SolidWorks Works

SolidWorks® 98

User's Guide

Document Number: UG0198-E

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SolidWorks Fundamentals

What is SolidWorks 98?

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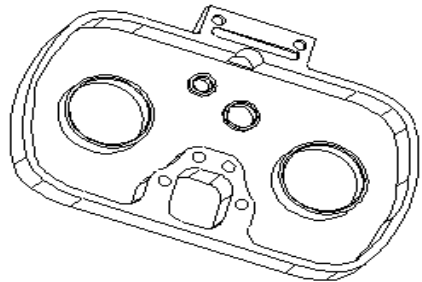
This chapter discusses some basic concepts and terminology used throughout the
SolidWorks 98 application. It provides an overview of the following topics:

❑ Installing the SolidWorks 98 software

❑ SolidWorks terms and document windows

❑ FeatureManager™ design tree

❑ Finding out what is wrong

❑ Right-mouse menus

❑ Selecting items

❑ Time saving tips

❑ Preparation for printing

❑ Rendering models with PhotoWorks

❑ Using the SolidWorks API development environment

Installing the SolidWorks 98 Software

System Requirements

❑ Microsoft Windows NT 4.0 or Windows® 95

❑ Pentium®-based computer, or DEC AlphaStation™

❑ 48 MB RAM minimum, 64 MB or more recommended

❑ Mouse or other pointing device

❑ CD-ROM drive

Required Information for Installation

The SolidWorks 98 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 98 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 on the back of the CD-ROM case.

❑ **SolidWorks Registration Code.** You will receive your registration code when you register your software. During the SolidWorks 98 installation, you have an opportunity to e-mail a request for your registration code. This is the quickest way to receive a registration code. (You can proceed with the installation and use the SolidWorks 98 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 to answer.

The Client/Server Installation

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

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 98**.

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.

Registry Copy Tool

The SolidWorks 98 software lets you select options to customize the product to best suit your needs. (For a complete discussion of the options and how to set them, refer to *Appendix A* in this manual.) Also, you can use the **Copy Options Wizard** to copy the options information set by one user and specify the same options on the machines of other users.

To use the Copy Options Wizard:

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

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

3 Specify the destination for the options information:

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

- or -

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

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.

Section A-A

MainSheet

❑ **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, dimension precision, and tolerances; detailing options such as arrow, witness line, balloon style or virtual sharp representation; font size and style; text placement, and so on
Drawings	Drawing sheet size, scale, projection type, model edge display, automatic update of BOM and model views, and inferencing behavior when dragging detail items
Edges	Display of hidden edges types, tangent edge display, repainting and highlighting behavior
Export	Settings for IGES, Parasolid, STL, DXF/DWG, ACIS and TIFF; trim curve accuracy; optional output as trimmed surfaces or 3D curves
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, number of backup copies per document, AutoRecover, and so on.
Grid/Units	Grid display, spacing, and snap behavior; unit types (inches, feet, millimeters, or centimeters); spin box increments
Import	Default solutions for problems that may occur when importing IGES files
Line Font	Style, and weight of edge lines for selected edge types
Material Properties	Material properties for the current part
Performance	Display quality for shaded, wireframe, and transparent models
Reference Geometry	Define the display filter for reference geometry, and specify the default names for planes in part or assembly documents

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

The **AutoRecover** option automatically saves information about your active part, assembly, or drawing document so you will not lose your work in case of a system crash. To set this option, click **Tools, Options, General** and select **Save AutoRecover info every (*n*) changes**. Use the scroll arrows to specify the number of changes that should occur before information is automatically saved. See *Appendix A*, **General Options** on page A-12.

❑ **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**.
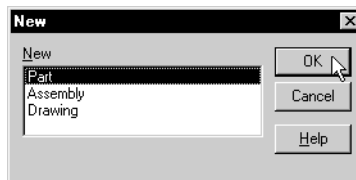
Opening an Existing SolidWorks Document

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 5-14 for more information.

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

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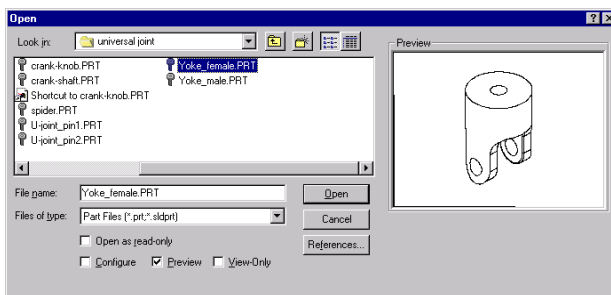
1 In this manual, "Click..." means point the cursor and press the *left* mouse button

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.

Multi-Threaded Retrieval

Documents saved in SolidWorks98 are retrieved in multi-threaded mode.

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

Opening an Existing Part from Explorer

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

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

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

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

 Quick View displays the part in a **SolidWorks Viewer** window.
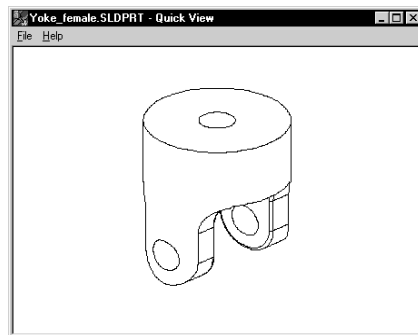
To open the part, drawing, or assembly from Explorer, do one of the following:

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

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

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

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

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

SolidWorks Terms

This section illustrates some common SolidWorks terms.

