face in a part or assembly drawing.
2. Select a new crosshatch pattern, if necessary.
3. Change the crosshatch angle and/or scale, if necessary.
4. View the changed crosshatch pattern in the **Preview** box.
5. Scroll the **Apply To:** box and select one of the available application modes:
   - **View.** All instances in the currently active section view.
   - **Component.** All faces on the currently selected component in an assembly.
   - **Region.** The currently selected surface only.
6. Click **Apply** to make the crosshatch change, or click **Cancel**.
7. Click **OK** to exit.

To remove the crosshatch for a specific layer (**Region**, **Component**, or **View**), select the layer and click the **Remove Crosshatch** check box. (The checkbox is only available for layers that can logically be removed.)

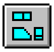
The crosshatch is removed a layer at a time. For example, if you remove the crosshatch from a region, any crosshatch applied earlier to the whole component is then revealed. If you remove crosshatch from a component, the default crosshatch for the view is revealed.

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. The part file or assembly file must be open and saved before you create a drawing of it.

You can also create Auxiliary, Detail or Section views from existing views.

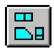
Note: An *Active* view allows you to sketch entities on that view. The border of an Active view is a red shadowed box. You can make a view active by clicking with the left mouse button twice on the view or using the right mouse button menu **Activate View**.
A *Selected* view allows you to show model dimensions in that view. A Selected view has a border of green. Select a view by clicking the view once with the left mouse button.

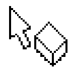
Standard 3 View 

The **Standard 3 View** option under **Insert**, **Drawing Views** creates three default orthographic views of a preexisting part or assembly displayed at the same time.

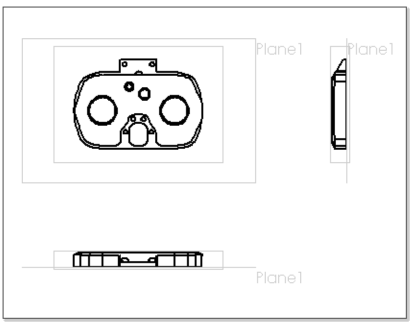
To create the Standard 3 View:

1 Open the saved part or assembly first, and then open a new drawing.

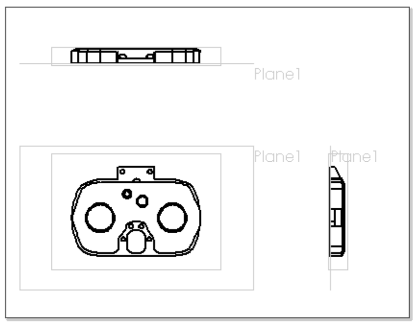
2 Click **Insert**, **Drawing View**, **Standard 3 View,** or  on the Drawing toolbar. The cursor changes to include a small box symbol.



3 Select the part or assembly by clicking in its window. The three views appear in the drawing automatically. The lower left view is the default **Front** view from the part or assembly.



1st Angle Projection



3rd Angle Projection

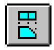
Note: In **Tools, Options, Drawings**, you can select the option **Automatic scaling of 3 view drawings** to specify that all three views will scale to fit on the drawing sheet regardless of the paper size selected.

Projection View 

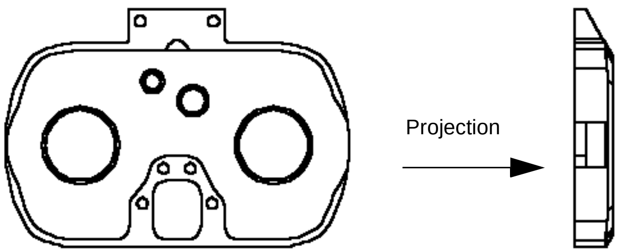
A **Projection** view is created by projecting another orthogonal view, using **First Angle** or **Third Angle** projection as specified in the **Sheet Setup** dialog. You can move a projected view only in the direction of the projection.

You can also break alignment by clicking a drawing view with the right mouse button and selecting **Break Alignment** from the menu. To align with another view, click **Align Horizontal** or **Align Vertical**.

To create a projection view:

1. Select an existing view by clicking once with the *left mouse button*. (The border of the view is highlighted in green, or the highlight color currently specified in **Tools, Options, Color** tab.)
2. Click **Insert**, **Drawing View**, then **Projection** or .
3. Select a projection direction with the cursor.

   The application automatically creates a projection view in the chosen direction (to the left, or right, above or below the existing view).



Orthographic View

Projection View

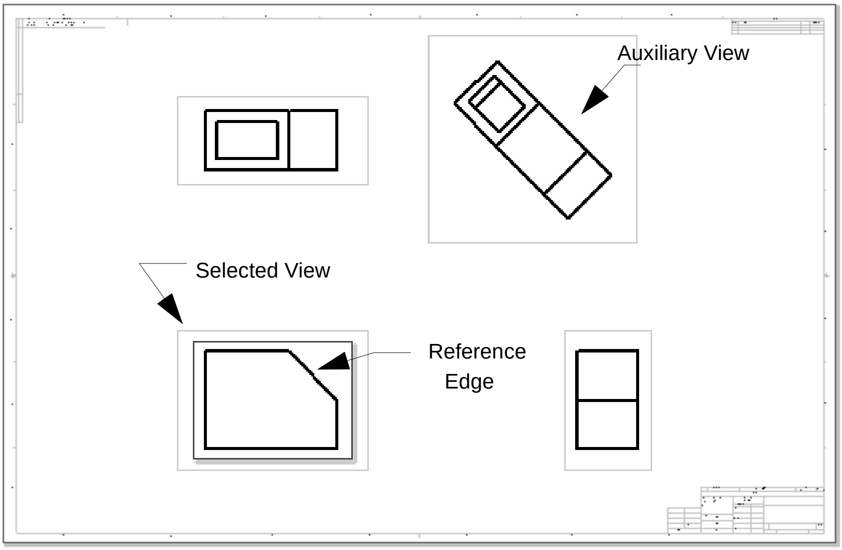
Auxiliary View 

An **Auxiliary** view is similar to a projection view, but unfolded normal to a reference edge in the reference view.

To create an auxiliary view:

1. Select an existing view with the left mouse button.
2. Select the reference edge (other than a horizontal or vertical edge which would give a standard projection view).
3. Click **Insert**, **Drawing View**, then **Auxiliary** or . The view appears automatically.

4 Move the view as desired along the auxiliary projection direction.



Auxiliary View

Named View 

You create a **Named** view by selecting from the view names in the model document. Named views include:

- Standard initial or stand-alone orthographic views (**Front**, **Top**, and so on)
- Default **Isometric** view
- Current model view
- Custom view created in the model by rotating a view and then saving it by name
- Exploded view of an assembly (For more information, see **Exploding an Assembly View** on page 5-19.)

For more information about creating your own named view in the **View Orientation** dialog box, refer to **Using Named Views** on page 1-14.

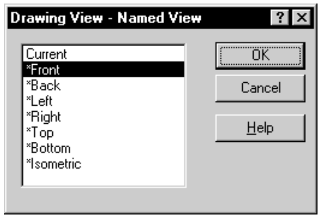
To create a named view:

1 For a custom view, first create and name a view in the part or assembly.

2 In the drawing, click **Insert**, **Drawing View**, **Named View** or .

3 Click in the part or assembly window with the Part Selection Cursor.

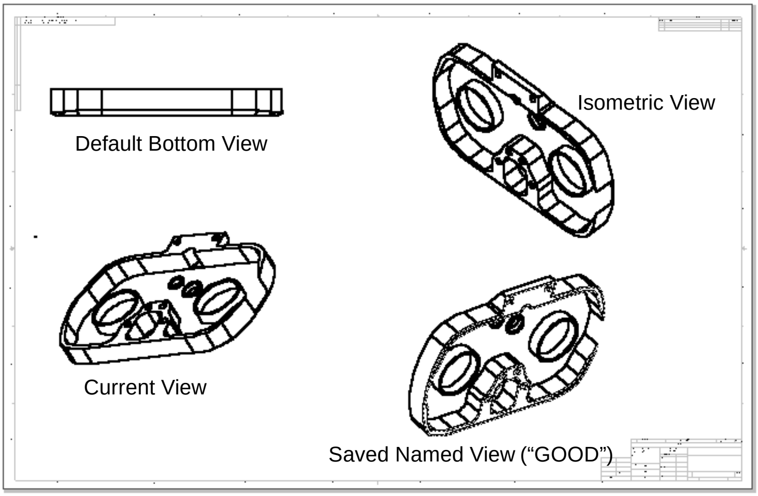
The **Drawing View - Named View** dialog box opens.



4 Double-click either your named custom view or one of the standard views on the list. (You can also select a name and click **OK**.)

5 If not visible, bring the drawing document forward by clicking in its window or by selecting it by name from the **Window** menu.

6 Click in the drawing where you want to place the part or assembly view.



Named Views

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 (Front, Top, and so on). For example, a true view of an angled surface perpendicular to the top surface of a model is created in this manner. The angled face is specified as **Front** and the top surface as **Top**.

To create a Relative to Model view:

1 Click **Insert**, **Drawing View**, **Relative To Model** or .

The part selection cursor appears.

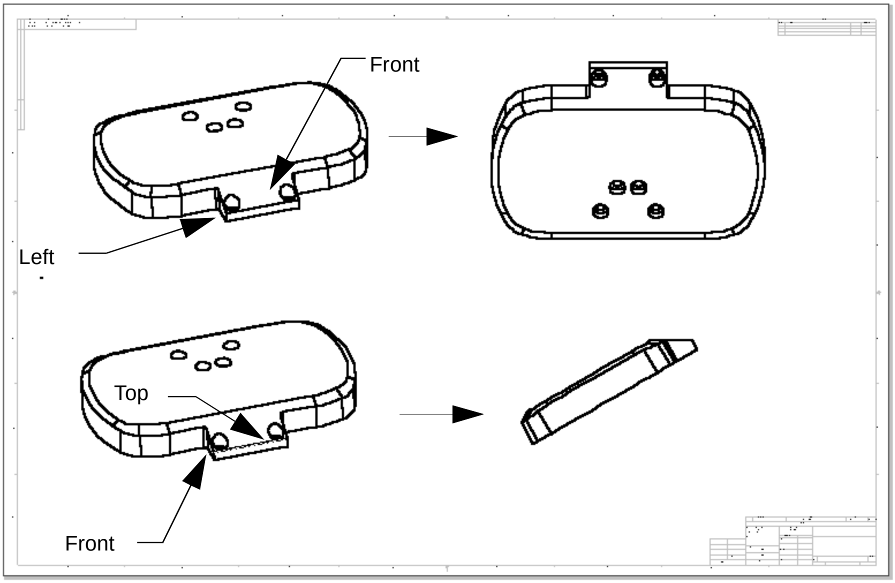
2 Select a face or plane in the model window that you want to have a particular orientation.

3 The **Drawing View Orientation** dialog box appears. Scroll to select an orientation (**Front**, **Top**, and so on), and click **OK**.

4 Repeat step 3 for another face or plane perpendicular to the first.

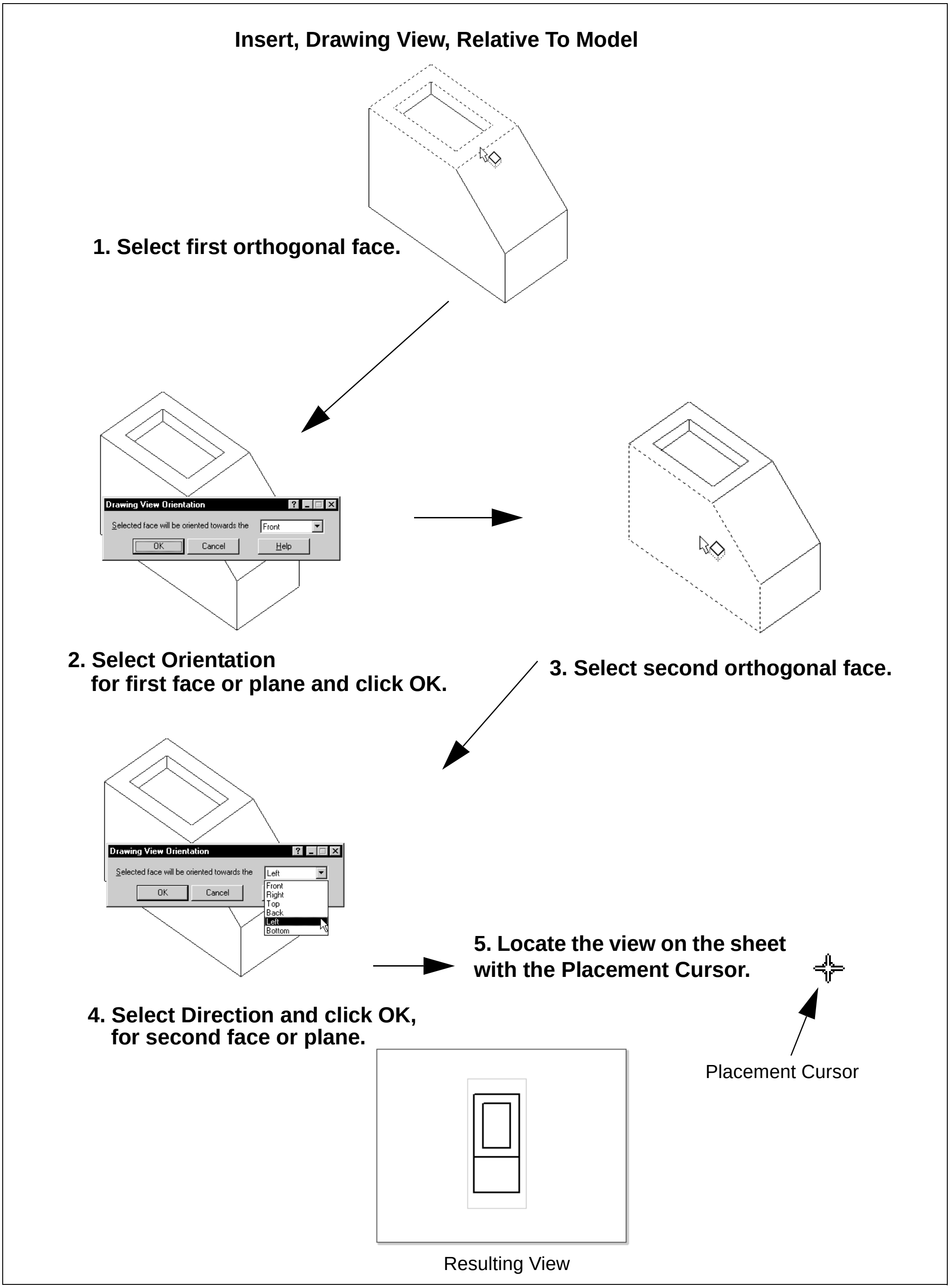
This view type is also used to set the first orthographic view in a drawing if it is to be something other than a default view.

Selections In Model | Resulting Views



Relative to Model Views

Note: If the inclined face in the above views changes its angle, then the views update to keep the orientation as originally defined.

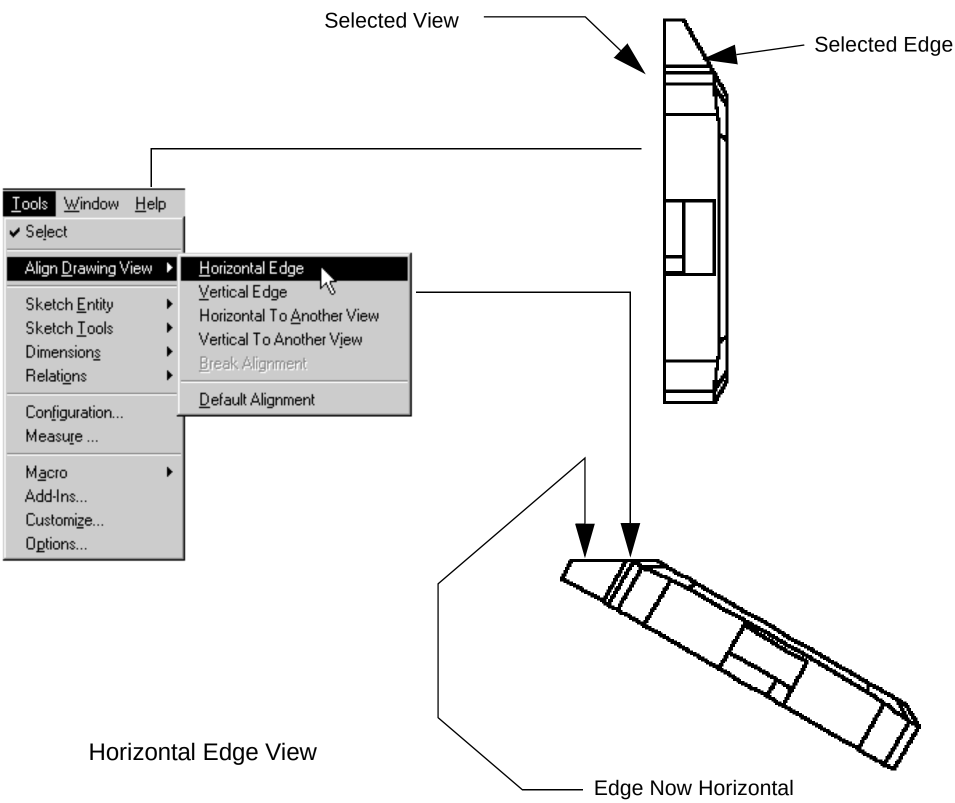

Insert, Drawing View, Relative To Model
1. Select first orthogonal face.
Drawing View Orientation
Selected face will be oriented towards the
Front
OK
Cancel
Help
2. Select Orientation
for first face or plane and click OK.
3. Select second orthogonal face.
Drawing View Orientation
Selected face will be oriented towards the
Left
Front
Right
Top
Back
Left
Bottom
OK
Cancel
4. Select Direction and click OK,
for second face or plane.
5. Locate the view on the sheet
with the Placement Cursor.
Placement Cursor
Resulting View

Align Drawing View

Align Drawing View is another option that you can use to adjust the view orientation. This option allows a view to be rotated so that one of its edges becomes horizontal or vertical.

To create an aligned drawing view:

1. Select a view in the drawing.
2. If you want to align the view to an edge, select a reference edge.
3. Click **Tools**, **Align Drawing View**, then select from the list of view choices. The view rotates automatically.



Note: If there are any views projected from the view you are changing with this option, they will update to maintain their projection.

Detail View

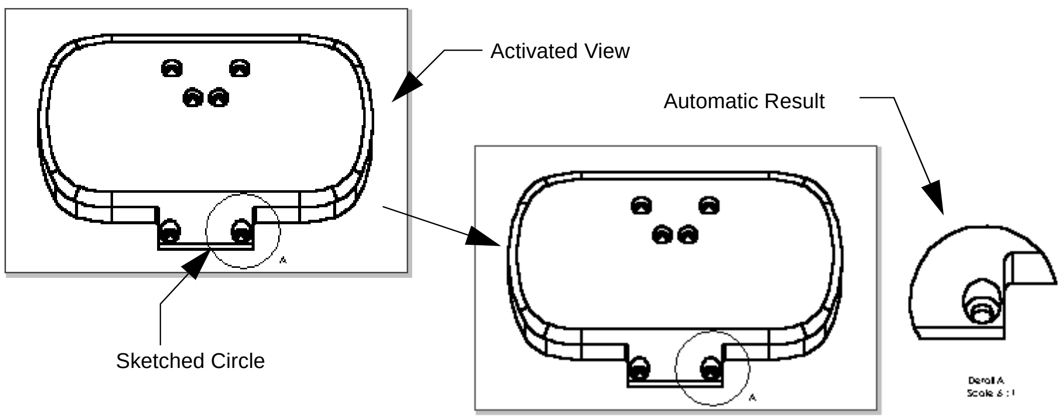
You create a **Detail** view to show a portion of a view, usually at an enlarged scale. This detail may be of an orthographic or 3D view or a section view.

To create a detail view:

1. Select an existing view with the *right mouse button* and select **Activate View**.
2. Select a sketch tool and create a closed profile around the area to be detailed. (The profile is usually, but not necessarily a circle.)
3. Click **Tools**, **Select**, or , and select one entity of the profile.
4. Click **Insert**, **Drawing View**, **Detail** or . A **Detail** view is placed on the drawing.
5. Click **Edit**, **Drawing View Scale** to set the scale of the **Detail** view.
6. You can then:
   - Move the view and/or detail note to the desired location on the drawing sheet by dragging with the mouse.
   - Double-click the detail note to edit it.
   - Double-click the detail circle letter label and change it.

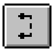
If you make changes to the detail view's label or scale, the detail circle reflects those changes, and vice versa.