Menu bar

Toolbars

Part document window

Toolbars

Drawing document window

Status bar

Graphics area

Model

FeatureManager design tree

Vertex

Edge

Axis

Origin

Plane2

Axis

Plane

Axis1

Face
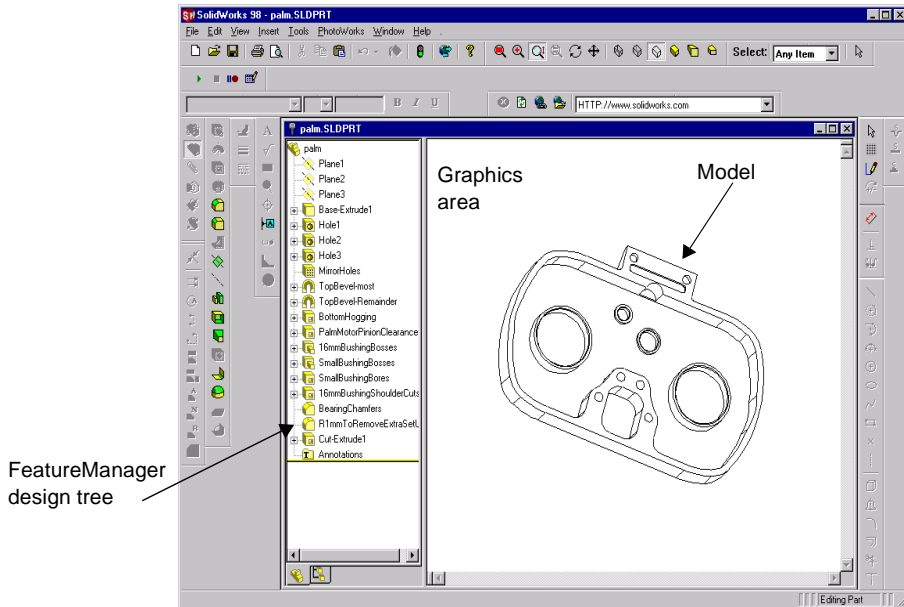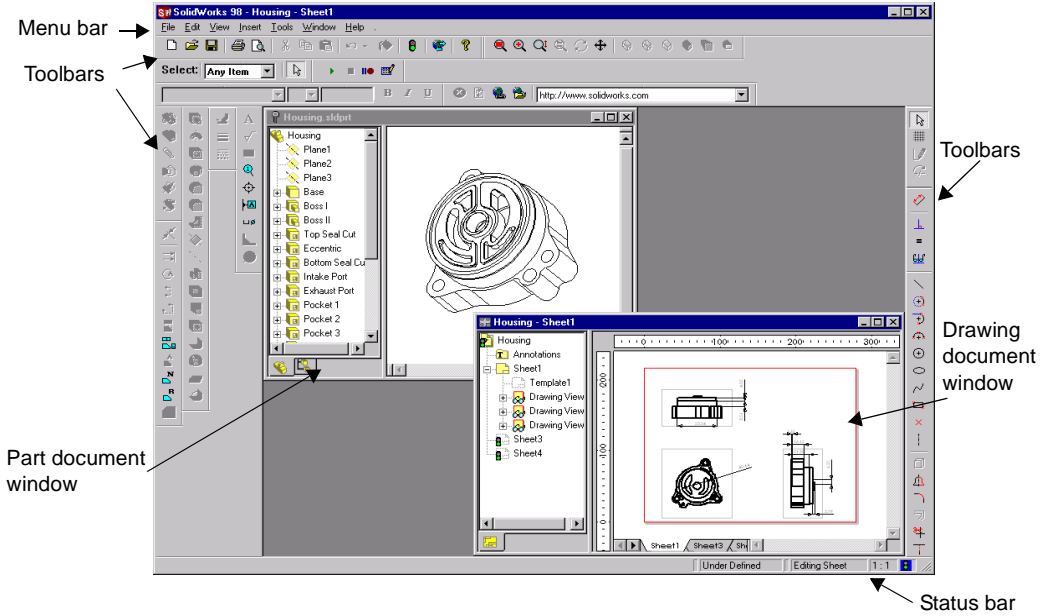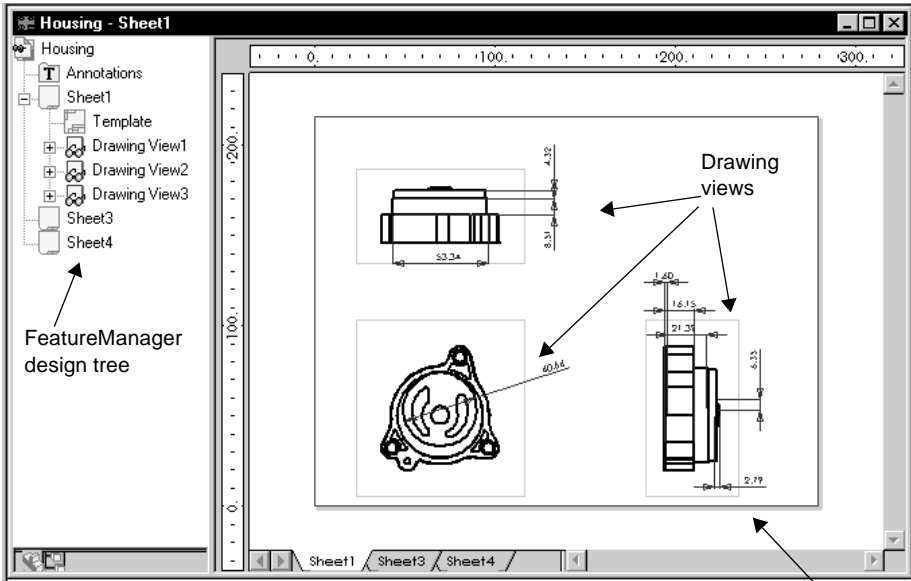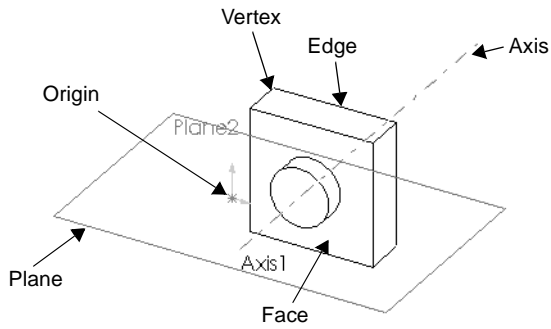
FeatureManager
design tree

Drawing
views

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.
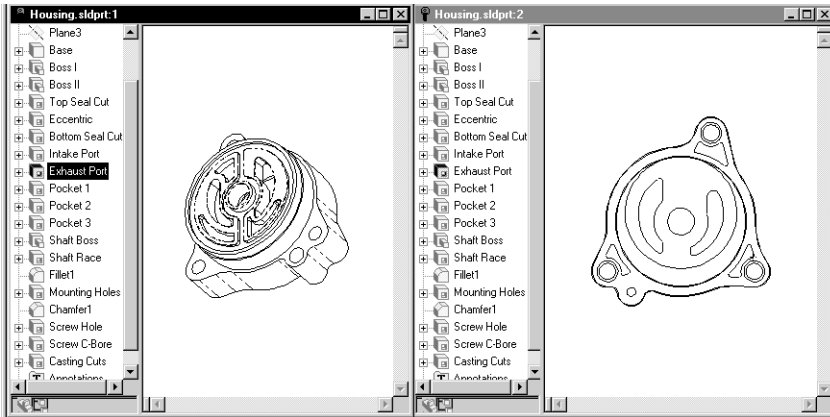
❑ Drag the horizontal and/or vertical **Split** controls to split the window into two or four panes. You can zoom, rotate and set the view mode for each of these views independently.

Window size control

Horizontal Split control

Vertical Split control
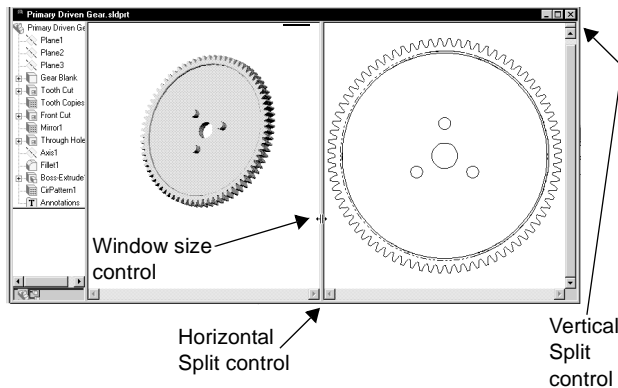
To move from one pane to the next, press the **Tab** key; to move to the previous pane press **Shift-Tab**. You can also press the **F6** key to move to the next pane, or **Shift-F6** to move to the previous pane.