You can reshape the detail circle by dragging it. The view automatically updates.



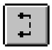
Detail View

To redefine the cutting edge of the detail circle, right-mouse click on the detail view and select **Edit Sketch**. Click and drag the profile to change the shape or size. Click **Rebuild** or to update the detail view.

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.

To create a section view:

1. Right-mouse click an existing view and select **Activate View.**
2. Click **Tools**, **Sketch Entity**, **Centerline**, or **Line** from the **Sketch Tools** toolbar.
3. Construct a single or stepped centerline through the selected view.
4. Click **Tools**, **Select**, or the **Select** icon.
5. Select the sketched centerline.
6. Click **Insert**, **Make Section Line** or .

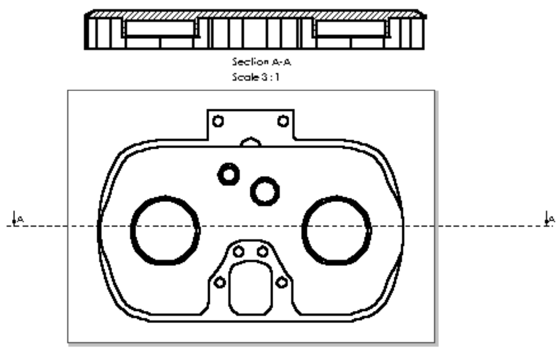
   Notice the arrows. If necessary, reverse the direction of the cutting line by double-clicking on the section line.
7. Click **Insert**, **Drawing View**, **Section**.

   A section view is created using the default crosshatch pattern, as defined in **Tools**, **Options**. (Changes to the crosshatch pattern selection take effect in the *next* SolidWorks session.) See **Crosshatch Preferences** on page 6-10.
8. Move the view to the desired location on the drawing sheet.

To change the direction of the view or make other changes, right-mouse click the section line and select **Properties** from the menu. Click **Edit**, **Rebuild** after making your changes. For more information, see **Section Line Properties** on page 6-22.

To edit the label text, double-click the section label.

To edit the section line, drag it and then click **Rebuild**. You can also right-mouse click the section line and select **Edit Sketch**, make changes, and click **Edit, Rebuild**.



Section View

Note: If you create a section view of an assembly, you can exclude some or all components from the section view. During the creation of an assembly section view, you are prompted to select components to exclude.

Aligned Section View

You can create an aligned section view through a model, or portion of a model, that is aligned with the selected section line.

To create an aligned section view:

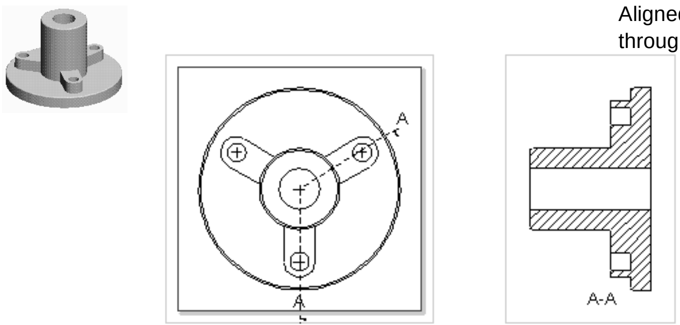
1. Double-click the drawing view to activate it.
2. Click or **Tools, Sketch Entity, Line** and draw the section line. The section line should be two connected lines at an angle to each other.
3. Click and select the line to which you want to align the view.
4. Click or **Insert, Drawing View, Aligned Section**. This converts the lines into a section line, with arrows indicating the direction of the view.
5. If necessary, double-click on the section line to reverse the direction of the view.

You can edit section lines by dragging them and then clicking or **Edit**, **Rebuild**. You can also right-mouse click the section line and select **Edit Sketch**, then make changes and click **Edit, Rebuild**.

Note: You cannot change the angle of the line that you selected to align the view; you can change its position, however.

To change the direction of the view or make other changes, right-mouse click the section line and select **Properties** from the menu. Click **Rebuild** after you make your changes. For more information, see **Section Line Properties** on page 6-22.

Aligned section, cut through model

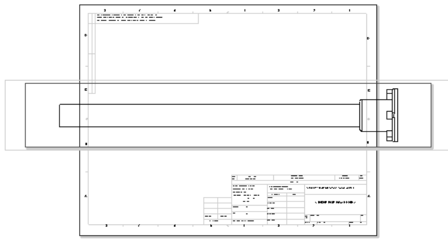


Aligned Section View

Broken View

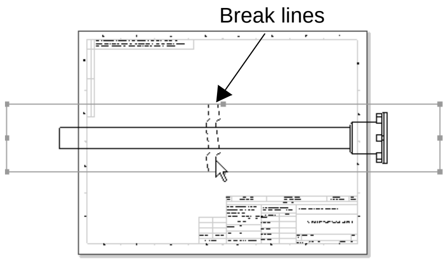
You can use a broken view (or an interrupted view) on the drawing of a long part that has a uniform cross-section. This makes it possible to display the part in a larger scale on a smaller size drawing sheet.

To create a broken view:



1 Select the drawing view and click **Insert, Vertical Break** (or **Horizontal Break**).

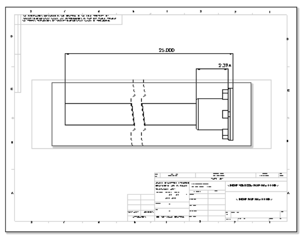
Two break lines appear in the view.



2 Drag the break lines to the places on the part where you want the breaks to occur.

3 Right-mouse click inside the view border (green border) and select **Break View** from the menu.

The part is displayed with a gap in the part geometry; however, the dimensional values associated with the broken area reflect the correct values, assuming the view displayed dimensions to start with.



4 You can continue to drag the break lines to adjust them after the view is broken, if necessary.

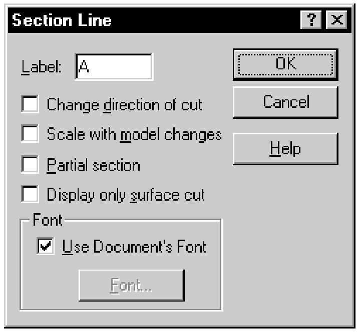
- ❑ To change the shape of the break lines, right-mouse click a break line and select a line style from the menu: **Straight Cut**, **Curve Cut** or **Zig Zag Cut**.

- ❑ To specify the width of the break gap, click **Tools**, **Options,** and select the **Detailing** tab. Enter a new value in the **Break Gap** box under the **Break Lines** option.

Note: True model dimensions can be inserted into a broken view.

Section Line Properties

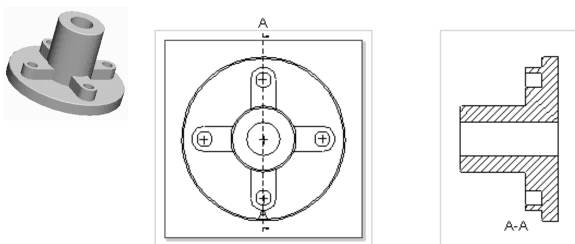
You can make changes to the **Section View** and **Aligned Section View** by clicking the section line with the right-mouse button. Select **Properties** from the right-mouse menu.



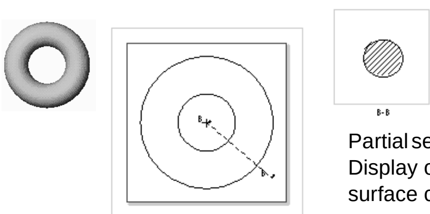
- **Label.** Change the letter that is associated with this section line by selecting the letter and entering a new label.
- **Font.** When the box is checked, the section line text uses the same font as the rest of the document. When *not* checked, you can click the **Font** button and specify the font style and size to use.

The following kinds of section cuts and displays may be used individually or in combination. The illustrations that follow help to describe the section cuts that are available. You may experiment until you get the results that you want. Remember to click **Rebuild** after you make changes on the **Section Line Properties** sheet.

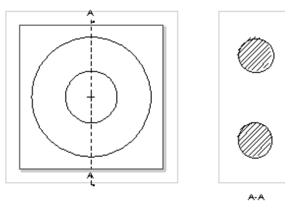
- **Change direction of cut.** Reverse the direction of the section line cut.
- **Scale with model changes.** The section line changes size parametrically with the model. This could result in a section cut that is not what you intended, however. The default for this option is off.
- **Partial section.** A simplified view with extraneous surfaces removed.
- **Display only surface cut.** Only the surface(s) cut by the section line are displayed in the section view.



Reverse direction,
Cut through model



Partial section
Display only
surface cut



Display only
surface cut

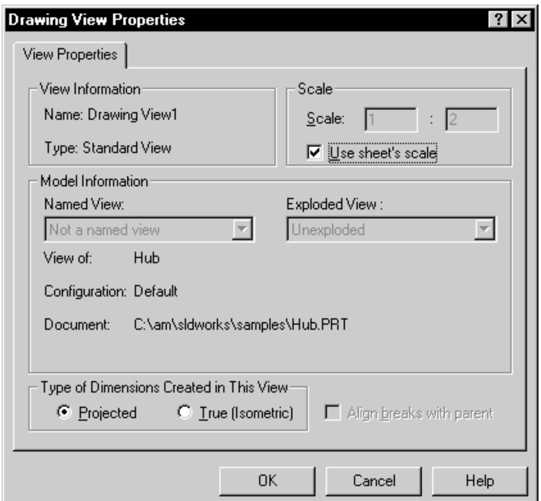
View Modification and Display

You can modify and display views in several ways.

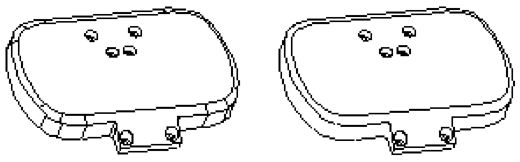
- ❑ To move a view, move the cursor over the edge of the view until the move cursor appears. Click and drag the view to a new location while holding down the mouse button. Note the following exceptions:
  - **First angle projection**. The alignment of the upper left view is fixed in relation to the other two views. When you move it, the other views also move. The other two views can move independently, but only horizontally or vertically, to or from the upper left view.
  - **Third angle projection**. The alignment of the lower left 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, to or from the lower left view.
  - **Auxiliary** and **Projection** views. These views are aligned with the views they are related to and only move in the direction of the projection.

  In each of the above cases, you can break the alignment and move the views independently. Right-mouse click the view and select **Break Alignment** from the menu.

- ❑ To change the scale of a view, right-mouse click the view, and select **Properties.** In the dialog box, choose whether or not to use the sheet scale with the **Use sheet's scale** check box. Set the view scale by entering values in the two scale ratio boxes, then click **OK**.



- ❑ To control the display of tangent edge lines in the drawing view, right-mouse click the view and select **Tangent Edges Removed** from the menu.



Tangent Edges Visible Tangent Edges Removed

View Visibility

You may 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.

To hide/show views:

1. Right-mouse click the view, or the view's name in the FeatureManager design tree, and select **Hide View**.
2. To display a view, right-mouse click over 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. In this case, a suppressed view displays a gray **X** through the view location.

You can selectively hide and show edges in a drawing view.

To hide or show edges:

1. Right-mouse click an edge in a drawing view.
2. Select **Edge Properties** and click **Show Entity** or **Hide Entity**.
3. Click **OK**.

Annotating Drawings

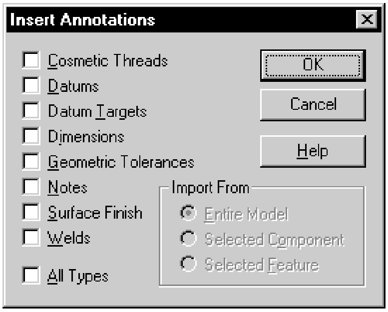
When preparing a drawing, you can choose to

- *Import* annotations and dimensions that are attached to the assembly and part documents
- *Add* annotations and reference dimensions to the drawing

Both methods are described in this section. For more information about using dimensions in drawings, see **Dimensions in a Drawing** on page 6-27. For more information about using annotations in drawings, see **Annotations** on page 6-34.

Importing Model Annotations

You can import model annotations and dimensions into a selected view in a drawing. You can select what types of annotations to import by clicking **Insert, Model Annotations**, and selecting annotations from the list.Select **All Types** if you want to import all model annotations available for the selected view.



If you are working with drawing of an assembly, you can choose where to import the annotations from: the **Entire Model**, the **Selected Component**, or the **Selected Feature**.

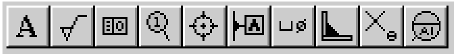
The imported items have the following behavior:

- They display in the **Imported Annotations** color defined in **Tools, Options, Color** tab.
- The attachment points for annotations can be dragged (edge/face attachment), but cannot be re-attached to another edge/face/vertex/free space.
- The attachment of the annotation in the drawing does not affect the attachment in the part or assembly.

Adding Annotations to a Drawing

You can add annotations and reference dimensions to a selected view in a drawing. Click **Insert, Annotations** and select from the available annotation types in the pull down menu.

You can also click one of the **Annotation** icons in the Drawing toolbar. (Only those annotation types that are appropriate for the selected view are available for selection; the others are displayed in gray.)



Reference annotations (those added in the drawing) display in the **Reference Annotations** color defined in **Tools, Options, Color** tab. Reference dimensions display in the **Reference Annotations** color also.

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 during the creation of each part feature and those dimensions are inserted into the various views when you click **Insert**, **Model Annotations** with **Dimensions** selected in the **Insert Annotations** list. (Only those dimensions appropriate to the selected view appear.)

You may create dimensions in drawing mode, but these become reference dimensions and cannot be used to change the dimensions of a part. However, the values of reference dimensions change when the model dimensions change.

One-way associativity can be set between drawings and models during the software installation. With this option set, changes to the drawing are not reflected in the model.

Note: You can find additional information about dimensioning in Chapter 2, **Dimensioning Sketches** on page 2-18.

Showing Model Dimensions

To show model dimensions in a drawing:

1. Select a view you wish to dimension. (The border of the view turns green when it is selected.)
2. Click **Insert, Model Annotations.** The **Insert Model Annotations** dialog box appears. (Only those options that apply to the selected view are available.)
3. Select **Dimensions**.
4. Click **OK**.

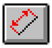
Reference Dimensions

By default, reference dimensions appear with parentheses. If you prefer to display them without parentheses, click **Tools**, **Options** and select the **Detailing** tab. Click the **Add Parentheses by Default** option to remove the check mark.

Note: You cannot change the model by modifying reference dimensions. However, when you change the model, the reference dimensions change accordingly. Reference dimensions are not driving dimensions.

You can add reference dimensions to a drawing by using the **Tools**, **Dimension, Baseline** menu option, the **Dimension** tool, or the right-mouse menu.

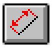
To add reference dimensions to a drawing:

1 Click **Tools**, **Dimension, Baseline** from the menu bar or select  from the **Sketch Relations** toolbar.

2 Select the geometry to dimension with the *left mouse button*.

3 Place the dimension with a left button click.

Alternatively, you can:

1 Click  from the **Sketch Relations** toolbar.

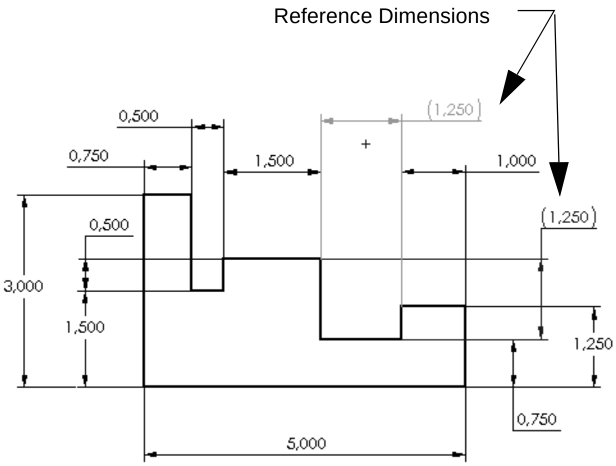
2 Press the right mouse button and select one of the following from the menu:

- **Horizontal Dimension**
- **Vertical Dimension**
- **Baseline Dimension**

3 Select the geometry to dimension with the *left mouse button.*

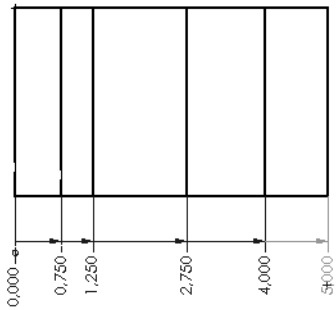
4 Place the dimension with a left button click.

Each dimension type has a unique cursor symbol.



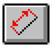
Ordinate Dimensions

To place ordinate dimensions:



1. Click **Tools**, **Dimension**, **Ordinate.**

   – or –

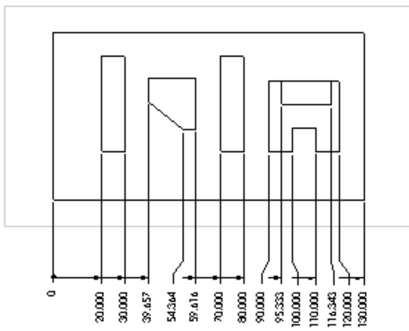
   Select  icon from the **Sketch Relations** toolbar, press the *right mouse button* and select **Ordinate Dimension, Horizontal Ordinate**, or **Vertical Ordinate.**

2. Pick the entity at zero position and place the dimension.

3. Pick successive entities and place their respective dimensions.

Note: To line up ordinate dimensions, click the right mouse button and select **Align Ordinate** from the menu. You can also add additional ordinate dimensions at any time by right-mouse clicking the original baseline dimension and selecting **Add to Ordinate** from the menu. Then select a new entity to dimension.

To jog ordinate dimensions for better visibility:



1. Right-click an ordinate dimension to move and select **Jog** from the pull-down menu. A check mark appears next to the menu item.

2. Pull the dimension in any direction.

   The leader line bends so that you can place the dimension freely.

3. Use the **Align Ordinate** command from the right-mouse menu to align all the dimensions in the ordinate.

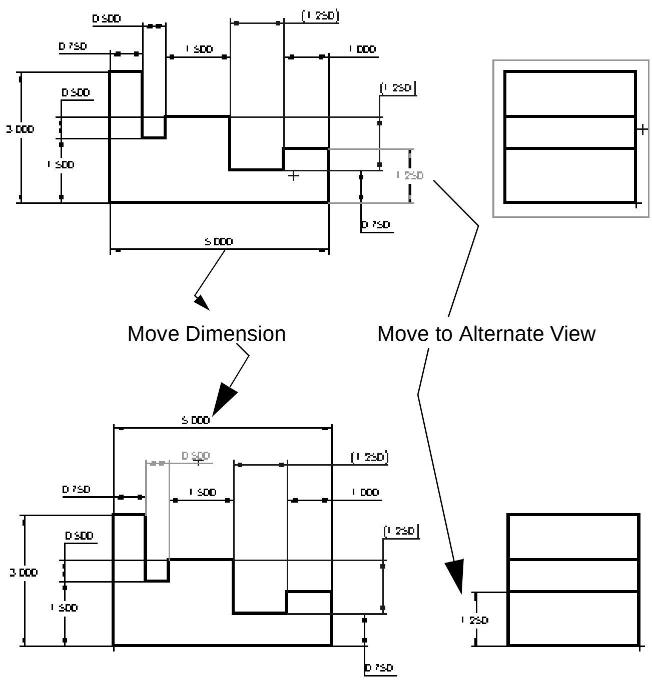
Note: Selecting **Jog** again returns the dimension to its original place and unchecks the menu item.

Moving Dimensions

Once dimensions are displayed, you can move them within a view or to another view.

To change the location of a dimension, drag it from one place to another. The dimension reattaches to the model, as appropriate.

- To move a dimension within the view, click and hold the *left mouse button* over the text of a dimension. Move the cursor to the new position and release the mouse button.
- To move a dimension from the active view to another view, hold the **Ctrl** key while you click and move the cursor to the alternate view.



Note: You can only move dimensions to a view where the orientation is appropriate for that dimension.

Modifying Dimensions

To modify a dimension in a drawing:

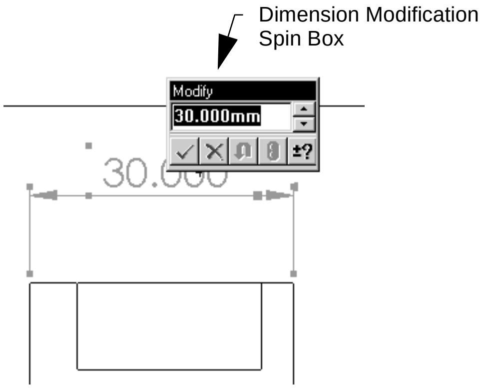
1 Double click on the text of the dimension. The dimension value is displayed in a spin box.