Selecting an item in one view selects it in all views. For instance, when creating a fillet you could select edges on the front of the model in one view and edges on the back in another view.
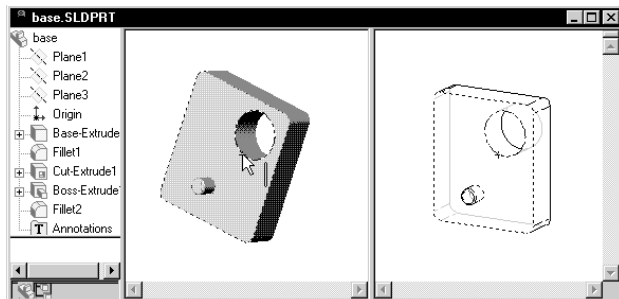
Using Named Views

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.
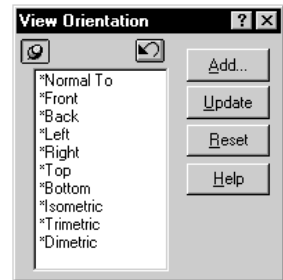
❑ Because the **View Orientation** dialog box is a useful tool, you may want to keep it available while you work. Drag the dialog box to a convenient location on your screen and click the *push pin* icon. This keeps the list open and on top of all other windows.

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.

   - or -

   To return your model to the previous view, click **View, Modify, Undo View Change**. (This can undo a single view change.)

❑ 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 the structure of the part, assembly, or drawing.

- A *Graphics Area*, where you create and manipulate the part, assembly, or drawing.
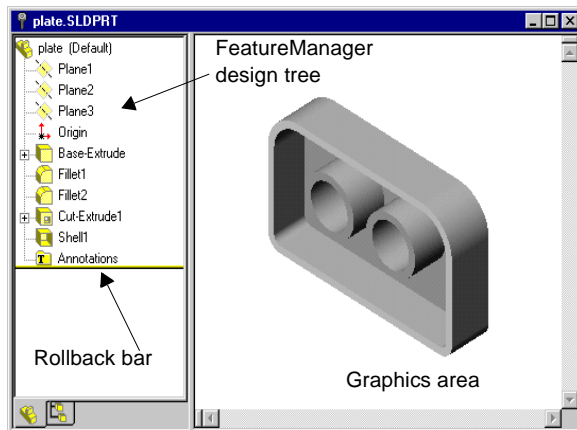
❑ The FeatureManager design tree makes it easy to:

- Select items in the model by name.

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

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

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

  Note: Feature names are not case sensitive: "Boss1" and "boss1" are the same.

- Suppress or hide selected features.

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

- Add a new equation, edit, or delete an equation by right-mouse clicking the **Equations** folder **E** , and selecting the action you want. See **Equations** on page 5-7.

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

- Control the display of dimensions and annotations by right-mouse clicking the Annotations folder **T** and checking the options. See **Displaying Annotations in Parts** on page 5-20.

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 2-35. (There is *no* prefix if the sketch is fully defined.)

- Assembly components in the FeatureManager design tree are preceded by **(+)** if their position is 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.

FeatureManager Design Tree Options

Select options for the FeatureManager design tree by clicking **Tools, Options** and selecting the **General** tab. (For more information, see **General Options** on page A-12.) Select from the following options:

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

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

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

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

What's Wrong?

The SolidWorks 98 application offers a "What's Wrong" functionality, that allows you to view any regeneration errors of a part or assembly. Down arrows appear 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
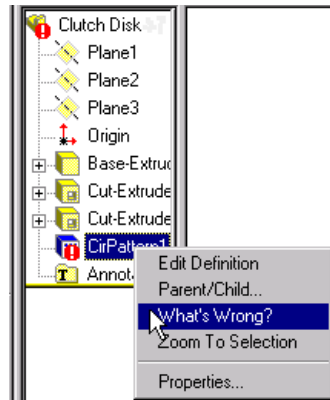
You can turn off the automatic display of errors by checking the **Display Errors at Every Rebuild** checkbox in the **Rebuild Errors** dialog box. The **Rebuild Errors** dialog box displays when the error is first generated, or you can display the dialog box by right-mouse clicking on the part in the FeatureManager design tree.

Note: The **Rebuild Errors** dialog box only affects the current session.
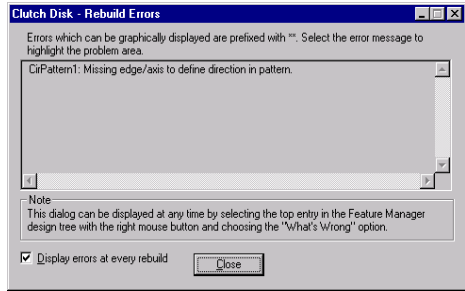
Regenerate Symbol

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. Look for the rebuild icon at the right-hand end of the Status Bar, also.

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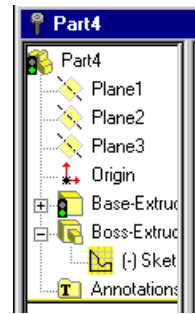
Drag and Drop

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**.
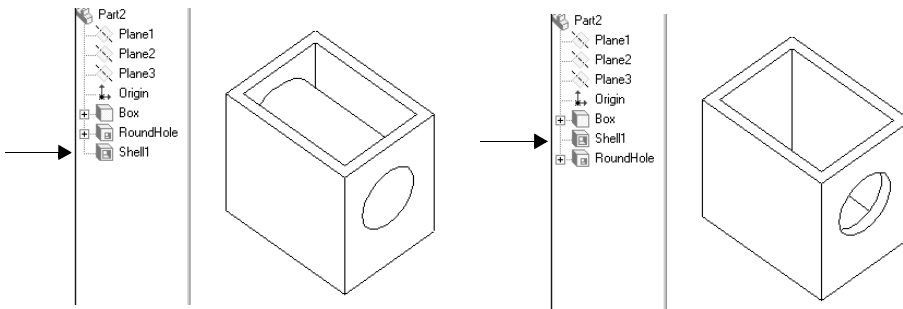
❑ **Moving and copying features.** You can also move features by dragging them in the model. For instance, you can move a hole to a different face.

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.

For more informations about moving and copying features, see **Feature Handles** on page 5-6 and **Copying Features** on page 5-7.

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

Right-Mouse Menus

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

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

For instance, using the right mouse button, you can

- Select a tool without moving your cursor to the toolbar

- Open and close sketches

- Change or view the properties of an item

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

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

- Open an assembly component for editing

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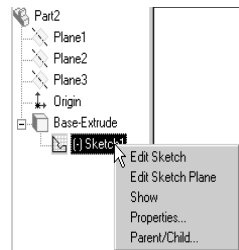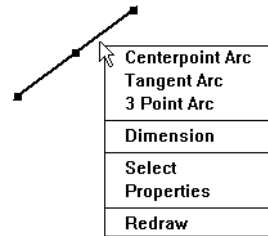
Selection Methods

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 [cursor] button (the selection tool), then click the item you want to select.

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

Note: For information about turning dynamic highlighting off or on, see **Edges Options** on page A-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 help you select items, the cursor symbol changes to indicate the kind of item that the cursor is currently pointing to. When you pass the cursor over a face, the cursor symbol is shaped like a flag; when it is over an edge, the symbol is a vertical bar. A *midpoint cursor* displays when you are over the middle of an edge.
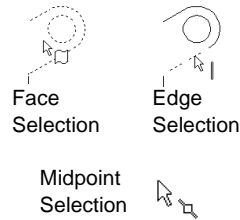
❑ **Selecting midpoint of an entity.** For easy selection of the midpoint of an edge, press the right mouse button over the edge and click **Select Midpoint**. A selection point appears in the middle of the entity.