2 Scroll to a new value with the up and down arrows, or enter a new value and press **Enter**.

3 Click  to accept the current value and exit.

– or –

click  to restore the original value and exit.



Note: To set the **Spin Box Increments,** click  under the spin box and enter a new value in the dialog box.

Editing Dimension Properties

You can modify the properties of dimensions at any time during the detailing process.

To modify the properties of a dimension:

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

2 Change values in the first dialog box or select **Modify Text**, **Font**, or **Tolerance** to open another dialog box and make changes there.

3 Click **OK**.

Options for dimension modification include the following:

- **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 short* **Name** *given above and the name of the sketch or feature in which it is found. For example, D2@Sketch1.*
- **Read Only —** Determines whether feature dimensions can be changed or not.
- **Arrow Style —** The arrowhead style: **Open**, **Filled**, and so on.

- **Precision —** Specifies the precision of the primary and alternate units of measure.
- **Tolerances — Allows the modification of dimension tolerances.**
- **Modify Text —** Allows the addition of prefix and suffix text and symbols.
- **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.

For additional online information about choices in the **Dimension Properties** dialog box, click the **Help** button when the dialog box is open.

Dimension Visibility

The visibility of a dimension is changed by clicking **View, Hide/Show Dimensions.** Model or reference dimensions are hidden or displayed to control the appearance of the drawing, temporarily or permanently.

1. Select a view. (The view boundary turns green.)
2. Click **View, Hide/Show Dimensions.**

   The cursor appears as shown. 
3. Select the dimensions that you wish to hide or show.
4. Click the **Selection** tool when done.

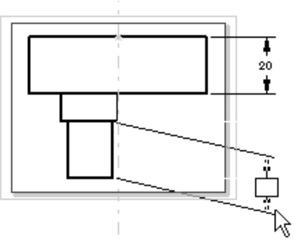
Note: Any previously hidden dimensions re-appear in a dimmed format when you select the view and choose **Hide/Show** menu.

Dimension Witness Line Control

To selectively display witness lines and dimension lines:

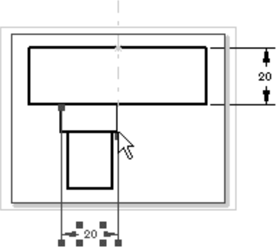
1. Right-mouse click a dimension and select **Properties**.
2. On the **Dimension Properties** dialog, click the **Display** button.
3. Use the checkboxes to specify whether or not to display **Witness Lines** and **Leaders**.
4. Click **OK** or **Cancel** when you are done.

To slant witness lines:



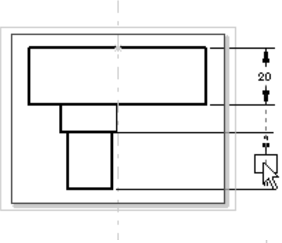
1. Click a dimension. Handles are displayed on the dimension and witness lines.
2. Drag the witness line handles to slant the witness lines as needed.

To change the model attachment point of dimension witness lines:

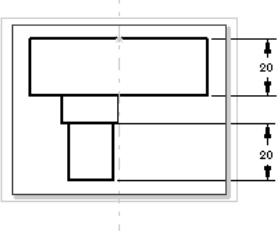


1. Click a dimension.
2. Select the end of the dimension line that you want to move.
3. Drag the mouse along the dimension line until it reaches the desired vertex and then release the mouse button.

To align dimensions with neighboring dimensions:



1. Click a dimension.
2. Drag the dimension until inferencing lines appear and the dimension snaps to align with its neighbor.



Annotations

You can place notes, a bill of materials, or a variety of symbols on your drawing to further clarify the information the drawing provides. These annotations may be imported from the part or assembly document or added to the drawing. (See **Annotating Drawings** on page 6-25.

Notes

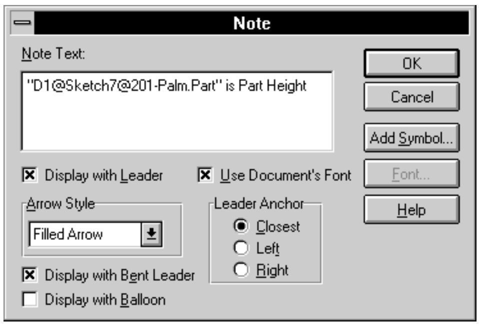
You may create a variety of notes. A note may be free floating or placed with a leader pointing to something specific in the drawing. It may contain simple text as well as parametric text. *Parametric text* changes when the driving parameters are changed. A note can have a maximum of 1023 characters.

To create a note:

1. Select a note location on the drawing or the location of the arrow point if the note is to include an attached leader.
2. Click **Insert**, **Annotations**, **Note** or on the toolbar.
3. Type the desired text into the **Note Text** dialog box.
4. Select the appropriate options.
5. Click **OK**.
6. If you select **Display with Leader**, select the note location on the drawing.

Note: Parametric values can be added automatically by selecting dimensions in a drawing view.

Parametric Note:
The dimension value is entered by selecting a dimension in the view.



Resulting Note: "2.000 is Part Height"

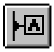
You can cut and paste a note along with its leader by selecting the note and using **Ctrl-X** and **Ctrl-V**. You can copy and paste a note using **Ctrl-C** and **Ctrl-V**.

Notes with multiple leaders

You can select any number of attach locations and then create the note. You can add, move, and remove leaders of existing notes.

Hyperlinks in notes

You can add an embedded hyperlink in a note. The hyperlink may be to a document on the Internet, your local network, or on your own hard drive. The hyperlink will not appear in the text, but the cursor dynamically changes to a hand when the cursor moves over the note or balloon that is hyperlinked. Selecting the hyperlink brings the user to the associated URL (document or Web site).

Datum Feature Symbol 

You can attach a datum feature symbol to the following places:

- A model surface or a section view surface that appears as an edge
- A geometric control frame

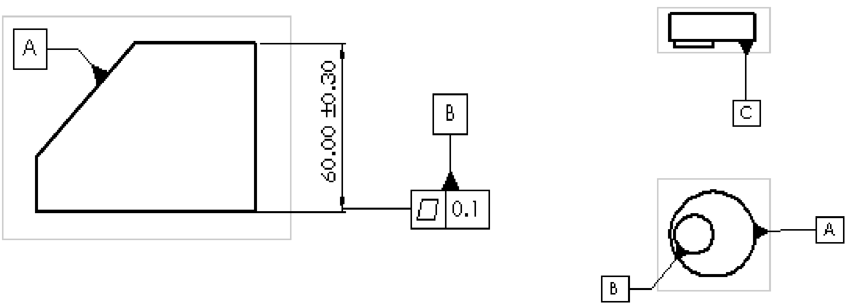
To insert a datum feature symbol:

1. With a drawing active, click  or **Insert, Annotations, Datum Feature Symbol.**
2. Click the surface on which you want to place a datum feature symbol.

   You can place multiple datum feature symbols; the datum letters are assigned alphabetically (the letters I, O, and Q are omitted in ANSI only).
3. To stop placing datum feature symbols, click  or **Insert, Annotations, Datum Feature Symbol** again.

You can place the datum feature symbol on an edge and drag it off the model edge to create an extension line.

To change the datum feature symbol, right-click the symbol and select **Properties** from the menu.



Datum Target Symbol

You can attach datum target symbols to a datum target point, or edge in a drawing view.

To create a datum target symbol:

1. Click a model edge or a datum target point in the drawing view.
2. Click or **Insert, Annotations, Datum Target Symbol**.

   The **Datum Target Symbol** dialog appears.
3. Enter a value for the **Target area size**; click if the value describes a diameter.
4. Click the **Display target area size outside** check box if you want to place the value outside the target symbol circle.
5. Enter up to three **Datum Reference** labels in the boxes.
6. Specify the leader style that you want:
   - Check **Bent Leader**, if appropriate.
   - Select an **Arrowhead Style** from the pull-down menu.
   - Select a **Line Style** from the scroll-down list. Typically, a **Solid** line means the target is on the near side; a **Dashed** line means the target is on the far side.
7. Click **OK** to accept the datum target symbol you constructed; or click **Cancel**.

To change the location of the datum target symbol, click and drag the circle.

To edit the datum target symbol, right-mouse click on the circle and select **Properties** from the menu. The **Datum Target Symbol** dialog appears. Make changes to the options as described above.

Datum Target Point

You can place target points on the model surface in a drawing.

To place target points on a drawing view:

1. Activate the drawing view.
2. Click or **Insert, Annotations, Datum Target Point**.
3. Point and click the mouse where you want to place a target point.

   Notice that an inferencing line is available to assist you in placing the target points in relation to each other.

   You can also apply constraints (**Add Relations**) on the datum target point, as you can with a sketch point.
4. Click again to end making target points.

Center Mark

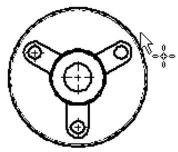
You can place axis lines for showing center marks on circles that can be used as reference points for dimensioning.

To create a center mark:

1 Click or **Insert, Annotations, Center Mark**.

2 Click the circle or arc.

3 To stop placing centerpoints, click or **Insert, Annotations, Center Mark** again.



Note: This function only places center marks on model circles.

To set default preferences for center marks:

Click **Tools, Options** and select the **Detailing** tab.

You can set the default size of center marks and choose whether or not to display extended axis lines.

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.

Cosmetic Thread

You can represent threads on a part, assembly, or drawing. You may display threads that have already been inserted in the part, or you may insert threads in the drawing of a model that does not already have threads.

To display cosmetic threads that exist in the model:

1 Select a drawing view.

2 Click **Insert, Model Annotations**.

3 Select the **Cosmetic Threads** option from the **Insert Annotations** dialog.

4 Click **OK** to display the thread; click **Cancel** to exit.

To insert cosmetic threads on a drawing view:

1 On a cylindrical part, click a circular edge where the thread should begin.

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.

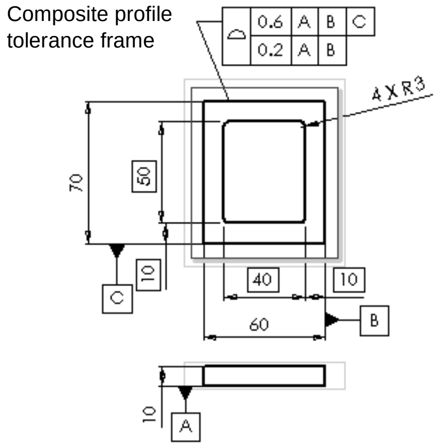
5 In the **Thread Callout** box, enter thread callout text, if desired.

6 Click **OK** to create the thread; click **Cancel** to exit.

Geometric Tolerancing

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:

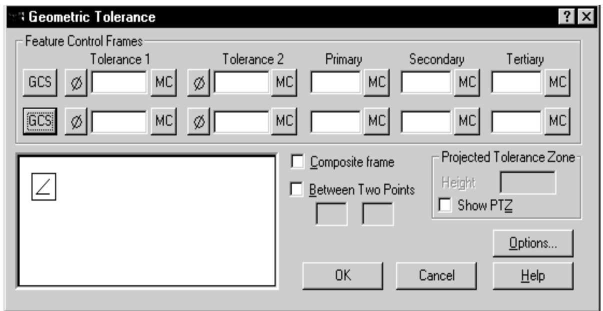
1. Click **Insert**, **Annotations**, **Geometric Tolerance** or the toolbar button on the drawing toolbar.

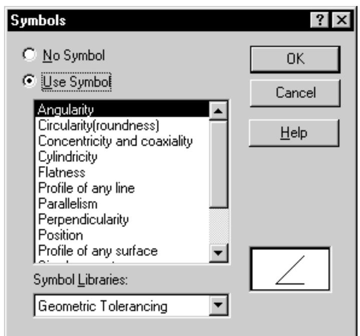
   The **Geometric Tolerance** dialog box appears.

2. In the first **Feature Control Frame** area, click **GCS**, then select a **Geometric Characteristic Symbol** from the **Symbols** dialog box, and click **OK**.

   Notice that as you make each selection, the information appears in the preview display box.

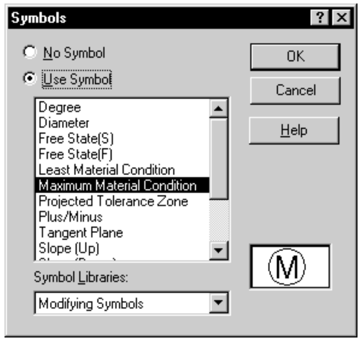
3. Enter a tolerance value in the **Tolerance 1** box. Click Ø if you want to include a **Diameter** symbol.





Geometric Characteristic Symbols

4 Click **MC**, to choose the material condition symbol for **Tolerance 1** and click **OK**.



Material Condition Symbols

5 Repeat steps 3 and 4 for **Tolerance 2.**

6 Enter tolerance values and material condition symbols for the **Primary, Secondary,** and **Tertiary** datums.

7 If you are creating a composite frame, repeat the process for the second **Feature Control Frame**,

– or –

Click the **Composite Frame** check box and complete entering values and material condition symbols.

8 Click the **Show PTZ** check box if you want to enter a **Projected Tolerance Zone** height. Enter a value in the **Height** box.

9 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.

10 When you are finished, click **OK** and drag the geometric tolerance frame to the desired position on your drawing.

To edit an existing tolerance control frame:

1 Double-click on the control frame. The **Geometric Tolerance** dialog box appears with the previously defined tolerance in the preview display box. Add or change values or callouts in the window.

2 Click **OK** to update the geometric control frame.

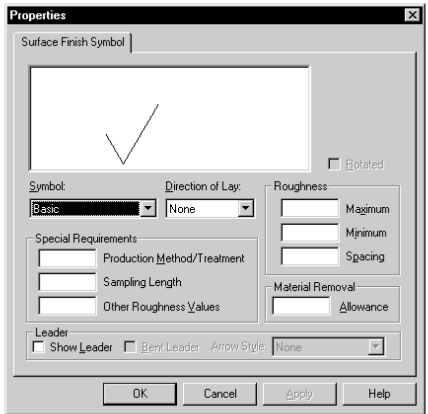
Surface Finish Symbols

You can define the surface texture of a part by placing a **Surface Finish Symbol** on any face. You select a symbol type (**Basic**, **Machining Required**, or **Machining Prohibited**) and then add relevant information to construct a symbol that is appropriate for the machining operation that you want.

Surface finish symbols can be attached to a model surface that appears as an edge.

To add a surface finish symbol to a part surface:

1. In a drawing, click an edge where you want to place the symbol.
2. Click or **Insert, Annotations, Surface Finish Symbol.**
3. Construct the symbol that is appropriate for the selected surface by selecting values from the options in the **Surface Finish Symbol** dialog box:



- **Symbol.** Select a symbol type from **Basic**, **Machining Required**, or **Machining Prohibited**.
- **Direction of Lay.** Select the direction of the surface pattern from a pull-down list.
- **Roughness.** Define 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.** Define the amount of stock to be removed by machining.
- **Leader.** Check the **Show Leader** box to display the symbol with a leader.
- **Bent Leader.** Check this box to allow the leader to have a bend.
- **Arrow Style.** If a leader is displayed, you can choose the type of leader end from a pull-down list.

4. Click **OK** when you are satisfied with the surface finish symbol you defined.

To edit a surface finish symbol, double-click the symbol and make changes to the **Surface Finish Symbol** dialog box that appears.

The surface finish symbol can be dragged to a new location. If it is moved off the model edge, an extension line is created.

Bill of Materials

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 spreadsheet program installed in order to insert a bill of materials into a drawing.

To insert a bill of materials into a drawing:

1. With a drawing view selected, click **Insert, Bill of Materials**.
2. Choose the kind of information that you want to include in the bill of materials. Select either: **Show parts only** or **Show top level subassemblies and parts only**.

   A bill of materials is displayed that lists the parts in your assembly.
3. Insert any additional information or columns in the spreadsheet using the appropriate Excel commands and icons, as necessary.
4. Click a corner of the spreadsheet to change its size.
5. Click anywhere outside the spreadsheet to close it.
6. Double-click on the spreadsheet to open it again for editing.

To move the bill of materials:

1. Click on the spreadsheet. Your cursor changes to the *Move* shape, .
2. Hold the left mouse button and move the cursor to where you want to place the bill of materials.
3. Release the mouse button.

To update the bill of materials:

1. Click in the drawing view originally selected to create the bill of materials.
2. Click **Insert**, **Bill of Materials**.

   The existing bill of materials updates to include any new components that you added to the assembly; components that you deleted are removed from the bill of materials.

Creating Balloon Callouts

You can create a balloon callouts in a drawing. If you previously inserted a bill of materials in the drawing, the balloons label the parts in the assembly and relate them to item numbers on the bill of materials. The balloon leader attaches to the part that you click in the assembly drawing.

There are six balloon styles: circle, box, diamond, hexagon, triangle, and circle with a split line. You can set the default balloon type by clicking **Tools, Options**, and the **Detailing** tab. You can globally set the balloon style to be used with bills of materials and notes.

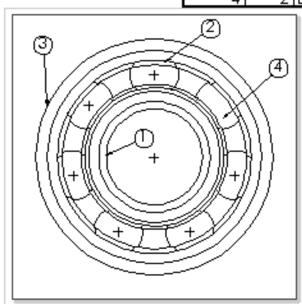
Balloons are automatically suppressed when the components they reference are suppressed.

To create a balloon:

1. Click or **Insert, Annotations, Balloon.**
2. Click on a part in a drawing view.

   A balloon that contains a number corresponding to the part number on the bill of materials attaches to the part that you clicked.
3. To move the balloon or leader arrows, drag them with the **Select** tool, .

ITEM N	QTY.	PART NO.
1	1	bearing1
2	7	ballbearing
3	1	bearing2
4	2	bearingcase



To change the balloon text, font style, or leader configuration:

1. Double-click on the balloon.
2. Make the changes that you want in the **Note** dialog and click **OK**.

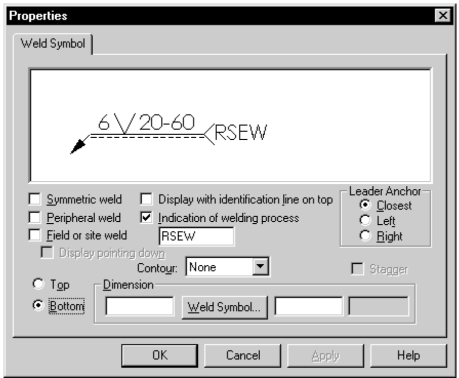
Weld Symbol

You can construct weld symbols to provide specifications of welds on your drawings.

To place a weld symbol on your drawing:

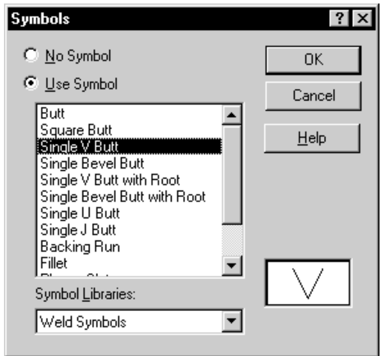
1 Click an edge in the drawing view where you want to indicate a welded joint.

2 Click **Insert**, **Annotations**, **Weld Symbol**.



3 In the **Dimension** box, enter a value for the weld dimension.

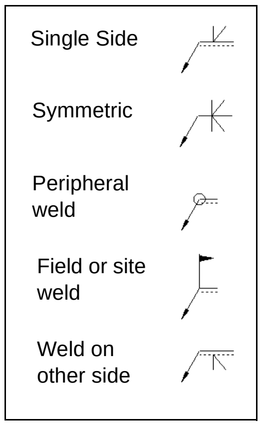
4 Click the **Weld Symbol** button and select a symbol type from the pull-down list of standard symbols from the **Symbols** dialog box.



5 Enter additional dimensions or values that may be appropriate for this specific symbol in the box to the right of the **Weld Symbol** button.

(In the example shown, the length and the pitch of the weld is entered in this box.

6 Continue constructing the weld symbol by choosing options from the dialog box.



- **Symmetric weld.** The same type of weld on both sides of the part.
- **Peripheral weld.** Weld surrounds the part.
- **Field or site weld.** Weld to be made out of the shop or in the field.
- **Display with identification line on top.**
- **Weld on the other side.** Weld on the opposite side of the part.
- **Stagger.** Intermittent fillet weld.

- **Contour Symbols.** (None, Flat, Convex, or Concave)
- **Indication of welding process.** Enter standard welding process symbols such as CAW (Carbon Arc Welding), FW (Flash Welding), and so on.

2D Sketching

You can create drawing geometry using 2D sketched geometry only, without reference to existing models or assemblies. However, 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.

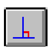
To start sketching, click one of the icons on the **Sketch Tools** toolbar or click **Tools**, **Sketch Entity**, and select a tool. Then you can:

- 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 . 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 2-15.

To define relations between one or more sketched entities:

1. **Ctrl-**select the entities and click  on the **Sketch Relations** toolbar or click **Tools, Relations, Add**.
2. Select from the relations in the dialog box. Choices are available based on the entities you picked; inappropriate are grayed out.
3. Click **OK**.

   For more information about adding and deleting geometric relations, see **Geometric Relations** on page 2-21 and **Display/Delete Relations** on page 2-26.

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.

7

Using SolidWorks 97Plus Effectively

The other chapters in this manual explain how to *operate* the SolidWorks 97Plus application: how to create sketches, features, parts, drawings, and so on. This chapter explains how to *use* the application efficiently to create robust models that are easy to modify if design requirements change.

This chapter helps you answer questions like:

- ❑ Where do I begin?
- ❑ How do I break a part down into features?
- ❑ Which features should I create first?

This chapter:

- ❑ Outlines the steps one should take to plan the design of a part or assembly
- ❑ Provides some tips for designing, sketching, and creating parts
- ❑ Describes using a part layout
- ❑ Explains how to use an assembly layout sketch in top down design

Think Before You Sketch

We asked some experienced SolidWorks users to tell us some of the things they learned that helped them use SolidWorks effectively. There was one consistent answer:

> Spend time thinking about the design of the part *before* you start sketching the first feature. Planning ahead saves time.

The planning can be divided into five steps:

1. Determine if the part is dependent on other parts, and then decide on the process: whether to work *top-down* (assembly first), *bottom-up* (parts first), or whether to use a *layout* (a sketch of the underlying design requirements).
2. Consider the design intent of the part.
3. Identify the base feature.
4. Identify other features required to complete the design.
5. Consider the sequence of features.

These five planning steps are described in the next few page. An example of creating a simple fixture while using this planning procedure begins on page 7-8.

Step 1: Consider dependencies and the best process

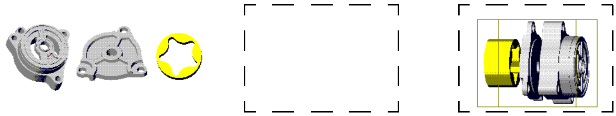
Determine if the part is dependent on other parts, and then decide on a design process. First, decide whether to begin with a *part* or an *assembly.*

If you are modeling an individual *part*, the choice is simple: you create a part file.

If you are modeling parts that make up an assembly, you can create the design using the *bottom-up* method or the *top-down* method.

- **Bottom-up Design**

  Create each part in a new part file, then combine them as components in an assembly.

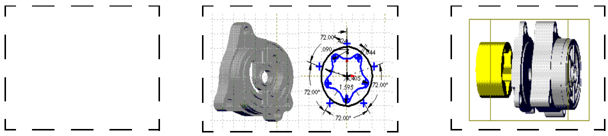


1. Create parts **2. Create assembly** **3. Insert and mate parts**

  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.

- **Top-down Design**

  Create an assembly, then create the parts in place.

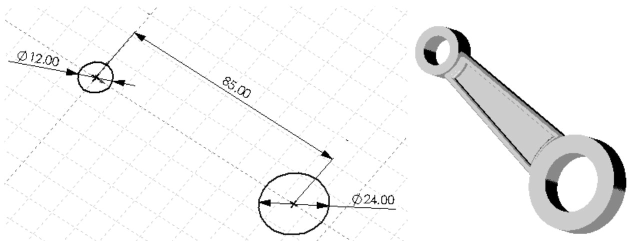


1. Create assembly **2. Create parts in place** **3. Create (or insert) other parts**

  Working *top-down*, creating the parts in context, allows you to reference faces, edges, and vertices between parts, so you can control the size or shape of one part by creating a geometric relation to another part.

- **Layout**

  Whether you are creating an individual part or an assembly, you should consider the option of starting your design with a *layout*.



Layout **Part**

  A layout is a sketch that specifies some important design requirements; for example, the critical distances between features in a part, or between components in an assembly.

  You can use the layout to control the size, shape, and location of features and parts. See **Using a Part Layout** on page 7-13 and **Using an Assembly Layout Sketch** on page 7-15

Step 2: Consider the design intent

Next, think about the design intent of the part. Ask yourself the following questions:

- **How does this part work? What does the part do?**

  For example, the part may be a mounting bracket that supports another part that weighs approximately 5 Kg.

- **What external elements does the part have to mate with (or avoid)?**

- **What is the manufacturing process for the part?**

  For example, if the part is molded, you must be aware of surfaces that should be drafted.

- **Why does the part look the way it does?**

  Consider each surface of the part and ask yourself what role it plays. A particular surface may be dictated by structural need, it may be a requirement of the manufacturing process, or it may be simply cosmetic.

- **Is there symmetry in the part?**

  Most parts contain some symmetry; some have many symmetrical elements around several planes or axes. For example, a housing may be symmetrical about a center plane. Mounting bolts may be symmetrical about the centerline of the front face.

- **Is this part a one-of-a-kind, or the base for a family of parts?**

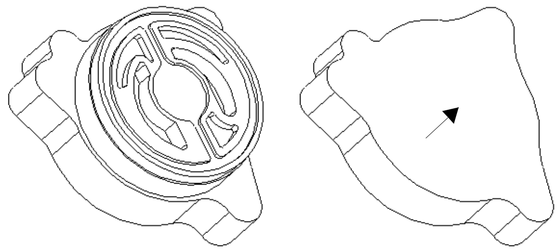
  If you are going to design many similar parts, you should try to make the design flexible and driven by a few significant dimensions.

Step 3: Identify the base feature

Visualize the part—or look at it, if you are working from an existing part or drawings—and decide which feature to create first. Here are some guidelines:

- ❑ **Make the obvious choice.**

  The shape that immediately appears to be the body of the part is probably the base feature. (The base feature is either extruded, revolved, swept, or lofted.)



Part **Extruded base feature**

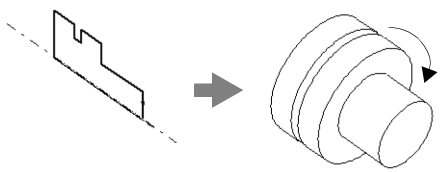
- ❑ **Consider each face.**

  Look at the top, front, and side of the part, and think about extruding it in each direction. For many parts, you can extrude any face to create the base feature. When you think about adding more features, however, it may become evident that extruding in one direction is better than extruding in any other direction.

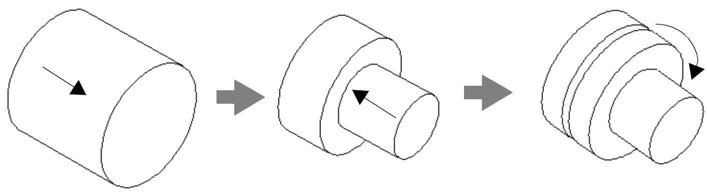
- ❑ **Consider revolved features.**

  Beginning users may think only in terms of extrusions. Often, it is better to use a revolved base feature, however.

  For example, to create the shaft shown below it is easier to create a revolved feature (**A**) than to extrude a cylinder and then remove material with other features (**B**).



A. Revolve complex sketch



B. Extrude cylinder **Extrude cut** **Revolve cut**

Step 4: Identify other required features

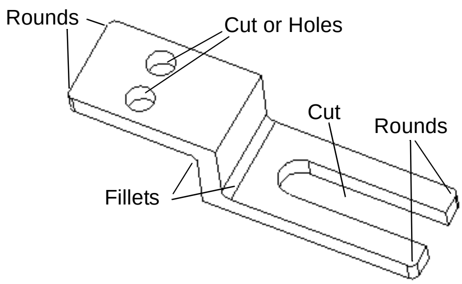
Identify the other features on the part.

- ❑ **Start with the major features.**

  In the beginning, ignore minor cosmetic features (small fillets, for example) and think about the features that are most important to the design.

- ❑ **Identify each feature by type.**

  Determine the type of feature to use to create each element of the part: bosses and cuts (extruded, revolved, swept, or lofted), shells, fillets, and so on.



- ❑ **Look for repeated features and consider using patterns.**

  When a feature is repeated (bolt holes, for example), you can add multiple individual features or create a pattern.

  If there are more than two instances of the feature, or if the features form a linear or circular array, you can probably create a pattern. Otherwise, it may be simpler to create a copy of the feature.

- ❑ **Look for symmetry.**

  Use mirroring wherever a feature or set of features is mirrored around a centerline or plane.

Step 5: Consider the sequence of features

After you identify the features, think about the process of creating them – step by step – before you start sketching.

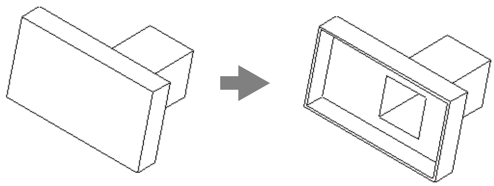
Here are some guidelines:

- **Start working from the base feature.**

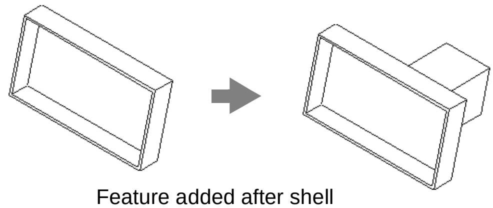
  You cannot add disjoint (disconnected) features to a part. Everything must be attached to the base feature.

- **Shell timing is critical.**

  In a shelled part, add features that appear on both the inside *and* the outside of the part *before* you shell the part.



Feature added before shell



Feature added after shell

- **Remember that you can reorder features later.**

  You do not have to be overly concerned about getting the sequence of features exactly right at this time because you can reorder features later, in most cases.

  However, by taking the time to think about the order in which you create the features before you begin sketching, you may learn things about the design intent of the part that will save you time later.

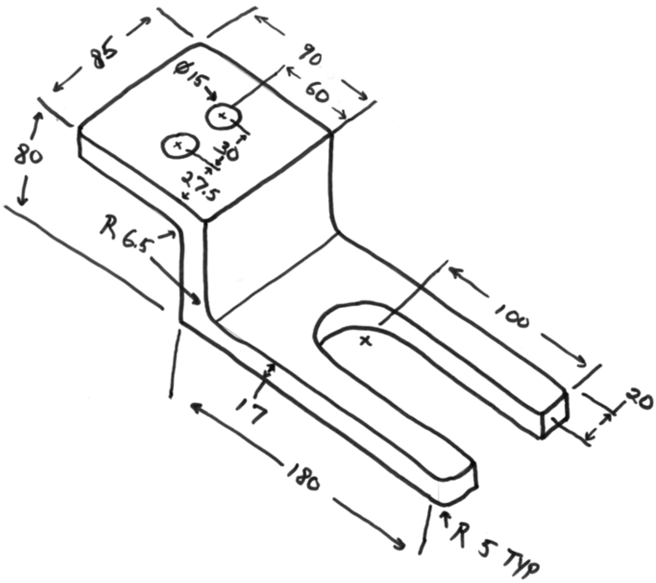
Note: While it is true that you can reorder features at a later time, you should remember that often “parent” and “child” relationships are formed between features. When this happens, you do not have flexibility in the reordering of features. For example, a hole (a “child” feature) cannot be reordered to precede the feature that it cuts through (the “parent” feature).

Example: A Simple Fixture

In this example, you have been given a drawing of an existing part and asked to create a SolidWorks model.

A new version of the part needs to be manufactured with some minor changes.

First you will analyze the part.



Step 1: Consider dependencies and the best process

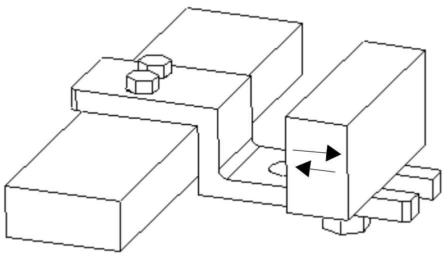
Because this is an independent part—and not part of an assembly—you can begin by opening a new part file.

Step 2: Consider the design intent

The part looks like a mounting bracket. It appears to be a machined part.

The two holes are probably used to bolt the bracket to another part.

The slot is probably used to adjust the position of the object the bracket is holding.

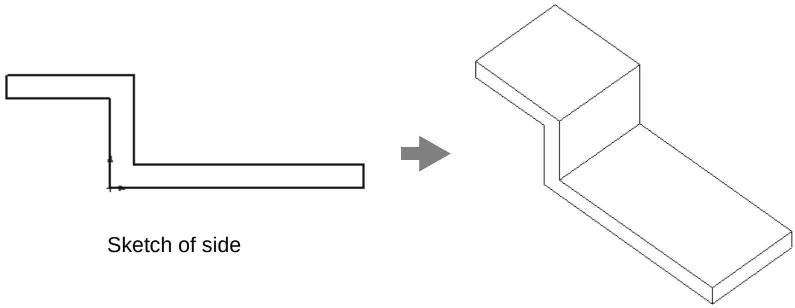


Step 3: Identify the base feature

The part is basically rectangular. This suggests that the base feature is an extruded block, not lofted, swept, or revolved.

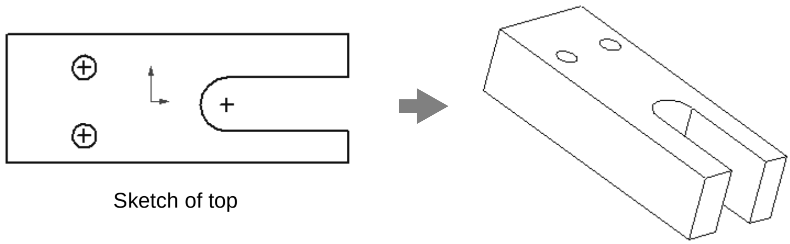
Therefore, you can look at the side, top, and front of the part and visualize extruding it in each direction:

- Extruding the sketched profile of the side view of the part seems to produce a useful base feature.



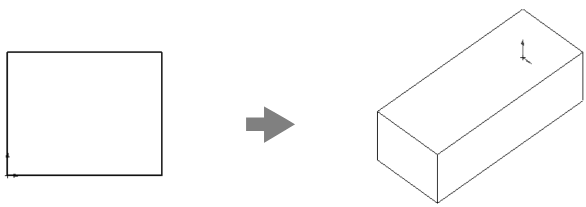
Sketch of side

- Extruding the sketched profile of the top view of the part produces a base feature that might work. However, two of the major design elements (the slot and the holes) are embedded in the same sketch, making them less flexible than if they were separate features.



Sketch of top

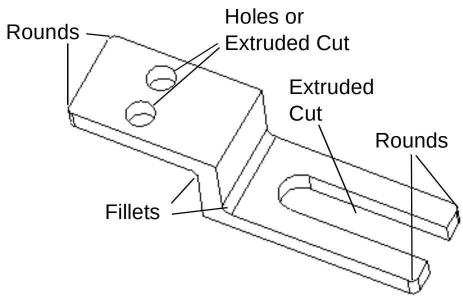
- Extruding the sketched profile of the front view of the part produces a block. This is not very useful.



So, extruding the profile of the side view seems to produce the most useful base feature.

Step 4: Identify other required features

The remaining features on the part are:



- Two holes, which could be one of the following:
  - two hole features
  - a hole feature and a mirrored copy of it
  - an extruded cut (from a sketch with two circles)
- The slot, which would be an extruded cut
- Four rounds
- Two fillets

Step 2 (Revisited): Consider the design intent

As you try to decide how to create the holes, you realize that even though the drawing shows the holes offset 27mm from the edge of the part, the actual design intent is probably for the holes to be centered between the two edges of the part, 30mm apart.

The slot also appears to be centered on the part, so the model should capture that design intent, too.

Step 5: Consider the feature Sequence

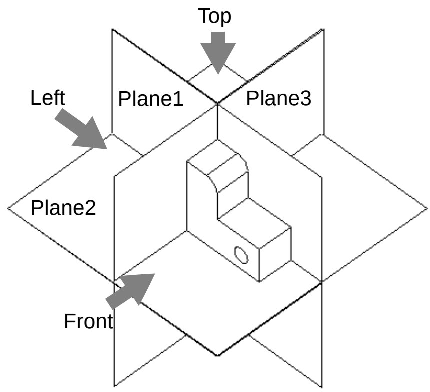
Think about the order in which you should create the various part features.

1. First, you extrude the base feature.
2. Add the two bolt holes. (Sketch the holes on top of the base and extrude a cut.)
3. Add the slot. (Sketch the slot on top of the base, and extrude a cut.)
4. Add the fillets and the rounds as the last features.

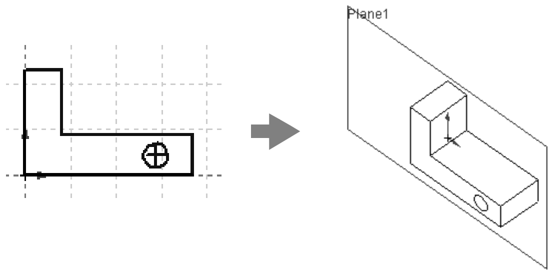
Does it matter where I start sketching?

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.

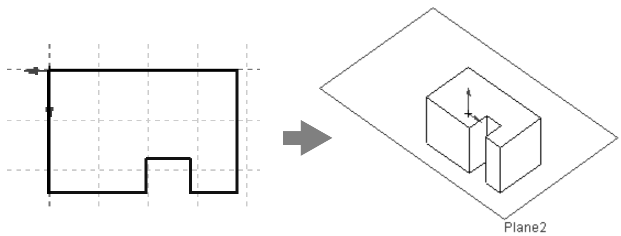
For example, if you choose ***Front** in the **View Orientation** dialog (or add a front view to a drawing), the view is normal to **Plane1**.



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**.

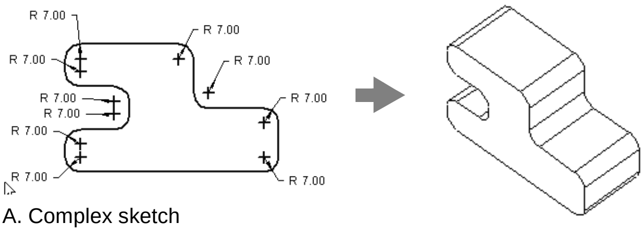
You do not have to use one of the default planes for your first sketch; you can create a new plane at any angle. The orientation of views is still determined by the default planes, however.

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.

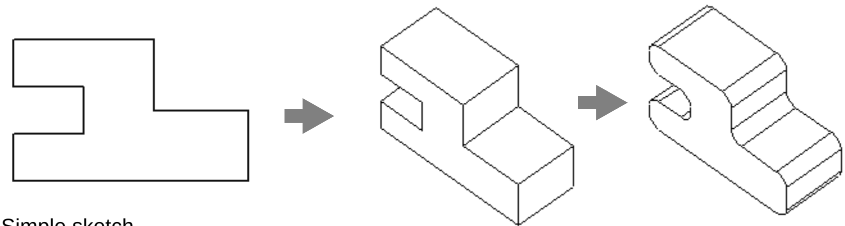
How complex should sketches be?

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.

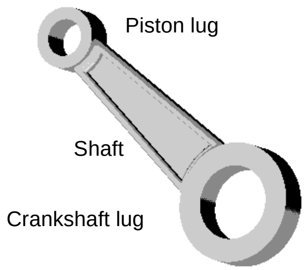
Using a Part Layout

In the previous example, you knew all the dimensions of the part. You were reproducing it, not designing it.

In most cases, however, you may have only a few critical dimensions and some other design criteria (material, weight, and manufacturing process, for example). Sometimes you do not have the dimensions of the part you are building, only the dimensions of related parts.

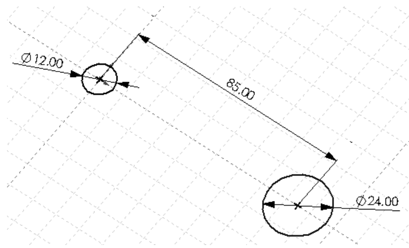
In this case you could begin with a *layout*: a sketch that captures significant related dimensions.

Consider this simplified connecting rod. The part consists of three features: the crankshaft lug, the piston lug, and a shaft that connects them.



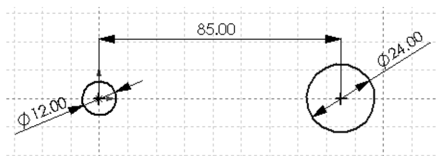
The dimension that is driving this design is not a dimension of any of the features you are building. In this case, it is the distance *between* the centers of the holes in the lugs.