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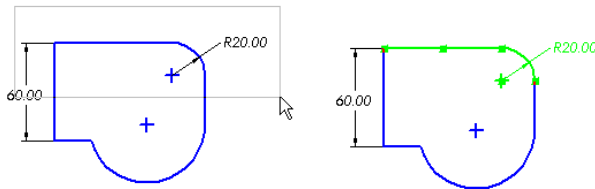
❑ **Selecting loops.** If you select a face, all *loops* (closed collections of connecting edges) on the face are selected, too. To select individual loops, click on the face, then hold down the **Ctrl** key and click on the loops you want to select. (This will deselect the face and leave the loops selected.)

Select face

Ctrl-click loop

Only selected loop is filleted
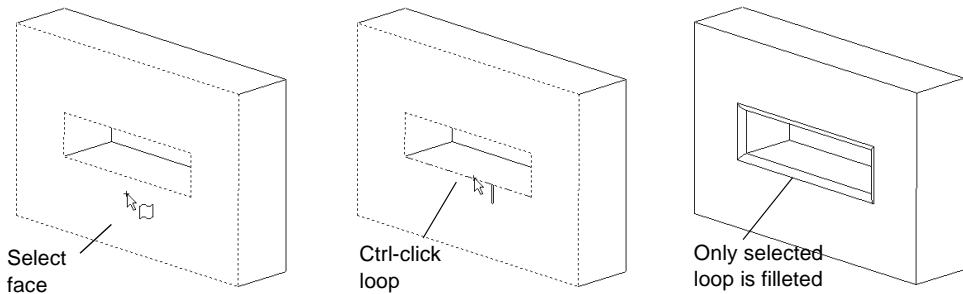
❑ **Changing a selection.** Some dialog boxes show you a list of the selected items. You can change the selection while the dialog is open by clicking or **Ctrl**-clicking in the model. To start over, right-mouse click the model and choose **Clear Selections**.

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.
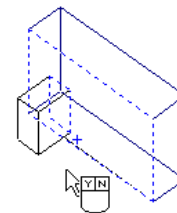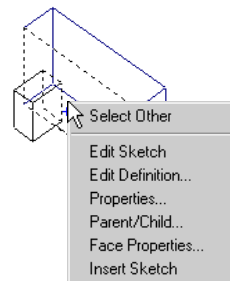
Selecting from the FeatureManager Design Tree

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.

Feature Selected

Sketch Selected

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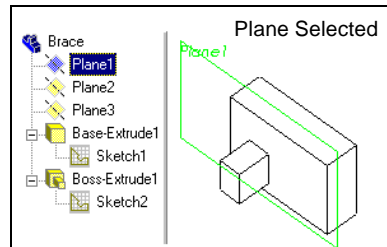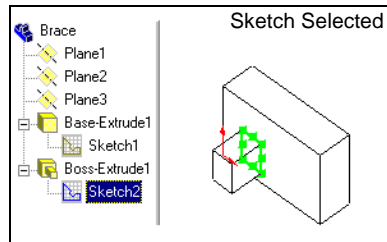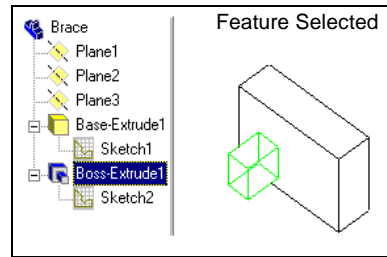
❑ You can select multiple consecutive items in
   the FeatureManager design tree by holding
   the **Shift** key while you select.

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

❑ You can select 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.
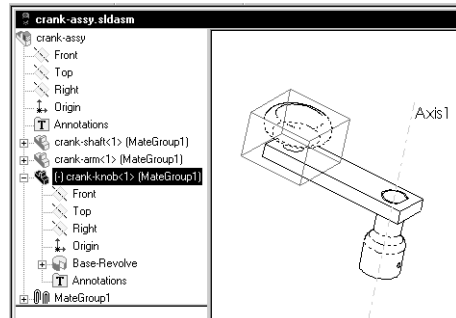
Toolbars

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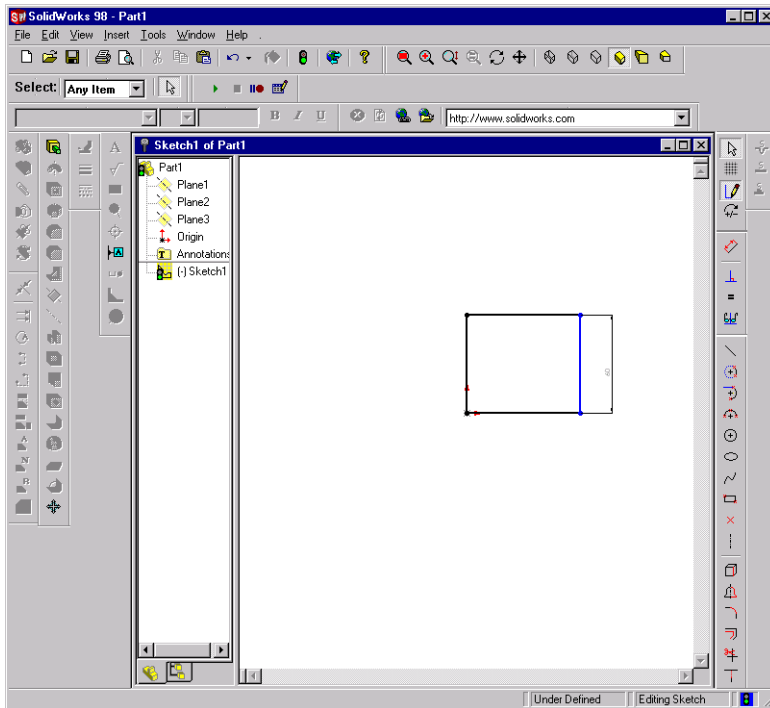
To display or hide individual toolbars or the selection filter,

- click **View, Toolbars**, or

- right-mouse click the SolidWorks window frame.

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

The last item in the toolbar list is **Customize**. Click **Customize** to move toolbar icons to different toolbars, change menus, or reset shortcut keys. See **Customize and Arrange Toolbars** on page 1-28.

Customize and Arrange Toolbars

To customize toolbars:

Click **Tools**, **Customize**, or

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

❑ On the **Toolbars** page, you can

- select individual toolbars to display or hide

- display the toolbar icons in a larger size

- display or hide the tooltips that pop up when the cursor moves over an icon

❑ On the **Commands** page, you can

- rearrange the command buttons on toolbars

- move buttons from one toolbar to another

- duplicate buttons and place them on more than one toolbar

- delete buttons that you never use

❑ On the **Menus** page, you can

- customize the SolidWorks menus to suit your style of working

- rename, relocate, or remove commands from the menus

❑ On the **Keyboard** page, you can

- customize the keyboard shortcut keys to suit your style of working

- assign new shortcut keys, remove shortcut keys, or reset the shortcut keys to their original setting.