The layout is simply a sketch that specifies the size of the holes and the distance between their centers.



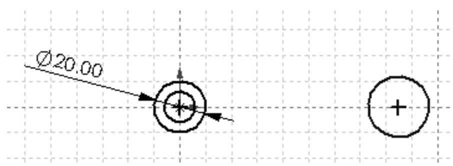
To design a part using the part layout technique:

1 Open a new part and create a layout sketch.



2 Close the sketch.

3 Open another sketch on the same plane as the layout sketch. The layout sketch is still visible behind the sketch you are working on.

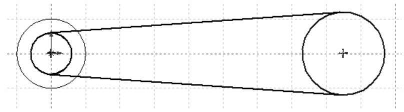


4 Draw a circle and make it coradial with the circle representing the inside of the

piston lug in the layout. Then, draw another concentric circle for the outside of the lug and dimension it.

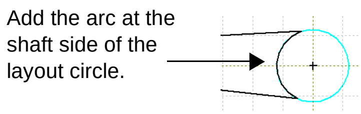
5 Extrude the lug.

6 Create a new sketch for the shaft.

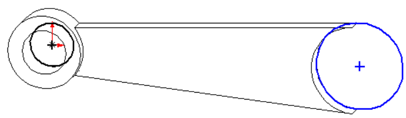


7 Sketch the edges of the shaft and make them tangent to the two circles in the layout.

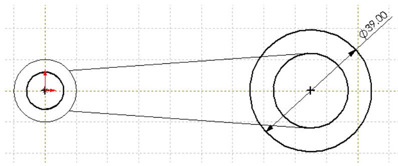
8 Add arcs at both ends of the shaft, coradial with the layout circles.

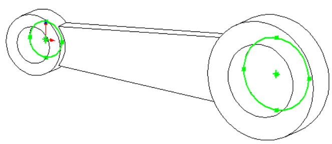


9 Extrude the shaft.

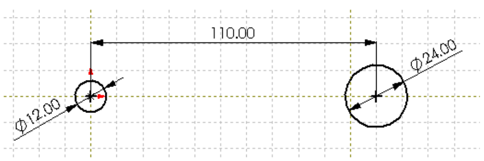


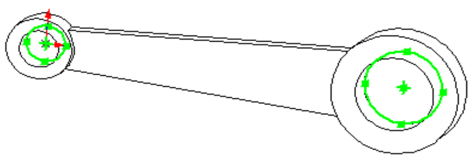
10 Repeat steps 4 and 5 to create the other lug.





If the distance between the related parts changes (the piston and the crankshaft, in this case), you can modify the length dimension in the layout sketch. Because of the coradial and tangent relations, the shaft adjusts accordingly.





Using an Assembly Layout Sketch

You can design an assembly from the top-down using layout sketches. You can construct a rough sketch showing where each assembly component belongs. Then, you can create and modify your design before a single part is created. In addition, you can use the layout sketch to make changes in the assembly at any stage in the design process.

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.

To create an assembly layout sketch, do the following:

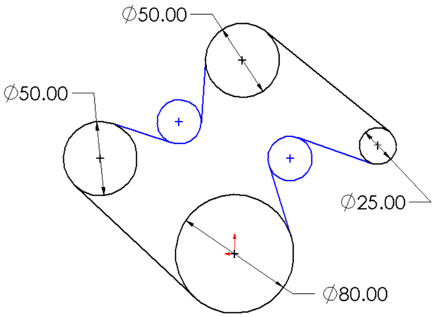
- Create a layout sketch in which various sketch entities represent parts in the assembly.
- Indicate a tentative location for each component, capturing the overall design intent.
- Reference the geometry in the layout sketch when the individual components are created.
- Use the layout sketch to define 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, all connected by a serpentine belt.

1 Create an assembly sketch.

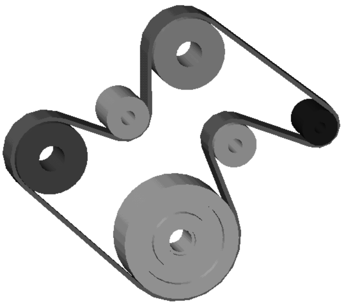
- Use circles to represent the outer rims of the pulleys.
- Use tangent lines between the circles to represent the serpentine belt.
- Dimension the size and location of each pulley.



2 For each pulley, set a circle to be coradial with the corresponding circle in the assembly sketch.

3 Extrude each circle to create the pulleys.

4 Using the layout sketch lines as a guide, create the belt around the pulley profiles.

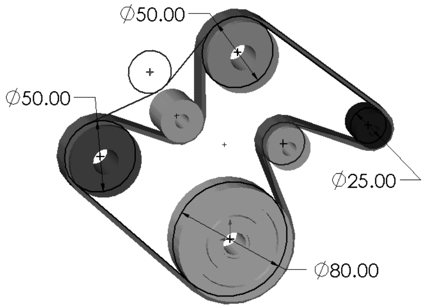


If necessary, you can make changes to the pulleys either in the part document or in the assembly.

To make changes to your design:

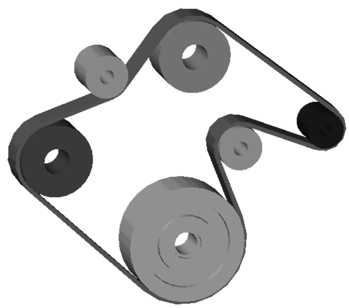
At this point in the design, there is a belt and a set of pulleys *whose locations are dictated by the assembly sketch.* The advantage of this type of design is clear when you have to make changes to the design.

For example, working within the layout sketch you can change the layout to



- Rearrange the pulleys so that a slipping pulley has more of its surface covered by the belt
- Make a 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 your design had been done without a layout sketch, you would have had to make many dimension changes or drag components within the assembly (followed by rebuilds) to see the result.

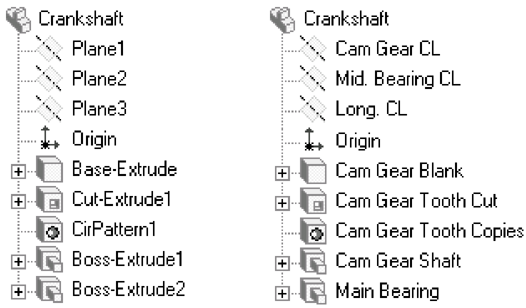
The layout sketch does not have to be the master plan for a design. If you had a model of the rest of the engine with some fixed pulley locations, you could relate your layout sketch to the model. Then, to design the belt system, you could set the corresponding circles in the layout sketch to be coincident with the known location of the pulleys in the existing model. You would still have the freedom to drag the other pulleys to different locations and your assembly would update automatically to show your changes.

Design Tips

Here are some additional tips for using SolidWorks effectively.

- **Use meaningful names.**

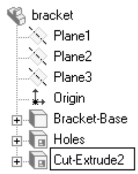
  Develop the habit of giving each feature a meaningful name. This makes it easier to edit the model, especially if more than one person is working on the same project.



Default names   Meaningful names

  Click **Tools, Options**, and select the **General** tab. In the **FeatureManager Design Tree** section, select the **Name feature on creation** option. With this option as the default, when you create a new feature, its name in the FeatureManager design tree is automatically selected and ready for you to enter a name of your choice.

  Otherwise, to rename a feature (or a sketch or plane), click two times on the name in the FeatureManager design tree. When the border appears around the name, select the text and enter a new name.



- **If it is possible, create any holes near the end of the design process.**

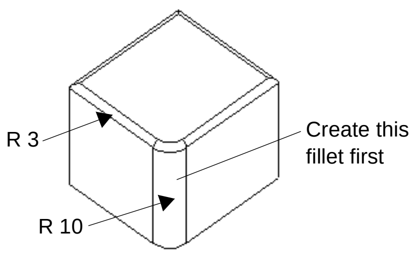
  This helps you avoid creating features that inadvertently add material inside an existing hole.

- **Add larger fillets before smaller ones.**

  In cases where several fillets converge at a vertex, create the larger fillets first.

- **Save cosmetic fillets for last.**

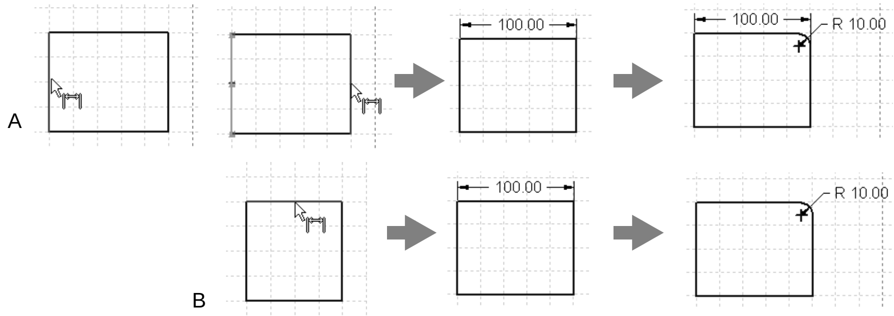
  Try to add cosmetic fillets after most other geometry is in place. If you add them earlier, it takes longer to rebuild the part.



- **Dimension between edges.**

  It is better to dimension *between* edges in a sketch (**A**), instead of dimensioning the length of an edge (**B**).

  If you dimension the length of the edge, the dimension can be lost if you modify the edge (by adding a sketch fillet, for example).



- **Use Configuration to produce “as cast” drawings.**

  When modeling a cast part that has machined features (drilled holes, for example), you can use the **Configuration** function to suppress the machined features and produce both “as manufactured” and “as cast” drawings.

- **Make your design process reflect the manufacturing process.**

  Try to create all of the cast features *before* the machined features, in the same order in which you would manufacture them. If you create the features out of order, you may make the mistake of creating cast features as the dependents of machined “parent” features. In this case, you would not be able to use **Configuration** to suppress machined features while keeping all cast features unsuppressed. For more information, see **Working with Part Configurations** on page 4-8.

- **Draw sketches to scale.**

  It is a good practice to draw the first line in your sketch at roughly the correct size. (The easiest way to do this is by watching the line length feedback that appears next to the cursor as you draw.)

  Otherwise, the shape of the sketch geometry can change when you begin adding dimensions and setting them to the correct values.

8

Sheet Metal

This chapter introduces the SolidWorks sheet metal features and describes:

- SolidWorks sheet metal capabilities
- Bend types
- Bend allowances
- Sheet metal features in the FeatureManager design tree
- Rolling back and rebuilding the design
- Auto Relief
- Creating a sheet metal part using sharp bends
- Adding additional features
- Creating a sheet metal part using round bends
- Creating a sheet metal part from a flat model
- Editing bends
- Creating a flat pattern configuration
- Sheet metal drawings

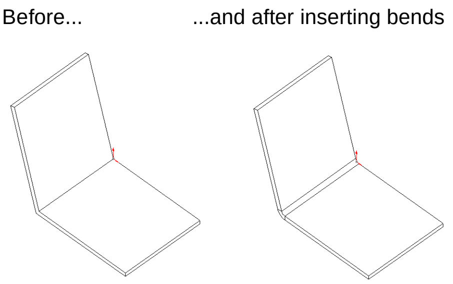
SolidWorks Sheet Metal Capabilities

Designing Sheet Metal Parts in Three Dimensions by Specifying Spatial Requirements

Sheet metal parts are generally used as enclosures for components or to provide support to other components.

You can design your 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.

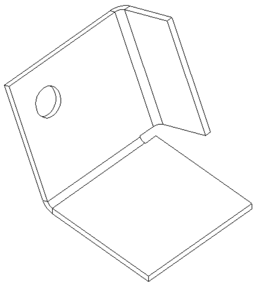
Creating Bends



Using sheet metal capability, 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.

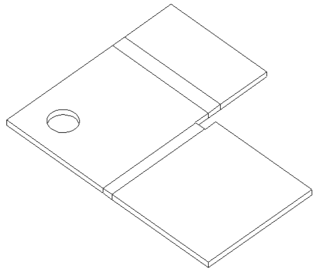
Creating Bend Reliefs, Walls, and Tabs



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 your design is complete (or at any time during the design process) you can create a flat pattern of your sheet metal part.

Bend Types

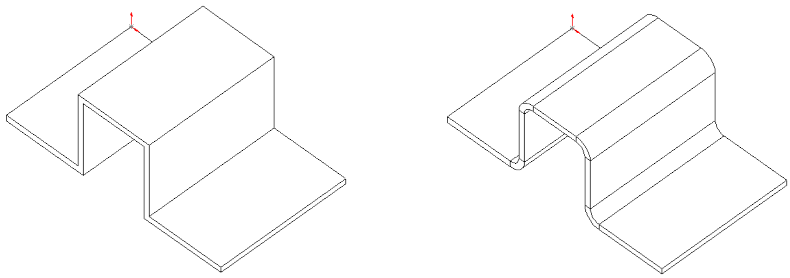
There are three types of bends in SolidWorks:

- Sharp Bends
- Round Bends
- Flat Bends

Sharp Bends

A *sharp bend* is created by adding bends to all sharp corners in the model.

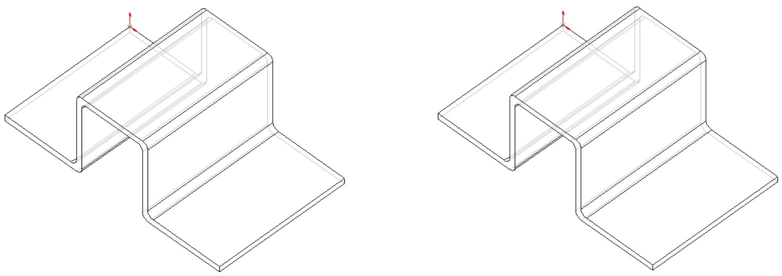
In this example, the model on the left has sharp corners. When you insert bends, you create the model on the right. See **Creating a Sheet Metal Part Using Sharp Bends** on page 8-10 for more information.



Round Bends

A *round bend* is created from filleted corners.

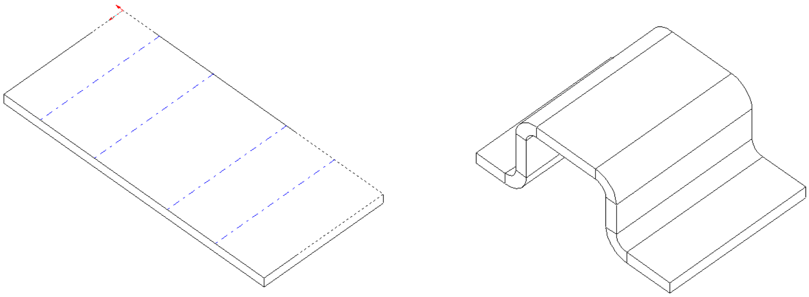
In this example, the model on the left has filleted corners. When you insert bends, the change is not visible in the bent-up state of the model. However, bend lines and allowances have been added to the part, and will be apparent when the model is flattened. See **Creating a Sheet Metal Part Using Round Bends** on page 8-13 for more information.



Flat Bends

Flat bends are created from bendlines specified in the flat state of a sheet metal part.

In this example, bend lines are sketched as shown in the flat model on the left. Inserting bends creates the model on the right. See **Creating a Sheet Metal Part from a Flat Model** on page 8-13 for more information.



Bend Allowances

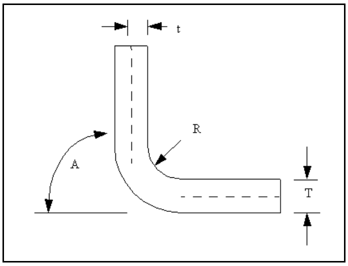
There are three ways to specify a sheet metal bend allowance. You can

- 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.

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.

Sheet Metal Features in the FeatureManager Design Tree

There are three features 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. When you create a sheet metal part using **Insert, Features, Bends,** two distinct stages are applied to the sheet metal part. First, the part is flattened and a bend allowance is added. Then, the flattened part is refolded to create the formed, folded version of the part.

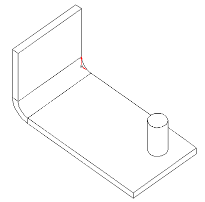
Sheet-Metal contains the definition of the sheet metal part itself and represents the result of the **Insert, Features, Bends** operation. This feature stores the default bend parameter information (thickness, bend radius, bend angle, and auto relief offset ratio) for the entire part.

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 bendlines of all sharp and round bends generated by the system. This sketch cannot be edited but can be hidden or shown.

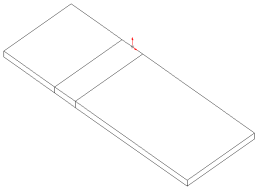
Process-Bends represents the transformation of the flattened part into the finished, formed part. Bends created from bend centerlines specified in the flat are listed under this feature. **FlatSketch,** listed under **Process-Bends**, is a placeholder for these bend centerlines.

Features after the **Process-Bends** 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 8-8)

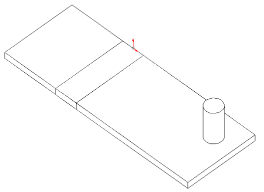
For example, in the sheet metal part shown here, the cylindrical boss was added after the **Process-Bends** feature.



If you rollback to **Process-Bends**, the flattened part does not include the boss.



To see the cylindrical boss in the flattened state, reorder the boss to occur before the sheet metal feature.



Recommended Design Approach

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:

- Always design your part in the bent-up state.
- Add **Insert, Features, Bends** only after your functional design is complete; that is, add all your 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.

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):

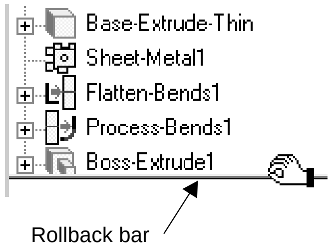
1. In the FeatureManager design tree, select the feature you want to rollback to.
2. Click **Edit, Rollback** or click the **Rollback** icon on the Dependency toolbar.

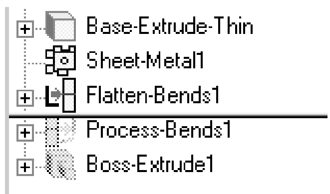
To rollback by dragging the rollback bar (Method 2):

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. Note also that the rolled back icons are grey.

Throughout the following sections, whenever you need to perform a rollback, you can use any of these methods.

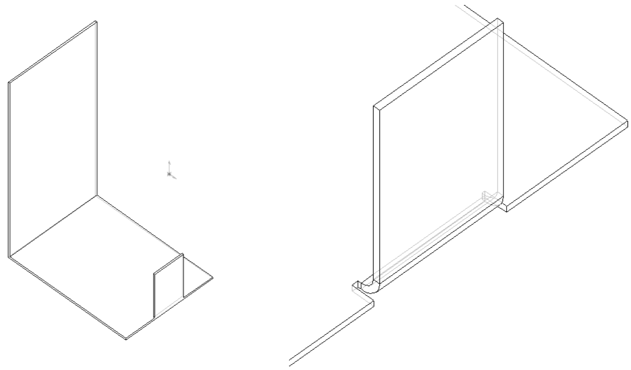
To rebuild (roll forward):

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.

Auto Relief

You can automatically add relief cuts wherever needed when inserting bends.

For example, you can use the autorelief 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 is cut needed, only one is added.) The cuts added are of the minimum size required to insert the bend and flatten the part.

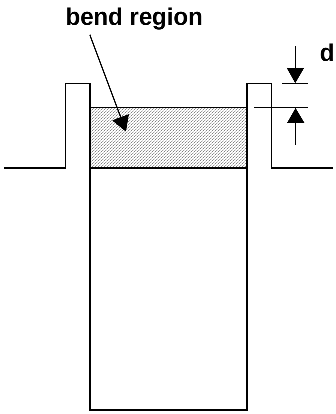
When you choose the **Autorelief** option, you must specify the **Offset Ratio.**

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.

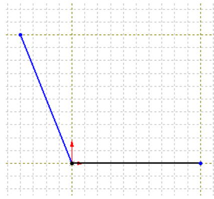


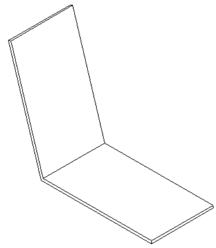
The higher the value of the offset ratio, the larger the size of the relief cut automatically added during insertion of bends.

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 which has a uniform thickness, and a thin feature extrusion is a quick way to create such a base.

To create a thin-feature base:

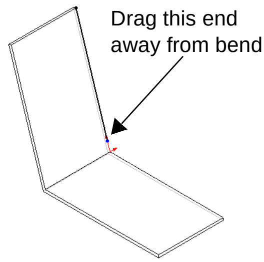


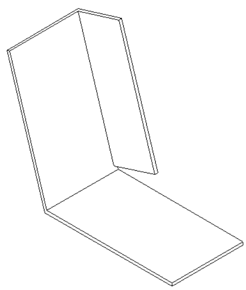


1. Create a new part, open a sketch, and sketch an open profile for the base.
2. Click **Insert, Base, Extrude**.
3. In the **Extrude Feature** dialog box, specify the desired **Depth** and **Type**.
4. On the **Thin Feature** tab, specify the desired **Type** and **Wall Thickness**.
5. Click **OK**.

   For more information about thin feature extrusions, see **Thin Features** on page 3-6.

To add a wall:



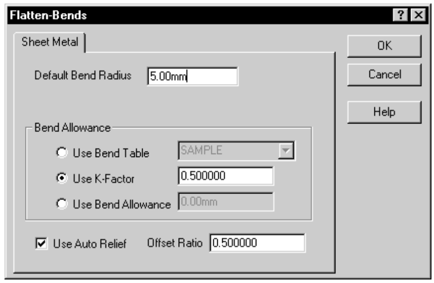


1. Open a sketch on the face of the part where the new wall will be attached.
2. Select the edge of the model and click **Convert Entities**  or **Tools, Sketch Tools, Convert Entities**.
3. Drag the vertex nearest to the bend a small distance away from the bend to allow for the bend radius.
4. Click **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 in this manner before inserting the bends.

To insert the bends:

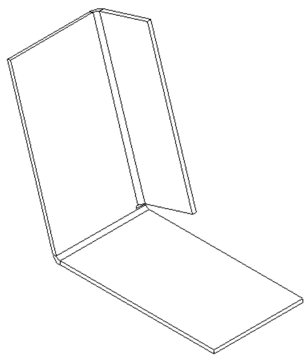
To insert bends, you must first pre-select a *fixed face*. The fixed face remains in place when flattening the part during the creation of the **Flatten-Bends** feature.



1. In the model, select the face you want to remain fixed.
2. Select **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 8-5 for more information.
   - If you select **Use K-Factor**, specify the value based on the calculation. See **K-Factor** on page 8-4 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 8-9 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.



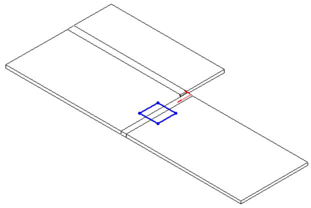
7. Double-click the **Flatten-Bends** feature (or click the ⊞ beside its name) to expand the feature.

   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 holes, cuts, tabs, and so forth. You can rollback to the flattened state of a sheet metal part by clicking the **Process-Bends** feature and then clicking **Edit, Rollback**, or **Rollback** on the Dependency Editing toolbar, or by dragging the rollback bar in the FeatureManager design tree.

To create a cut across a bendline:



1 Rollback the **Process-Bends** feature.

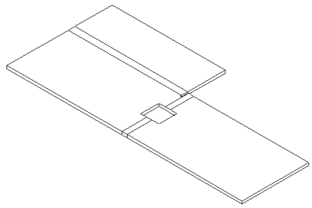
   The part is shown in the flattened state.

2 Select the top face of one of the flat regions and open a sketch.

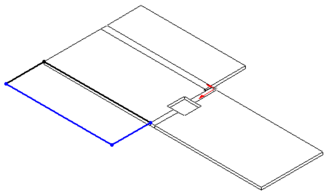
3 Sketch the profile of the cut.

4 Click **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.

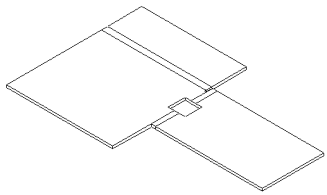
2 Sketch a tab with one side coincident with a model edge.

3 Click **Insert, Boss, Extrude**.

4 In the **Extrude Feature** dialog box:

   - Click **Link to Thickness** to set the boss thickness equal to the base thickness.

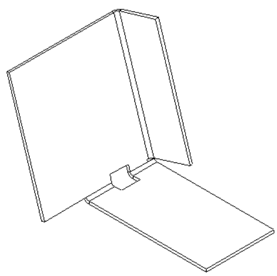
   - Click **Reverse Direction** if the extrusion is extending the wrong way.



5 Click **OK**.

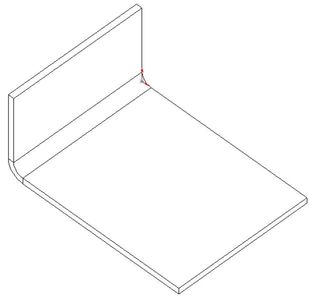
6 Rebuild the part by dragging the rollback bar to the bottom of the FeatureManager design tree.

   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, Features, Bends**.
4. In the **Flatten-Bends** dialog box:
   - Specify a **Round Radius** of zero.
   - Specify a **Bend Allowance** method and value.
5. Click **OK**.

   Note that a **RoundBend** appears in the FeatureManager design tree under **Flatten-Bends**.

Creating a Sheet Metal Part from a Flat Model

While it is recommended that you 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 will need to know the developed length required for your part and the locations of the bend lines for all bends in the part.

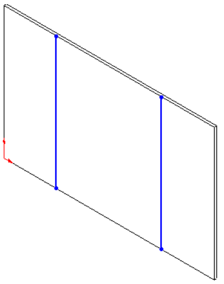
To create a sheet metal part from a flat model:

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, 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.

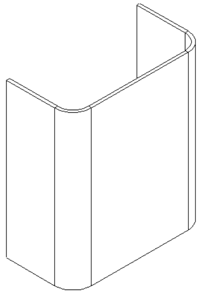
6 Right-click the **Flat-Sketch** under **Process-Bends** and select **Edit Sketch**.

7 Sketch lines or centerlines where you want bendlines.



8 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 Chapter 4, *Working with Parts,* for information about creating a configuration.)

2 Select the last **Process-Bends** feature in the model and all features after it.

3 Click **Suppress**  on the Dependency Editing toolbar, or **Edit, Suppress**.

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. See Chapter 6, *Drawing and Detailing*, for more information on creating drawings.

To create a drawing of a flat pattern:

1. Create a part configuration showing a flat pattern.
2. 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.
3. Create a new drawing.
4. Click **Insert, Drawing View, Named View**.
5. Change to the part window, click anywhere, then select the desired view from the **View Orientation** list.
6. Return to the drawing window and click to place the view on the sheet.

To remove display of the bends:

1. In the drawing window, right-click the drawing view in the FeatureManager design tree.
2. Select the **Remove Tangent Edges** option.

To show the bend lines in the view of the flat pattern:

1. In the part window, expand the **Flatten-Bends** and **Process-Bends** features.
2. Right-click **Sharp-Sketch** and **Flat-Sketch** in the FeatureManager design tree, and click **Show**.

9

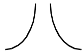
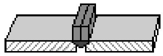
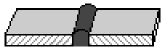
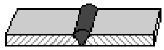
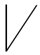
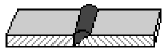
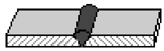
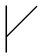
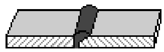
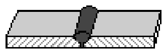
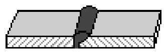
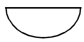
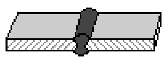
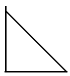
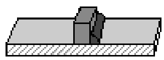
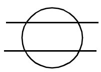
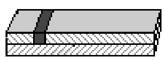
Welding

This chapter introduces the SolidWorks welding feature and describes:

- ❑ Weld types
- ❑ Top surface delta and radius calculations
- ❑ General procedures for creating a weld
- ❑ Creating V-Butt, Square Butt, Fillet and Multi-face welds
- ❑ Editing a weld

Weld Types

You can add a variety of weld types to an assembly. The weld types and their symbols are shown in the following table:

Weld Type	Symbol	Illustration
Butt		
Square butt		
Single-V butt		
Single-bevel butt		
Single-V butt with broad root face		
Single-bevel butt with broad root face		
Single-U butt with parallel or sloping sides		
Single-J butt		
Backing run		
Fillet		
Seam		

Top Surface Delta and Radius Calculation

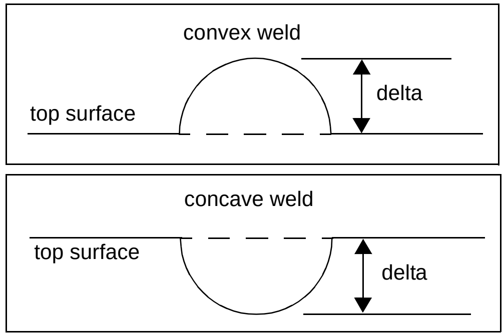
When you are adding any type of weld to the model, you must specify the top surface delta. For a fillet weld, you also specify the radius, and for a backing run weld, you specify the bottom surface delta.

Top Surface Delta

The top surface delta is the distance between the edge of the bead and the top surface of the welded components.

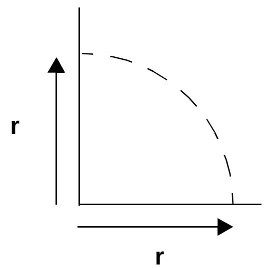
These examples illustrate the top surface delta for a convex weld and a concave weld. The calculation is the same but the distance is measured *below* the top surface for a concave weld rather than *above* for a convex weld.

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 corner of the parts to be welded.



General Procedure for Welding Components

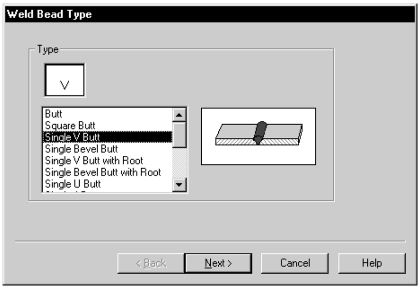
To weld two components:

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 sections that follow 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.

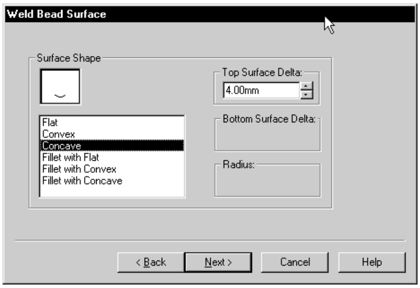
4 Click **Next**.



5 In the **Weld Bead Surface** dialog box:

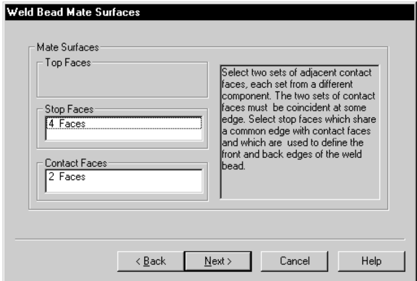
- Select a **Surface Shape** from the list.
- Specify the **Top Surface Delta**.
- Depending on the type of weld you are adding, specify the **Bottom Surface Delta** or **Radius** as required.

6 Click **Next**.



7 In the **Weld Bead Mate Surfaces** dialog box:

- Click the **Contact Faces** box, then select the contact faces for the weld.
- Click the **Stop Faces** box and select the two sides of each component (four sides total) that define the beginning and end of the weld bead.
- Click the **Top Faces** box and select the top face of each component.



8 Click **Next**.

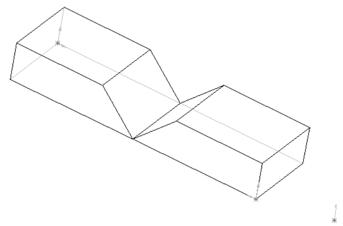
9 In the **Weld Bead Part** dialog box, accept the default name for the **Weld Bead Part** or enter a new part name.

10 Click **Finish**.

V-Butt Weld

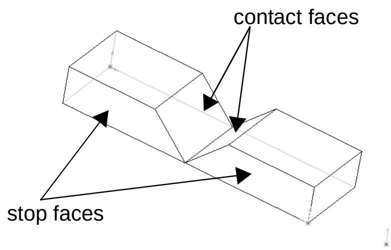
The assembly shown here calls for a V-Butt type weld. The components are mated using three coincident mates:

- Between the edges where the contact faces meet
- Between the top faces
- Between the stop faces on the front of the assembly



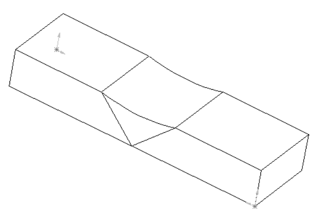
To add a V-Butt weld:

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** of **Concave**, 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 sides 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**.

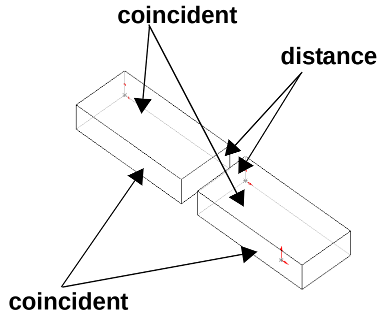
   The weld bead is added between the two components.



Square Butt Weld

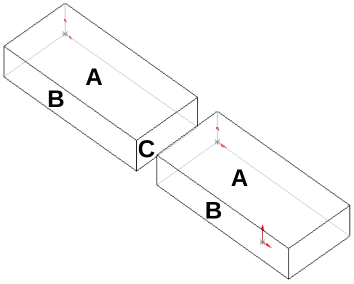
The assembly shown here calls for a Square Butt type weld. The components are mated as follows:

- A distance mate between the contact faces
- A coincident mate between the top faces
- A coincident mate between the stop faces



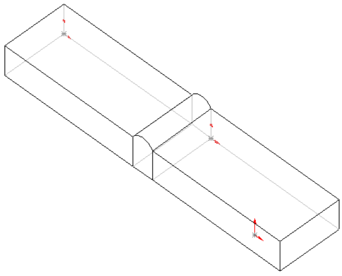
To add a square butt weld:

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, click **Convex**, specify the **Top Surface Delta**, and click **Next**.
4. 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 face opposite it).



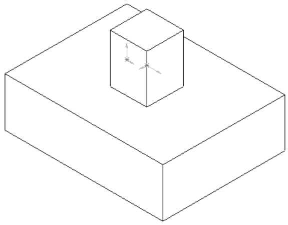
5. Click **Next**.
6. In the **Weld Bead Part** dialog box, name the weld bead part or accept the default name.
7. Click **Finish**.

   The weld bead is added between the components.



Fillet Weld

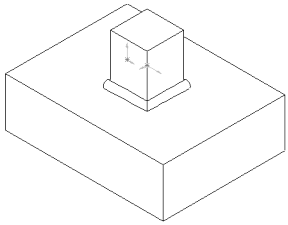
Begin with an assembly like this. Use a coincident mate between the contact faces.



To add a fillet weld:

1. Click **Insert, Assembly Feature, Weld Bead**.
2. In the **Weld Bead** dialog box, select **Fillet**, then click **Next**.
3. In the **Weld Bead Surface** dialog box, click **Convex**, specify the **Top Surface Delta** and the **Radius**, then click **Next**.
4. In the **Weld Bead Mate Surfaces** dialog box, click the horizontal top face of the lower component, then click the four vertical faces of the upper component.
5. Click **Next**.
6. In the **Weld Bead Part** dialog box, either accept the default bead name or enter a different name.
7. Click **Finish**.

   The fillet weld bead is added to the assembly.

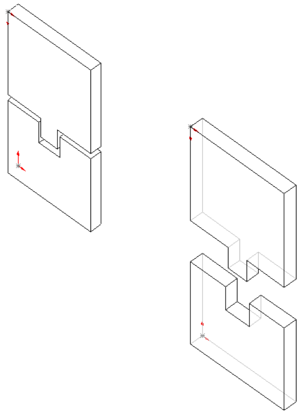


Multi-Faced Weld

You can weld components that have multiple faces such as the ones shown here.

When performing a weld operation on multi-faced components, it is easier to select the correct contact faces if the distance between the components allows you to easily see the faces.

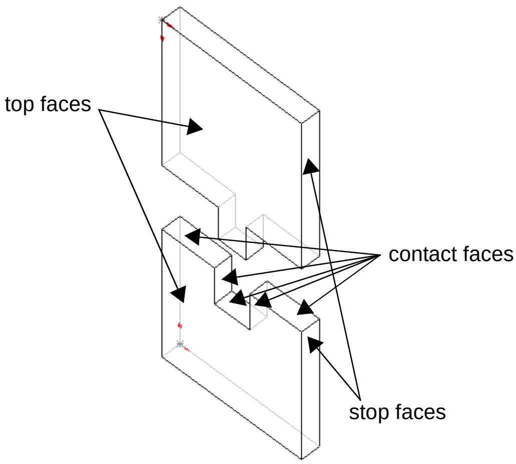
In the FeatureManager design tree, right-click the distance mate between the contact faces, select **Edit Definition,** and increase the distance mate until you can easily see the contact faces.



Viewing the assembly with hidden lines in gray may also make face selection easier.

To add a multi-faced weld:

1. Click **Insert, Weld Bead**.
2. In the **Weld Bead** dialog box, select **Square Butt**, then click **Next**.
3. In the **Weld Bead Surface** dialog box, click **Convex**, set the **Top Surface Delta** to a large value, then click **Next**.
4. In the **Weld Bead Mate Surface** dialog box, select the faces as indicated in the illustration.
5. Click **Next**.
6. Accept the default name for **Weld Bead Part** or enter a new part name.
7. Click **Finish**.



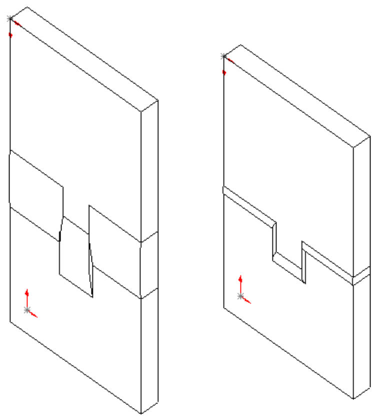
The weld bead is added between the two components.

8 Edit the distance mate between the two components:

   a) In the FeatureManager design tree, right-click the distance mate, and select **Edit Definition**.

   b) Specify the desired distance and click **Apply**.

The weld bead between the two components is updated to reflect the final mating distance.



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 Right-mouse click and select **Edit Assembly** to return to the **Edit Assembly** mode.

10

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 document including a part, assembly or drawing 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

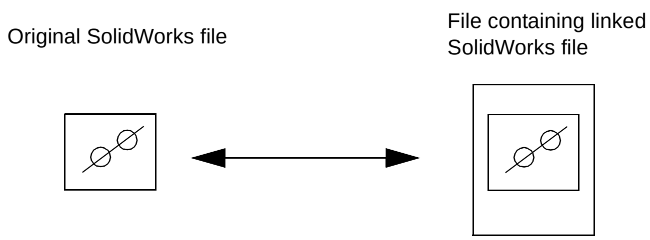
When using OLE, you can *link* or *embed* files. Linking files allows you to continuously change the contents of a file in all places where it appears without having to edit each individual occurrence. Embedding a file allows you to keep the edits to the file specific to the place in which you embedded 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.



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.

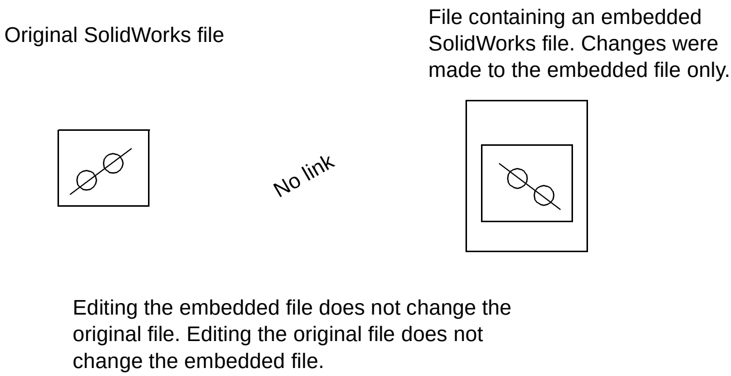
Note: Keep track of the files to which you link data to avoid changing information in places you do not expect.

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.



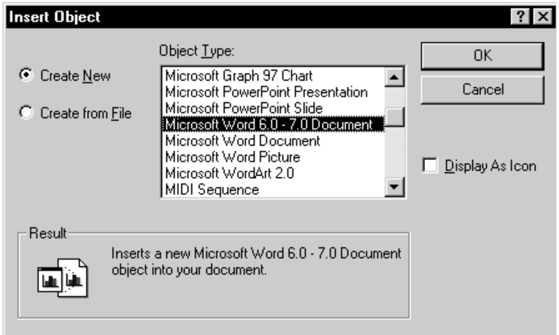
Using Data from Other Applications in SolidWorks

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 spreadsheet. You can then link or embed the math application or Excel spreadsheet to the desired dimension of the part in SolidWorks. Refer to the “Using a Design Table” chapter in *Learning to Use SolidWorks 97Plus* for details.