To arrange toolbars:

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

If you drag a toolbar away from the window edge, it becomes a floating palette.
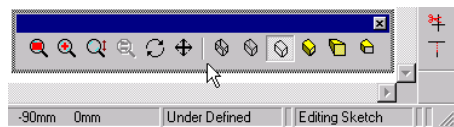
The View Toolbar

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.
	Zoom to Selection	Zooms in on a portion of the model, assembly, or drawing that you select. Click an entity to which you want to zoom, or hold the **Ctrl** key and click several entities. To zoom back out, select **Zoom to Fit**.
	Rotate View	Dynamically turns the part or assembly image around a view center as you move the mouse. (Not for drawings.) • *To rotate the image on a vertical axis*: Move the mouse left to right. • *To rotate the image on a horizontal axis*: Move the mouse up and down. • *To rotate the image diagonally*: Move the mouse diagonally. • *To rotate the image about a specific point, line, or plane:* Select the **Rotate** icon, select the entity to rotate about, use the mouse to rotate, as usual. You can also use the keyboard arrow keys to rotate the image.
Note: To change the speed of the view rotation, click **Tools, Options**, and select the **General** tab. Use the slider to change the speed of rotation with the mouse; you can change the angle increment used when rotating with the arrow keys.		

	Pan	Dynamically moves the image. Press and hold the left mouse button while moving the cursor around on the screen.
	Wireframe	Displays all the edges of the part or assembly.
	Hidden in Gray	Displays edges that are obscured from view in a light gray line. Visible edges are displayed normally.
	Hidden Lines Removed	Displays only those lines that are visible at the angle the model is rotated. Obscured lines are removed.
Note: To change the default display color of lines in the above three view types, click **Tools, Options, Color**. Select the line type, click **Edit**, and select a color from the color palette. Click **OK**.		
	Shaded	Displays a shaded image of the model.
Note: To change the default color of the shaded part, click **Tools, Options, Color**. Select **Shading**, click **Edit**, and select a color from the color palette. Click **OK**. You can also change the color, intensity and direction of the light that strikes the surface of the shaded part. Click **View, Lighting** and choose from the options available for **Ambient**, **Directional**, and **Spot Light.**		
	Perspective	Displays a perspective view of the model. A *perspective* view is the most normal view as seen by the eye or a camera. Parallel lines recede into the distance to a vanishing point. Perspective may be used in combination with any of the view modes.
	Section View	Creates a section view of the model or assembly for display purposes. You cut through the model or assembly using one of the following: • one or more standard planes (Plane1, 2, or 3). • one or more planes that you create for the purpose (Click **Insert, Reference Geometry, Plane**, and create a new plane.) See **Creating a Construction Plane** on page 3-2. • one or more planar face(s) on the model. • the viewing plane (The invisible plane that is parallel to the screen you are looking through. The initial position for this plane is the center of the part body bounding box.) See **Section View of a Model** on page 5-21.

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 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.
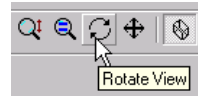
❑ **Use spin box arrows to change values.** If a value box has arrows next to it, you can increase or decrease the value by clicking on the arrows. (You can change the spin box increment value with the **Tools**, **Options** command. Click the **Grid/Units** tab.)
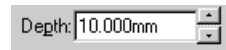
❑ **Modify dimensions with the spin box.** You can increase or decrease dimension values by clicking on the arrows. In addition, you can use the row of buttons below the value window to do any of the following:
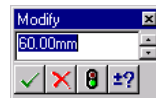
  ✓ Accept the current value and exit.

  ✗ 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.
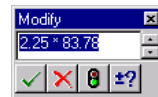
❑ **Check the status bar for prompts.** The *status bar* at the bottom of the SolidWorks window describes commands as you scroll through the menus or point to buttons on toolbars. It also provides prompts for some commands that require multiple steps.
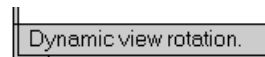
> **Note:** You can hide or display the status bar. Click **View**, **Status Bar**. A checkmark next to the menu item means the status bar is visible.

Keyboard Shortcuts

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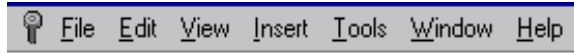
Press the **Alt** key and the underlined letter to display the menu. For example, press **Alt** + **F** to pull down the **File** menu.

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.
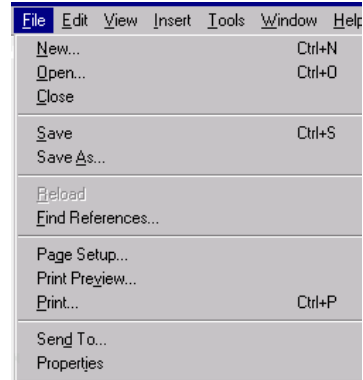
Some commands also have accelerator keys that are displayed on the menu next to the command. For example, the combination **Ctrl** + **N** opens a new file.

Note: You can customize the keyboard shortcut keys to suit your style of working.

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**

Preparation for Printing

Page Setup

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

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

To set print options:

1 Click **File, Page Setup**.

2 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.
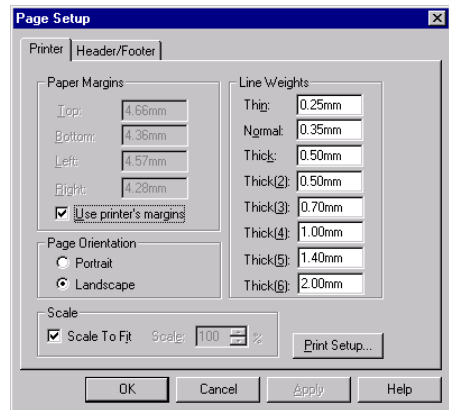
To create a custom header and/or footer for a document:

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**.
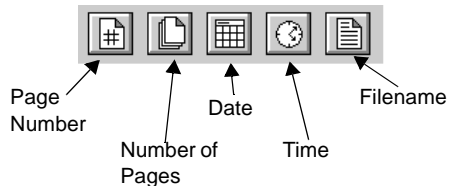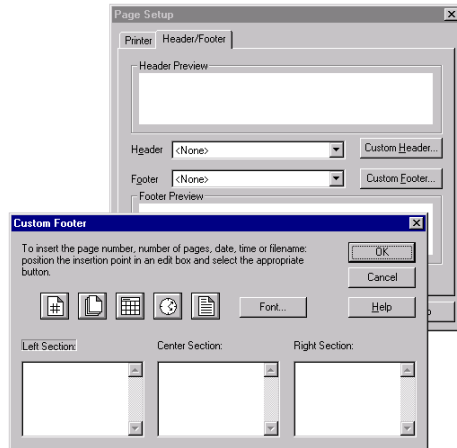
Rendering Models with the PhotoWorks Application

PhotoWorks is a photo-realistic rendering application, available through the SolidWorks 98 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 98*, Chapter 15.

SolidWorks Application Programming Interface

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

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

Sketching with SolidWorks

Most SolidWorks features begin with a sketch. Becoming comfortable with using the sketch tools is, therefore, very important. The discussion of sketching in this chapter is enough to get you started, but you should also do the "40-Minute Running Start" in the *Learning to Use SolidWorks 98* 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

❑ Selecting a starting plane for a sketch

❑ Deciding how complex the sketch should be

Starting a New Sketch

To open a new sketch, you can either

- Click the **Sketch** tool , or

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

A new sketch opens on Plane1 (the default plane) and activates the three sketch toolbars: *Sketch*, *Sketch Tools*, and *Sketch Relations*. These toolbars and tools are described on the next few pages.
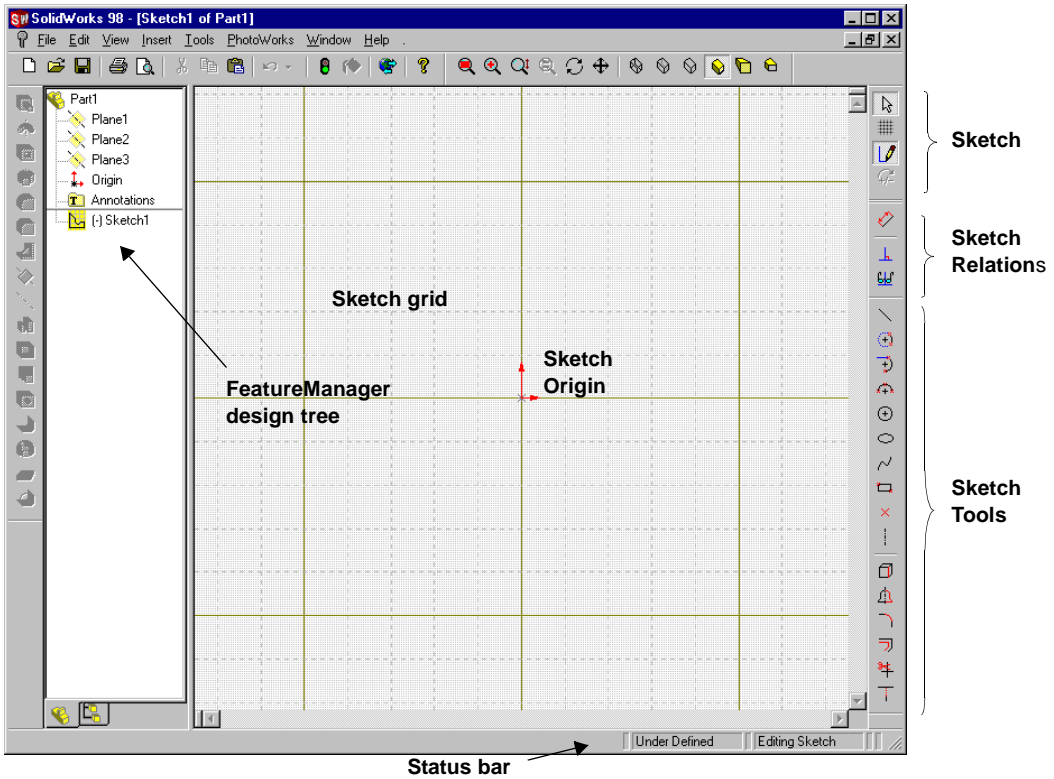
For information about options that you can select to customize the sketching functionality, refer to *Appendix A*, **General Options** on page A-12.

The Sketch Toolbar

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

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

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

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

Grid gives you access to grid options such as grid visibility, grid spacing, or snap behavior. Use the **Grid** button, or click **Tools, Options**, and select the **Grid/Units** tab. See **The Sketch Grid** on page 2-38.

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 2-18.

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 2-11.

Construction Geometry converts sketched entities (lines, arcs, splines, ellipses, etc.) in a drawing to construction geometry (menu item for drawings only). See **2D Sketching** on page 8-36.

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

To use sketch entities as construction geometry, see **Converting Sketch Lines to Construction Geometry** on page 3-6.

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 trims a sketch element.

Extend extends a sketch element.

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

Automatic Relations automatically creates relations as you add sketch entities.

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

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

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

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

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

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

For descriptions of using the Sketch Tools, see **Using the Sketch Tools** beginning on page 2-13.

The Sketch Relations Toolbar

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

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

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

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

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

Display/Delete Relations displays relations that were either manually or automatically applied to sketch elements, and lets you delete relations that you no longer want.

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

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

Using the Sketch Entity Tools

This section discusses how to create and use sketch entities.

Line

To sketch a line:

1 Click the **Line** tool.

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

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

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

To change the length of the line:

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

2 Drag to lengthen or shorten the line.

To move the line:

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

2 Drag the line to another position.

To change the angle of a diagonal line:

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

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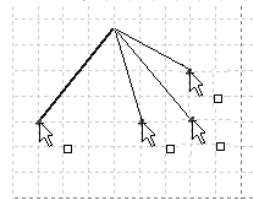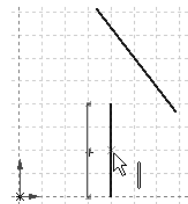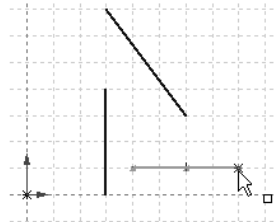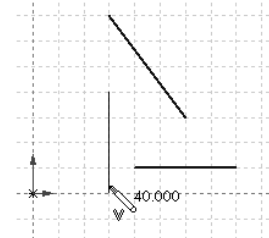
Rectangle

To create a rectangle:

1 Click the **Rectangle** tool.

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

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

While sketching the rectangle, its dimensions are dynamically displayed.

Circle

To sketch a circle:

1   Click the **Circle** tool.

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

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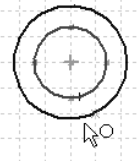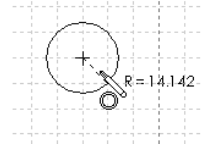
To move the circle:

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

To change the radius:

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

Arcs

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

1   Click the **Centerpoint Arc** tool.

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

3   Release the mouse button. The circumference guideline remains.

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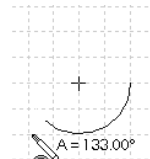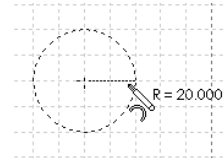
To sketch an arc, tangent to a sketch element:

1   Click the **Tangent Arc** tool.

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

3   Drag out the arc to the desired shape.

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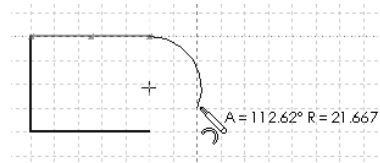
To sketch an arc through three points:

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

2 Point where you want the arc to start.

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

4 Release the mouse button.

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

6 Release the mouse button.
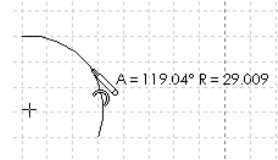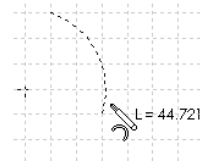
Ellipses

To create an ellipse:

1 Click the **Ellipse** tool.

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

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

4 Release the mouse button.

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

To create a centerpoint ellipse:

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

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

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

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

5 Release the mouse button. The circumference guideline remains.

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

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 and add relations or dimensions:

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

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

You can:

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

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

- add relations to spline points

- drag the endpoints to reshape the spline

Note: A newly created spline is *non-proportional* by default; you can dimension and add relations to points of a non-proportional spline. You can change its shape by dragging a spline point or endpoint.
A *proportional* spline retains its shape when you drag the endpoints; the entire spline resizes proportionally. You cannot dimension or add relations to its points.
To change the proportional status of a spline, right-mouse click the spline and select **Properties**. Click to check or uncheck the **Proportional** checkbox.