To use data from another application in a SolidWorks file:

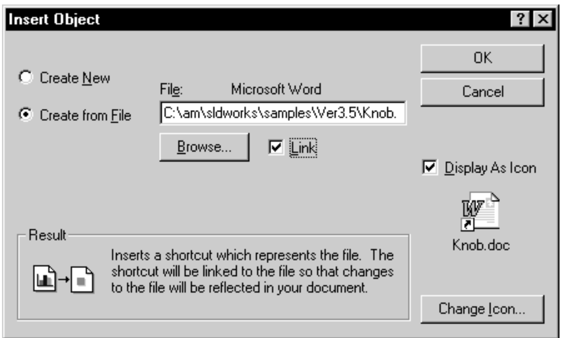
1 With a SolidWorks file open, click **Insert**, **Object**.

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**.

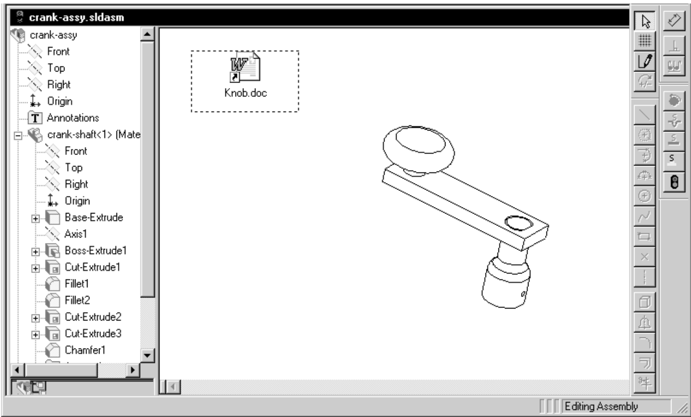
3 To link the object to the SolidWorks file, click the **Link** check box in the **Insert Object** dialog. Otherwise the object is embedded.

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.

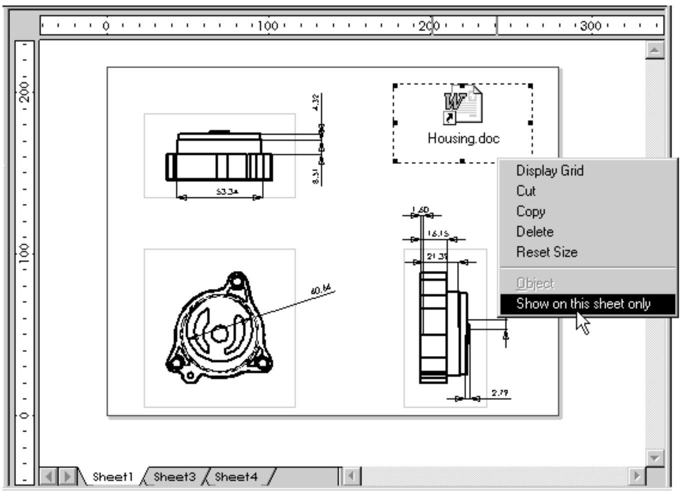
A Microsoft Word File in an Assembly

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**.

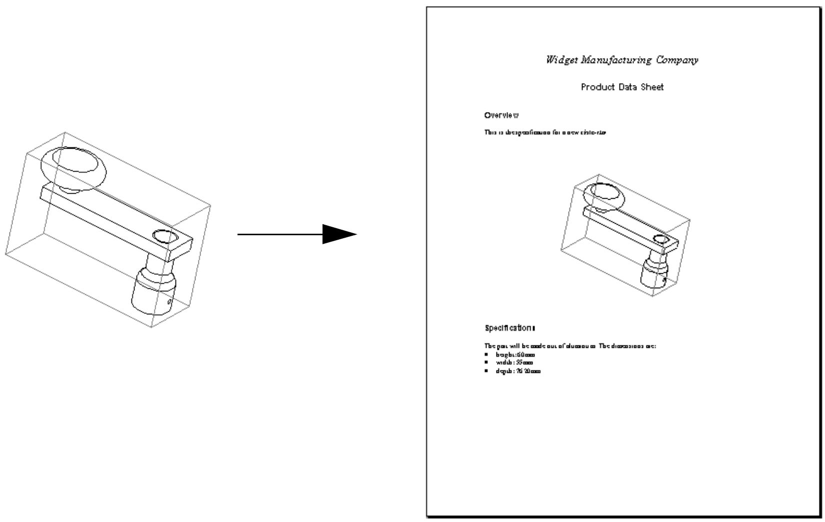
   A dialog box appears allowing you to insert an existing object or to dynamically create and insert a new object. The dialog box that appears depends on the application you are using.

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.

11

Importing and Exporting Files

This chapter describes these SolidWorks file import and export options:

- Setting import/export options
- Importing and exporting an IGES file
- Exporting a Parasolid file
- Exporting an STL file
- Importing and exporting an ACIS file
- Importing and exporting a DXF/DWG file
- Importing and exporting a VRML file

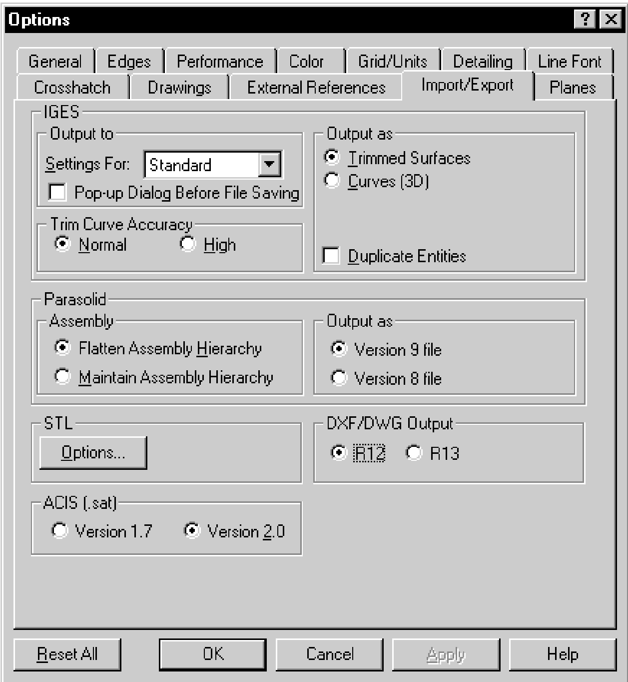
Setting Import/Export Options

You can export SolidWorks files in a number of formats for use with other applications. Before exporting a file from SolidWorks, you need to check the **Options** settings to meet the needs of the target application.

To set Import/Export Options:

1 Click **Tools, Options**.

2 Click the **Import/Export** tab.



On this tab, you can set parameters for exporting files in these formats:

- IGES
- Parasolid
- STL (stereolithography)
- ACIS
- DXF/DWG

3 Choose options as desired, and click **OK**.

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 also export a SolidWorks solid or selected faces in the IGES format for use in other applications. The following table shows the IGES entity types that are supported for import and export.

IGES Entity Type	Entity Name
144	Trimmed (parametric) surface
142	Curve on a parametric surface
128	Rational B-spline surface
126	Rational B-spline curve
122	Tabulated cylinder
120	Surface of revolution
118 Import only	Ruled surface
112 Export only	Parametric spline curve
110	Line
102	Composite curve
100	Circular arc

Note: When importing IGES files, the wireframe entities (IGES types 126, 110, 102, 100) are only supported if they are used to define one of the surface types (144, 128, 122, 120, 118). These entities are not supported for standalone wireframe IGES import.

When exporting IGES files from the source system for use in SolidWorks, export the surfaces as trimmed surfaces (Entity type 144) or as untrimmed surfaces (Entity Types 118, 120, 122, and 126). For the best results, use trimmed surfaces.

To create a base feature 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**.

   The surfaces in the file are read and an attempt is made to “knit” them into a solid.

   - If the attempt 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 attempt fails, the IGES surfaces are grouped into one or more reference surfaces (named **RefSurface1, 2, ...**) in a new part file.

Error and report files (**.rpt** and **.err**) containing information about any unsupported entities and errors encountered are written in 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 **RefSurface** feature is added to the part. The surface is positioned relative to the part origin, using the global coordinates in the IGES file.

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 in this manner, 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 3-46.

Exporting IGES Files (Parts Only)

IGES settings

- **Trimmed Surfaces.** The faces of the solid part 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

  If you select **Trimmed Surfaces**, click the **Settings For** drop-down list and select the export format for the application that will use the IGES file.

- **Curves (3D).** The solid body is converted to 3D wireframe representation in the 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 3-D curve.

Type of 3-D 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.

- **Pop-up Dialog Box Before File Saving.** Check this option if you want the ability to choose a new format every time you save an IGES file.

- **Trim Curve Accuracy.** Select **Normal** or **High**.

To export an IGES file:

1. Select the faces you want to export. (If no faces are selected, the entire solid is exported.)
2. Click **File**, **Save As**.
3. Select **IGES Files (*.igs)** in the **Save File Type** box.
4. Enter a name for the file. (SolidWorks automatically adds the **.IGS** extension.)
5. Click **Save**.
6. If you selected any faces, indicate whether you want to export the **Selected Face(s)** or the **Complete Body**, then click **OK**.

Exporting Parasolid Files (Parts and Assemblies)

Parasolid settings

- **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 which contain imported features.
- **Maintain Assembly Hierarchy.** A file in which the assembly hierarchy is maintained mirrors the original assembly, its nested subassemblies, and its parts.
- **Version.** Select the appropriate version for the target system.

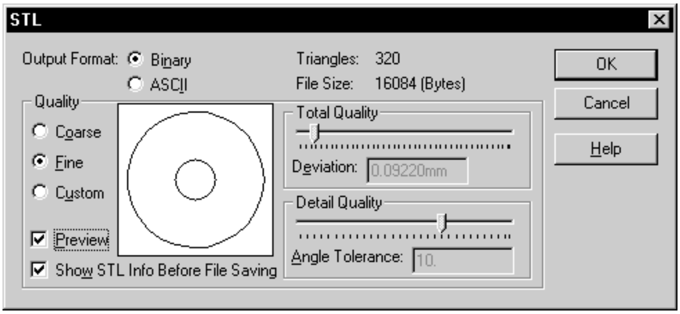
To export a Parasolid file:

1. Click **File**, **Save As**.
2. Select **Parasolid Files (*.x_t)** or **Parasolid Binary Files (*.x_b)** in the **Save File Type** box.
3. Enter a name for the file. (SolidWorks automatically adds the **.X_T** or **.X_B** extension.)
4. Click **Save**.

Exporting STL Files (Parts Only)

STL settings

To access STL settings, click **Options** in the **STL** area of the **Import/Export** tab.



- ❑ **STL Output Format.** Choose **Binary** or **Ascii**.

- ❑ **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**. **Detail Quality** 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 will vary as a result of the settings.

  Note: Files generated with higher accuracy settings (increased tessellation) are larger in size and slower to generate. Experiment with these **STL Quality** settings to determine the best settings for your own STL equipment. Click **Reset All** to return all settings to their preset, default values.

- ❑ **Preview.** A faceted version of the model is displayed and the number of triangles and file size (in bytes) is reported in the dialog box. (You may need to move the dialog box to the side to view the faceted model.)

- ❑ **Show STL Info before saving.** Choose this option if you want to see the file size and number of triangles each time you save an STL file. In the pop-up, click **Yes** to save the file in the indicated location; click **No** to cancel the operation.

To export an STL file:

1 Click **File**, **Save As**.

2 Select **Stl Files (*.stl)** in the **Save File Type** box.

3 Enter a name for the file. (SolidWorks automatically adds the **.STL** extension.)

4 Click **Save**.

Importing/Exporting ACIS Files (Parts Only)

ACIS settings

❑ **Version.** Select the appropriate version for the target system.

To export an ACIS file:

1 Click **File**, **Save As**.

2 Select **ACIS Files (*.sat)** in the **Save File Type** box.

3 Enter a name for the file. (SolidWorks automatically adds the **.SAT** extension.)

4 Click **Save**.

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**.

The selected file is opened in a part window.

Importing/Exporting DXF/DWG Files (Drawings Only)

DXF/DWG settings

- ❑ **Version.** Select the appropriate version for the target system.

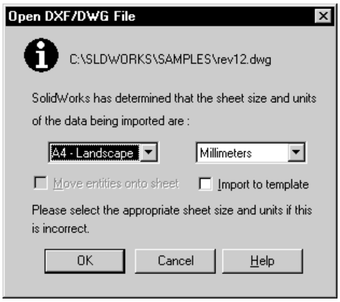
To export an DXF/DWG file:

1. Click **File**, **Save As**.
2. Select **DXF Files (*.dxf)** or **Dwg Files (*.dwg)** in the **Save File Type** box.
3. Enter a name for the file. (SolidWorks automatically adds the **.DXF** or **.DWG** extension.)
4. Click **Save**.

To import a DXF/DWG file:

1. Click **File, Open.**
2. In the **Open** dialog box, set **Files of Type** to **DXF Files (*.dxf)** or **Dwg files (*.dwg)**.
3. Browse to the desired file, and click **Open**.

   The **Open DXF/DWG File** dialog box appears.



4. Adjust the sheet size and units settings if necessary, and click **OK**.

   The selected file is opened in a drawing window.

For an example of importing a **.dwg** file, see the *Learning to Use SolidWorks 97Plus Tutorial*, Chapter 10.

Importing/Exporting VRML Files (Parts and Assemblies)

VRML files can be used to display 3D graphics over the Internet.

To export a VRML file:

1. Click **File**, **Save As**.
2. Select **VRML Files (*.wrl)** in the **Save File Type** box.
3. Enter a name for the file. (SolidWorks automatically adds the **.WRL** extension.)
4. Click **Save**.

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 is opened in a part window.

12

Library Features

A *library feature* is a feature, or combination of features, that you create once and then save in a library for frequent use. If you often use a particular set of features in your parts, you may find it convenient to store that set as a library feature to re-use whenever you need it. You could use several library features as building blocks to construct a single part. Or, you could create commonly used features such as punches, holes, and slots and save them as library features to be used as standard features in sheet-metal design, for example.

This chapter introduces library features and describes:

- Library feature concepts
- Creating a library feature
- Adding a library feature to a part
- Editing a library feature
- Adding color to a library feature

Library Feature Concepts

- A library feature can contain single-sketch features and fillets. (Features created from a single sketch are bosses, cuts, or simple holes.)
- You cannot insert a library feature that is a base extrusion into a part that already has a base extrusion because you cannot have two base extrusions in a single part. When you create a library feature, first create a base part and then make the boss(es), cut(s), or fillet(s) that you save as the library feature.
- 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 part and the original library feature; if you edit one, the other does not change.
- To add a library feature to an assembly, you must add it to a part in the assembly; it cannot be added to an assembly on its own. You can insert a library feature while editing a part in the context of an assembly.

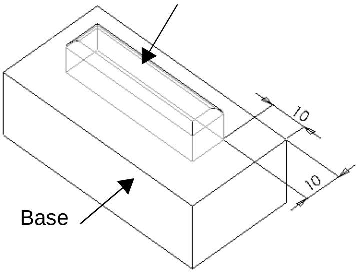
Creating a Library Feature

To create a library feature, you must first create a base feature and then make the bosses, cuts, holes, or fillets that you want as part of the library feature. When you save the library feature, *do not* save that portion that is the base feature.

You save library features with the **.sldlfp** file-type extension.

To create a library feature:

1. Open a new part, sketch a profile, and create a base.
2. Open a new sketch on a face of the base and create a boss or a cut.



3. Continue creating features as needed to complete the library feature.
4. Dimension the library feature to the base part if you want to use dimensions to locate the library feature when you place it on the target part.
5. In the FeatureManager design tree, select the feature(s) you want to save in the library feature. To select more than one feature, hold the **Ctrl** key while you select.
6. Click **File, Save As**. The **Save As** dialog box appears.

7 From the **Save As Type** list, select the library feature type **Lib Feat Part Files** (***.sldlfp**). Enter a name in the **File name** box and click **Save**.

Note: To create a library feature from an existing part, simply select those part features that you want to use in your library feature and save them as a library feature type (***.sldlfp**). Remember that you cannot save the base feature of the part in your new library feature.

Adding a Library Feature to a Part

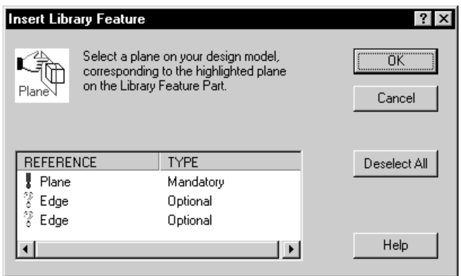
To add a library feature to a part:

1 With the target part open, click **Insert, Library Feature**. The **Insert Library Feature** dialog box appears.

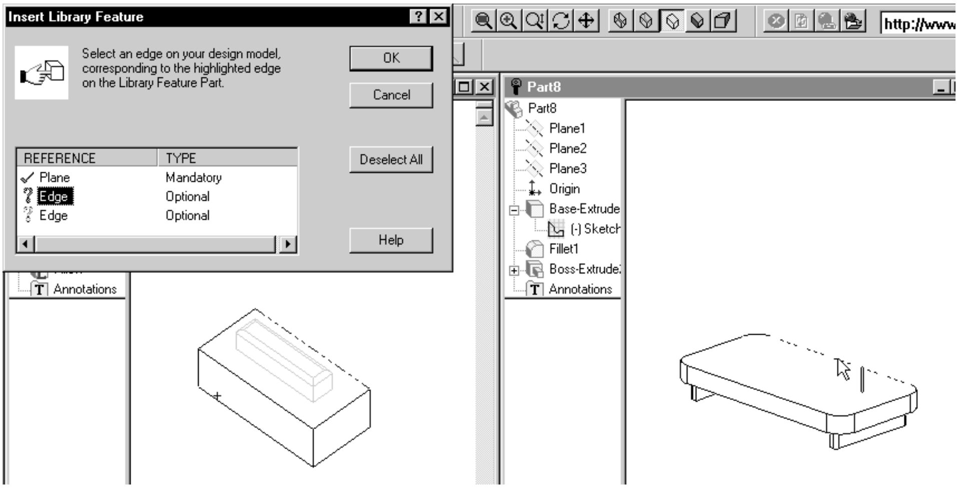
2 Navigate to the directory where the library feature is located, if necessary, and select the library feature file (**.sldlfp*). The library feature's image appears in the **Preview** area of the dialog box if the **Preview** option is checked.

3 Click **Open.** Two windows and a dialog box appear: the library feature window, the target part window, and the **Insert Library Feature** dialog box. (The windows tile automatically.)

In the **Insert Library Feature** dialog, there are mandatory and optional references. 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 red exclamation point in the **Reference** area changes to a check mark.

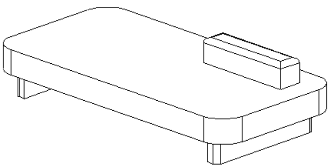


5 Notice that when the reference text, **Edge**, highlights, the corresponding edge becomes a blue dashed line in the library feature window. Select the corresponding edge in the target part window.

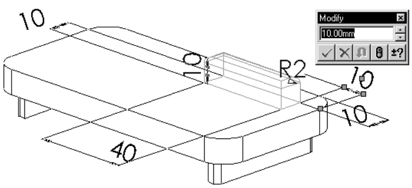
Note: To deselect a selected item, either double-click the checkmark or click **Deselect All**.

6 .Click **OK**.

The library feature is added to the target part.



7 You can drive the library feature to a different location on the target part face by modifying the distance dimensions. Double-click the library feature icon in the FeatureManager design tree to expose the dimensions.



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.

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 and advanced visualization properties to a library feature either before or after you insert it in a target part.

To add color or advanced properties to a library feature:

1. Select any feature in the library feature part that is a part of the actual library feature (not the base).
2. Click **Tools, Options**, and select the **Color** tab.
3. In the **Features** box, scroll to select **Library Feature**.
4. Click the **Edit** button and select a color from the color palette (or create a custom color) and click **OK**.
5. To make the **Advanced** button available so you can add advanced visualization properties, click **Shading** in the **System** box.
6. Click the **Advanced** button and adjust the sliders in the **Material Properties** dialog for **Transparency**, **Shininess**, **Diffusion**, etc.
7. Click **Apply** and **OK**.

To view the changes, make sure your part is in **Shaded** view mode.