To delete spline points:

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

Point

To create a sketch point (reference point):

1   Click the **Point** tool. ☒

2   Click where you want to create the sketch point.

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

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

2   Click the **Point** tool. ☒

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

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

Centerline

To create a centerline:

1 Click the **Centerline** tool.

2 Point where you want the line to start.

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

4 Release the mouse button.

Text

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

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

To insert text on a model face:

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

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

The cursor becomes a "Note" cursor.

3 Click on the model face where you want the text to begin.
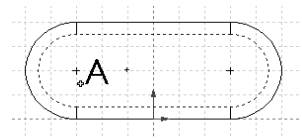
The **Text** dialog box opens.

4 Type your text into the edit box.

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

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

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

8 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. (For more information, see **Modify Sketch** on page 2-18.)

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
2-18.)

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 2-27.

To convert an entity:

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

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

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**. Symmetry symbols appear at both ends of the centerline to indicate that automatic mirroring is active.

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

4 To turn mirroring off, click ⚠ again.

Fillet (Sketch)

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

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

Note: You can either hold the **Ctrl** key and pre-select two intersecting sketch entities, or you can make your selections after the dialog box appears.

To create a fillet in a sketch:

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

A **Sketch Fillet** dialog box appears.

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

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

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

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

- or -

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

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

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

5   Click ⬜ to cancel the last fillet.

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

For information about creating a fillet feature, see **Fillet/Round** on page 4-20.

Offset Entities

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

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

To create a sketch curve offset from a model edge:

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

2 Click 🗗 or **Tools, Sketch Tools, Offset Entities**.

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

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

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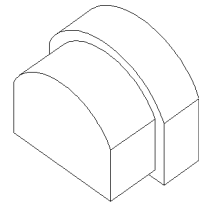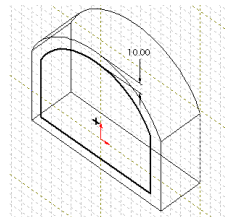
To change the size of the offset:

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

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

Trim

Use the **Trim** tool to:

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

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

To trim a sketch element:

1 Click the **Trim** tool. ✂

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

3 Click the highlighted segment to delete it.

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

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

Extend

Use the **Extend** tool to extend a line, arc, or centerline to meet a line, arc, circle, ellipse, spline, or centerline.

To extend a sketch element:

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

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

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

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

   - or -

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

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

Detach Segment On Drag

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

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

To detach a sketch segment:

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

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

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

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

Override Dims on Drag

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

To override dimensions by dragging:

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

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

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

3 Click 🔵 or 🔧 to rebuild the part and exit the sketch.

Automatic Solve

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

To turn automatic computation on or off:

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

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

Align Grid

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

To align the grid to a model edge:

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

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

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

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

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

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

– or –

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

Modify Sketch

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

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.

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** (or **Insert, Cut, Extrude**) and specify the **End Conditions** in the **Extrude Feature** dialog box (as illustrated),

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

Note: The examples show the the effect of extruding a boss with sketches closed in both directions
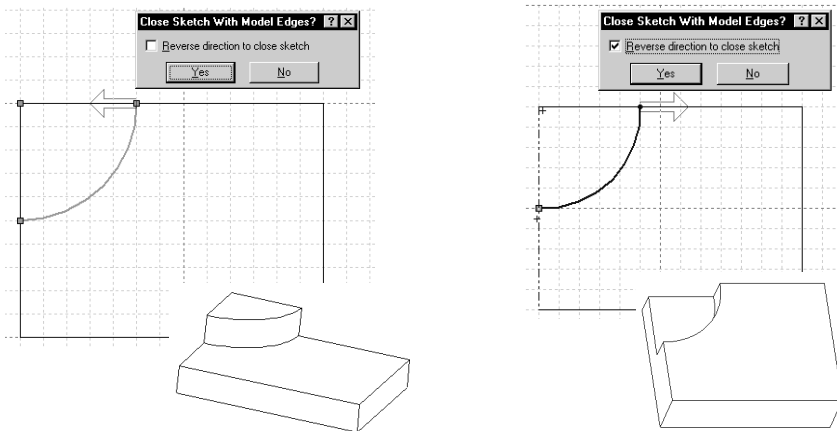
Check Sketch for Feature

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

To check a sketch:

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

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

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

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

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

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

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

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

Legal Contour Types:

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

Dimensioning Sketches

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

There are two methods of dimensioning a sketch:

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

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

    - or -

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

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

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

To add a specific type dimension to a sketch:

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

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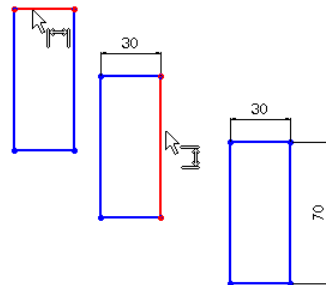
To preview a dimension:

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

2 Move the cursor around and observe the dimension preview.

3 Click when the preview displays the dimension that you want.
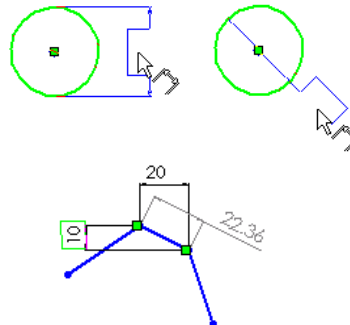
Dimensioning with Equations

You can insert dimensions using mathematical relations between dimensions or parameters, using dimension names as variables in the equation. Equations can also be used outside of a sketch. For a more complete discussion about using and editing equations in part or assembly documents, see **Equations** on page 5-7.

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 Select the dimension 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 selecting other dimensions to paste their names.
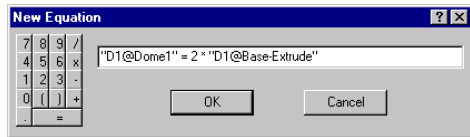
Equations are solved left to right, in the order in which they appear in the equation list.

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 dialog, the value of the angle is interpreted as radians. For example, sin(90) is evaluated as .89 (90radians) not 1.0 (90deg).

Changing Dimensions

To change a dimension:

1 Double-click on the dimension.

The **Modify** dialog box appears.

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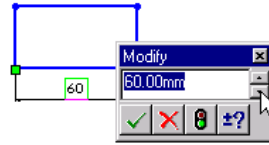
In addition, you can use the row of buttons below the value window to do any of the following:

   ☑ Accept the current value and exit.

   ✖ Restore the original value and exit.

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

   ±? Reset the spin box increment value.