A

SolidWorks 97Plus Options

This appendix provides a list of all the options available for customizing part, assembly, and drawing documents in SolidWorks 97Plus. To access these options, click **Tools, Options** and select one of the following tabbed pages:

- ❑ Color
- ❑ Crosshatch
- ❑ Detailing
- ❑ Drawings
- ❑ Edges
- ❑ External References
- ❑ General
- ❑ Grid/Units
- ❑ Import/Export
- ❑ Line Font
- ❑ Material Properties (for parts only)
- ❑ Performance
- ❑ Planes

After you make your selections on an **Options** page, click **OK** to accept the changes; click **Cancel** to discard the changes and exit the dialog; click **Reset All** to return to the installed system defaults.

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.

Reset All. Restores the original default color settings.

Apply To: Determines how the color choices you made will be applied.

- **System Defaults** – all new documents that are created
- **Active Document** – the document that you are currently working on
- **All Possible** – the current document as well as all new documents

Crosshatch Options

Specifies a crosshatch pattern to be used on hatched views, and displays a preview of the pattern selection.

Type

Displays the currently selected pattern.

Properties

- **Pattern.** Displays a list of available crosshatch patterns.
- **Scale.** Specifies the scale use for the pattern.
- **Angle.** Specifies the angle used for the pattern.

Apply To: Determines how the choices you made will be applied.

- **Active Document** – the document that you are currently working on
- **System Defaults** – all new documents that are created
- **All Possible** – the current document as well as all new documents

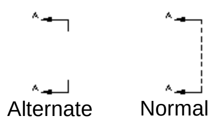
Detailing Options

Lets you set options for detailing and dimensioning in your parts, assemblies, and drawings.

Dimensioning Standard

- Specifies the standard to use: ISO, ANSI, DIN, JIS, or BSI.
- **Sections.** The **Sections** button is only available when you are in a drawing. It displays the **Section Arrows** dialog which lets you specify the **Height**, **Width**, and **Length** of the section arrows used in drawings.

Note: An alternate section arrow display is available if you are using the ANSI standard:



- **Display with Broken Leaders** specifies how dimensions are displayed.
- **Dual Dimensions Display** specifies that two dimension types are used.
- **Display datums per 1982.** Click this checkbox to use the 1982 standard for the display of datums. **Note:** This option is available only if you use the ANSI dimensioning standard.

- **Trailing Zeros.** Select one of three settings:

  Smart – Trailing zeros are trimmed for whole metric values. (Conforms to ANSI and ISO standards.)

  Show – Dimensions have trailing zeros up to the number of decimal places specified in **Tools, Options, Grid/Units**.

  Remove – All trailing zeros are removed.

  Note: Tolerances are not affected by this option.

Arrows

- **Arrows.** Sets arrow size (**Height**, **Width**, and **Length**), and style of dimension arrows (**Open** or **Filled** arrowheads, **Simple Arrow**, **Open** or **Filled** dots, **Slash,** or **None**).
- Set the 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.

Break Lines

- **Break Gap.** Lets you specify the size of the gap between break lines in a broken view in a drawing.

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** and **Extension** of witness lines.

Dimensions

- **Dim Font.** Lets you specify the font type and size used for dimensions.
- **Tolerance.** Specifies the type of tolerance to display: **None**, **Basic**, **Bilateral**, **Limit**, **Symmetric, MIN,** or **MAX.** Specifies tolerance values and indicates whether the dimension is linear or angular.
- **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.
- **Precision.** The **Precision** button lets you set the precision of **Primary Units**, **Angular Units**, and **Alternate Units**.

Notes

- **Note Font.** Lets you specify the font type and size used for notes.
- **Balloons.** Lets you specify 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.
- **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.

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.

Apply To: Select an option from the pull-down list:

- **System Defaults.** Applies your selections to all new documents.
- **Active Document.** Applies your selections to the currently active document.
- **All Possible.** Applies your selections to both new documents and the active document.

Drawings Options

Lets you set 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.

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. Specifies the way models or assemblies are displayed in drawings.

- Select one 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 one 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.

  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 borders around individual drawing views. This is the default.

Automatic scaling of 3 view drawings. When checked, if you insert a **Standard 3 View** drawing, the three views are scaled to fit on the drawing sheet, regardless of the paper size selected.

Detail Item Snapping

- **Infer when dragging corner.** When you click a corner and drag a detail item (for example, a note or dimension), the corner you clicked can infer to the corners of stationary detail items.
- **Infer when dragging center.** When you click the inside of a detail item (a note or dimension) and drag it, the center can infer to the center of stationary detail items, and vice versa.

Detail View Scaling. Specifies the scaling for detail views. The scale is relative to the scale of the original drawing.

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

- **Tangent Edges Visible** – Tangent edges displayed.
- **Tangent Edges With Font** –Tangent edges display as a line using the default font for tangent edges defined in **Tools, Options,** on the **Line Font** page.
- **Tangent Edges Removed** – Tangent edges are not displayed.

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.

Import/Export Options

Specifies the settings for the import and export of SolidWorks 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.

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

- **Version 9 file.** Outputs parasolid transmit file in Version 9 format.
- **Version 8 file.** Outputs parasolid transmit file in Version 8 format.

STL

Output Format

- Select either **Binary** or **ASCII** as the STL file format.

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 or Fine.** 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

- **Deviation.** 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.

ACIS (.sat)

Select either **Version 1.7** or **Version 2.0**. This allows you to specify if a file saved with a **.sat** extension will be in version 1.7 ACIS or 2.0 ACIS format.

DXF/DWG Output

Select either **R12** or **R13**. This allows you to specify if a file saved with a **.dxf** or **.dwg** extension will be in R12 or R13 format.

External References

Specifies how assembly, part, and drawing files with external references are opened and managed.

External References

Open referenced documents with read-only access. Specifies that all referenced documents will be opened for read-only access by default.

Search document folder list for external references. Specifies that the document folder list is searched to update any external references.

Folders

Show folders for: Displays documents of various types.

You can specify the search rules for locating external documents. Folders are searched in the order in which they are listed in the **External References** display box.

- To add a new directory path to search order, click the **Add** button.
- To delete a directory path from the External References display box, select the path, and click the **Delete** button.
- To change the search order, select a directory path listed in the External References display box and click either the **Move Up** button or the **Move Down** button, depending on how you want to change the search order.

Assemblies

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.

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 = Underdefined, Red = Over-defined, Green = Selected. (Overdefined 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**.

Note: Enabling silhouettes may effect 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**.)

Overdefining Dimensions

- **Prompt to set driven state.** When checked, specifies that when you add an overdefining 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 overdefining dimension to a sketch, the dimension is set to be *driven* by default.

Note: The above two checkboxes can be used together or alone, resulting in one of four different behaviors when you add an overdefining dimension to a sketch.

a) A dialog box appears that defaults to *driven*.

b) A dialog box appears that defaults to *driving*.

c) The dimension comes in *driven*.

d) The dimension comes in *driving*.

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. Scroll to 0 (for none) 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.*”
- **Maximize document on open.** When checked, each document opens to its largest size within the SolidWorks window.
- **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.

Grid/Units

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.)

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.

View System Defaults

- When checked, the system default for each of the options is displayed; when not checked the selections used by the active document are displayed.

Apply To: Select an option from the pull-down list:

- **System Defaults.** Applies the selections that you made to all new documents that you create.
- **Active Document.** Applies the selections you made to the active document.
- **All Possible.** Applies the selections that you made to both the currently active document and the system defaults.

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 scroll 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 scroll list, choose a line weight to apply. Choose from **Normal, Thin,** or **Thick** lines. The **Preview** box displays the selected line.

Apply To: Select an option from the pull-down list:

- **Active Document** – the document that you are currently working on
- **System Defaults** – all new documents that are created
- **All Possible** – the current document as well as all new documents

Material Properties

Specifies the material properties for the current part. (This option is available when you are working with an active part document.)

The **Material Properties** page is also used by some Add-In applications. Refer to the instructions of the specific application for further information.

Properties

- **Density.** Lets you specify material properties by selecting the text in the box, and entering the appropriate density specification.

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** or **Fine.** Preset resolutions.
- **Custom.** Allows you to choose any 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.

Windows95 Zooming

- **Enable clipping for zoom limitation.** When using SolidWorks on Windows95, there is a limit beyond which you cannot zoom in on a model. This 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.

Planes Options

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.

B

API Documentation

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 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 (**www.solidworks.com**) under the Technical Support area.

Topics in this Appendix:

- Getting started and installing with C++ and Visual Basic
- Compiling and running your application with C++ and Visual Basic
- Syntax used in the documentation
- Programmer's guide and overview of the OLE automation interface
- Programming topics
- SolidWorks API objects

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

If you wish to compile your project in DEBUG, SolidWorks has to be installed with the **/API** qualifier. To install SolidWorks with the **/API** option, you have to cancel out of the automatic setup utility. Perform the following steps:

1. Insert your SolidWorks CD and click the **Cancel** button on the SolidWorks introductory screen.
2. Select the **Start, Run** from the Windows toolbar.
3. In the dialog box, enter:

   <*cdDrive*>**:setup /API**

   (where <*cdDrive*> is the drive where the SolidWorks CD is loaded.
4. Press **Enter**.

Installing SolidWorks with the **/API** option will update any old DLL’s that are needed at run time.

Note: If you install with the **/API** option, any third-party software (such as PhotoWorks) will not be installed. This is because you will be using the debug Microsoft libraries which are incompatible with the released versions shipped by our third-party vendors. If you install your own copy of third-party software (such as CimLogic's Toolbox/SE) it may have library conflicts and crash.

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 MBCS; Windows NT should use the Unicode settings.
   - If SolidWorks is installed with **/API**, your project should be compiled as DEBUG.
   - If SolidWorks is not installed with **/API**, your project should be compiled as RELEASE.
4. Add your own code.
5. 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 Your C++ Applications

- Running your application as a Release DLL

  If your application was built as a Release DLL, follow these steps:

  1. Start a SolidWorks session.
  2. Select **File, Open** and change your file selection filter to **Add-Ins(*.dll)**.
  3. Select the desired .DLL file and choose **OK**.

     This brings your application into SolidWorks. Any DLL found in the **\APPS** directory of your SolidWorks installation is automatically loaded when SolidWorks is started.

- Running your application in Debug mode

  If you are running your application in Debug mode, then it makes sense to start SolidWorks from your development environment. This allows you to step into your code by setting break points in your application. Be certain that you have installed a SolidWorks debug build. If your application was built as a Debug DLL, 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.
  3. Once SolidWorks is running, select **File, Open** and change your file selection filter to **Add-Ins(*.dll)**.
  4. Select the desired DLL file and choose **OK**. This brings your application into SolidWorks.

In both situations described above, you can also place your DLL in the **\APPS** subdirectory of your SolidWorks installation. Any DLL found in the **\APPS** directory of your SolidWorks installation is automatically loaded when SolidWorks is started.

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 Select **Tools, Macro, Run**.

  2 Choose the desired BASIC source file and select **OK**.

- 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.

Syntax Used in the Documentation

In this appendix and in the online documentation, each API function is shown using C++ syntax. The syntax used to describe each API function is as follows:

```
ReturnValue Object::Function(Parameters)
```

All SolidWorks API functions support the COM interface. The API Help documentation will only show the COM syntax if the argument or return types are different from the Dispatch syntax (see below). 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.

For example, the following syntax shows the function `SelectByID` which requires a `ModelDoc` object and will return a boolean value. It also has five arguments which are passed into the function.

Syntax

boolean ModelDoc::SelectByID (BSTR *selID*, **BSTR** *selType*, **double** *x*, **double** *y*, **double** *z***)**

BASIC Usage

```
result = ModelObj.SelectByID("Point1","SKETCHPOINT", .2, .3, 0)
```

C++ Dispatch Usage

```
result = ModelObj.SelectByID(_T("Point1"),_T("SKETCHPOINT"), .2, .3, 0);
```

C++ COM Usage

```
hres = ModelObj->SelectByID(_T("Point1"),_T("SKETCHPOINT"),.2,.3,
0,&result);
```

The next example shows the function `InsertSketch` which returns nothing `(void)` and accepts no arguments. This method will simply insert a sketch into the current document (`ModelDoc`).

Syntax

void ModelDoc::InsertSketch( )

BASIC Usage

```
ModelObj.InsertSketch
```

C++ Dispach Usage

```
ModelObj.InsertSketch();
```

C++ COM Usage

```
hres = ModelObj -> InsertSketch();
```

The following syntax shows that the function `GetType` also requires a `ModelDoc` object and returns a *long* value. This function takes no arguments. Instead, it simply uses the current `ModelDoc` object and returns its type.

Syntax

long ModelDoc::GetType( )

BASIC Usage

```
docType = ModelObj.GetType
```

C++ Dispatch Usage

```
docType = ModelObj.GetType();
```

C++ COM Usage

```
hres = ModelObj ->GetType( &docType );
```

Likewise, the following example will get the number of edges from the current face object and returns that value as a *long* to the calling routine.

Syntax

long Face::GetEdgeCount( )

BASIC Usage

```
edgeCount = FaceObj.GetEdgeCount
```

C++ Dispatch Usage

```
edgeCount = FaceObj.GetEdgeCount();
```

C++ COM Usage

```
hres = FaceObj ->GetEdgeCount( &edgeCount );
```

You will also notice that many functions have several different syntax types which are used under varying circumstances. A function will have multiple interfaces only when required.

Syntax

IDispatch *Body::GetFirstFace( )(OLE Automation)

HRESULT Body::IGetFirstFace(LPFACE* retval)(COM Object)

Basic Usage

```
Set FaceObj = BodyObj.GetFirstFace
```

C++ Dispatch Usage

```
FaceObj = BodyObj.GetFirstFace();
```

C++ COM Usage

```
hres = BodyObj->IGetFirstFace( &FaceObj );
```

Notice the Dispatch interface (labeled as `OLE Automation`) returns a dispatch pointer, while the COM interface returns an `LPFACE` pointer. The COM interface will use Object pointers instead of Dispatch pointers and pointers instead of Variant Safearrays. Since the argument types are different, any API function that handles Objects or Arrays will have two distinct interfaces, one for COM and one for Dispatch (OLE). In the example shown above, the Dispatch and COM interfaces are `GetFirstFace` and `IGetFirstFace`, respectively.

Here is another example with two distinct interfaces. Notice that this time we are handling an array.

Syntax

VARIANT ModelDoc::GetMassProperties( ) (OLE Automation)

HRESULT ModelDoc::IGetMassProperties(double* retval)(COM & DLL object only)

Basic Usage

```
massProps  = ModelObj.GetMassProperties
```

C++ Dispatch Usage

```
massProps  = ModelObj.GetMassProperties();
```

C++ COM Usage

```
hres = ModelObj->GetMassProperties( &massProps );
```

When an API function has more than one interface, the different syntax types will contain one of the notes described below:

(Basic Language Syntax Only)

This note describes the syntax used for Basic programmers when there is a difference between Basic syntax and C/C++ syntax. In general, you will see this note used with Properties where C/C++ must include a `Get` or `Set` at the beginning of the function name.

(C/C++ Syntax)

This note describes the syntax used for C/C++ programmers when there is a difference between Basic syntax and C/C++ syntax. In general, you will see this note used with Properties where C/C++ must include a `Get` or `Set` at the beginning of the function name.

(OLE Automation)

This note describes the syntax for a Dispatch style interface. The Dispatch interface will pass Objects as Dispatch pointers and handle passing arrays by packaging them up into Variant SafeArrays so that they can be understood by languages such as Basic.

(COM Object)

This note describes the syntax for a COM interface.This function interface is typically trying to pass an Object. The COM interface allows the Object to be passed as pointer to that object instead of as a Dispatch pointer.

(COM and DLL Object Only)

This note describes the syntax for a COM interface which can only be used from a DLL. This function interface is typically using a pointer to pass or return array data. This type of variable passing can only be used from a DLL since you cannot read or write to memory which is allocated in a different .exe process. This limitation excludes the use of dispatch pointers, interface pointers, and pointers to Safearrays which can be used from an .exe or DLL.

For example, the `GetFirstFace` method returns an interface pointer. Because of this, we have two different interfaces for the `GetFirstFace` method. The syntax in the documentation appears as follows:

```
IDispatch *Body::GetFirstFace()        (OLE Automation)
HRESULT Body::IGetFirstFace(LPFACE*retval) (COM Object)
```

Notice the dispatch style interface (labeled with the “OLE Automation” note) returns a dispatch pointer, while the COM implementation does not. All COM implementations have an `HRESULT` return value and any other values to be returned are passed in as arguments. In this example, the COM implementation requires that you pass in a pointer to an `LPFACE` object. The pointer to `LPFACE` allows SolidWorks to fill in the `retval` variable and return it to you with the `Face` object.

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-5.

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.

Any licensed user of SolidWorks is free to use any or all of these samples in connection with building applications related to SolidWorks, and is granted a royalty free, non-exclusive license for these samples, or parts thereof. Intellectual property rights of the samples remain with SolidWorks. Any confidentiality provisions of the SolidWorks license apply to the samples.

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-16). 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;
```

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 **Syntax Used in the Documentation** on page B-7 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 Varient return values, see **Helpful Hints** on page B-14.

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.

API Object Descriptions

Object	Object Description
AssemblyDoc	Perform assembly functionality: add new components, add mate conditions, hide, and explode components.
Attribute	From this attribute instance you can get: the attribute definition, the associated entity, the parameter values, or the instance name.
AttributeDef	Attribute definition containing default values, allows you to create instances of this definition on entities in your model.
Body	Allows access to the faces on a body and the ability to create surfaces for sewing into a body object.
CoEdge	This “edge with a defined direction” allows access to the underlying edge and loop as well as various CoEdge data.
Curve	Allows access to the curve type and the curve parameters in their native form or in terms of B-curve data.
DatumTag	Allows access to display information for datum tags.
DatumTargetSym	Allows access to display information for datum target symbols.
Dimension	Allows you to get and set dimension values.
DrawingDoc	Perform drawing operations: create, align, and access views; create dimensions, notes, compound notes, and so on.
Edge	Allows access to its defining CoEdge and adjacent faces, and its underlying curve and vertices; as well as edge data.
Entity	Allows you to get an attribute instance that was stored on an entity.
EnumBodies	Manipulate an enumerated list of bodies.
EnumCoEdges	Manipulate an enumerated list of CoEdges.
EnumEdges	Manipulate an enumerated list of edges.
EnumFaces	Manipulate an enumerated list of faces.

EnumLoops	Manipulate an enumerated list of loops.
Environment	Allows you to analyze the text and geometry used to create a symbol.
Face	Allows access to the underlying edge, loop, and surface; to the owning body or feature; and to face tessellation, trim data, and so on.
FeatMgrView	Allows you to access your own FeatureManager design tree object.
Feature	Allows access to the feature type, name, parameter data, and to the next feature in the FeatureManager design tree.
Frame	Allows you to modify, check, and add to the SolidWorks drop-down and pop-up menus.
Gtol	Allows you to get and set geometric tolerancing parameters.
Loop	Allows access to the owning face and to the list of edges and CoEdges contained in the loop.
Mate	Allows access to various assembly mate parameters.
MateEntity	Allows access to mated objects and the assembly mate definition.
Member	Allows access to the next assembly member, the member type, name, the member's ModelDoc, and the ability to get and set its transform.
MidSurface	Allows access to mid-surface information.
ModelDoc	Dimension solids, view operations, sketch operations, set parameters, select objects, save, create and edit features, create wireframe.
Modeler	Provides an interface for the management of temporary body objects.
ModelView	Allows you to get and set the model view's orientation, scale, and translation; as well as the Microsoft handle to the window.
Note	Allows you to get standard note information and to create geometry and text inside a compound note.
Parameter	Allows you to get and set values in an attribute.

PartDoc	Allows you to create bodies and features, perform suppress operations; and to get part extents, part tessellation, entities by name, and so on.
RefAxis	Allows access to reference axis definitions.
RefPlane	Allows access to reference plane definitions.
SelectionMgr	Allows you to get information about selected objects and to get your selection coordinates interpreted in model or sketch space.
SFSymbol	Allows access to display information for surface finish symbols.
Sheet	Allows access to get and set sheet information, and to access objects on the sheet such as Bill of Material tables.
Sketch	Allows you to get information about sketch elements and the sketch orientation.
SldWorks	Create, open, close, and quit documents, arrange icons and windows, change the active document, create an attribute definition, and so on.
Surface	Allows you to get the surface type and various data, as well as evaluate and reverse evaluate locations on the surface.
Vertex	Allows you to get the associated edges in an enumerated list and to return the coordinates of the vertex.
View	Allows you to get information about all the objects on a drawing sheet or drawing view, as well as the drawing view bounds, xform, and so on.
WeldSymbol	Allows access to display information for weld symbols.

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