Dimensioning Arcs and Circles

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

To change dimension options for arcs and circles:

1 Right-mouse click the dimension.

2 Select **Properties** from the menu.

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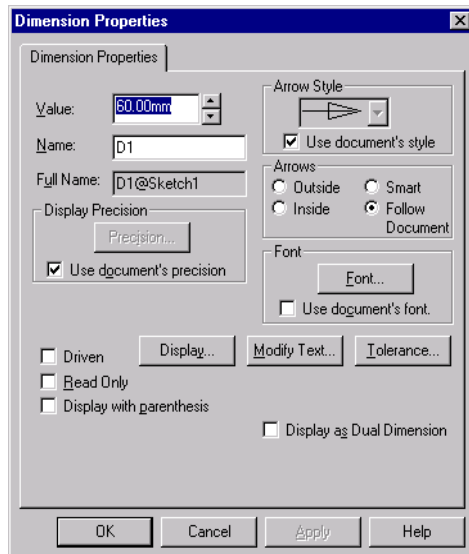
The following illustrations show some of the options for dimensioning between arcs and circles.
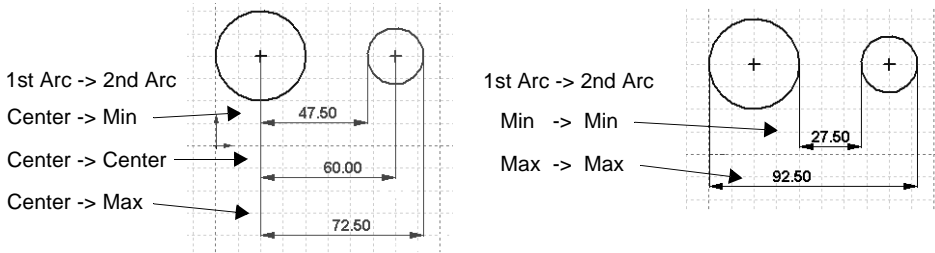
To dimension the true length of an arc:

1 Click , or **Tools, Dimensions**.

2 Click the arc.

3 Click both endpoints of the arc.

4 Click to place the dimension.

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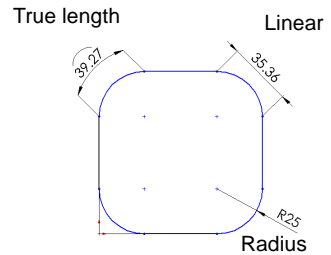
For dimensions on circular features, you have the following options:

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

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

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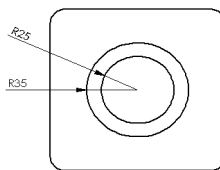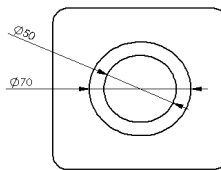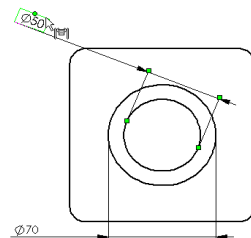
Radius   Diameter   Linear

Dimension Properties

You set the default properties for dimensions on the **Detailing** tab of the **Options** dialog box. See *Appendix A*, **Detailing Options** on page A-3. or **Dimension Properties** on page 9-11 of the *Detailing* chapter for more information. Also, see **Dimensioning Arcs and Circles** on page 2-24.

You can modify properties of selected dimensions at any time.

To modify the properties of a dimension:

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

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

2   Change values as described below.

3   Click **OK**.

The following properties are common to all types of dimensions.

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

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

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

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

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

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

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

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

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

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

Using Silhouette Edges

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

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

To use a silhouette edge:

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

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

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

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

To use silhouettes in a complicated part or large assembly:

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

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

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

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

Geometric Relations

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

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

Geometric relations can be created two ways:

❑ Automatically created by the SolidWorks software

❑ Manually created by the user

Automatic Relations

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.

Turn **Automatic Relations** on or off for the current document by clicking **Tools, Sketch Tools, Automatic Relations**. Specify **Automatic Relations** for all documents by clicking **Tools, Options, General**.

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

| Horizontal | Vertical | Coincident | Midpoint |

| Intersection | Tangent | Perpendicular |
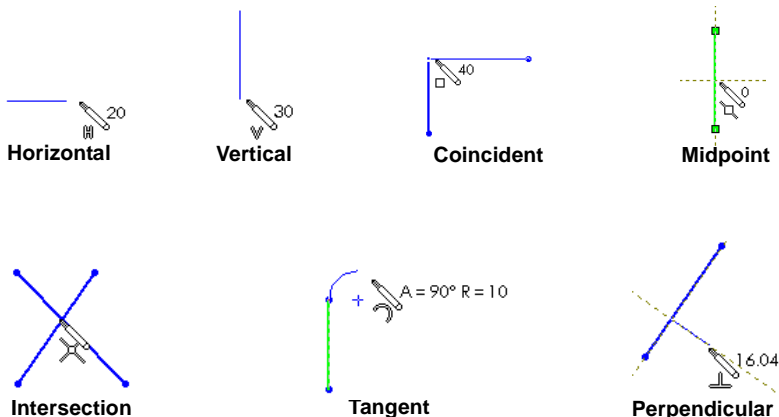
Manually-Created Relations

To apply relations manually:

1 Click ![perpendicular icon], or click **Tools, Relations, Add**.

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

For many relations, you can select more than two entities. (For a complete discussion of all the sketch relations, refer to the **Add Relations Table** on page 2-30.)

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

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

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

To remove the last relation you added, click the **Undo** button, ![undo icon].

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

Note: 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."
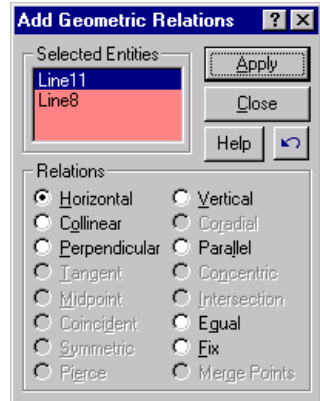
Add Relations Table

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

Table continued on next page

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

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

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

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

*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 [icon] or **Tools, Relations, Display/Delete**. If your sketch becomes "overdefined" or "not solved," use **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 has two tabbed pages:

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

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

Using the Relations page

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

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

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

For example, you can choose **Dangling** to list all the dangling relations in the sketch.
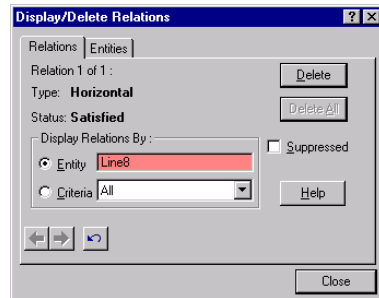
2   Click the **Previous** arrow, [icon], or the **Next** arrow, [icon], to view each relation in turn (if there are more than one).

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

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

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

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

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

Using the **Entities** page

1 Click the **Entities** tab.

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

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

- If the entity is defined as existing in an **External Model**, the **External Information** section lists the **Assembly** name and the **Owner** part within the assembly.
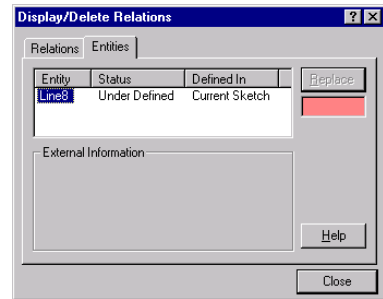
2 To replace an entity, in the list box select an entity that you want to replace.

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

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

4 Click the **Replace** button.

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

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

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

Reattaching Dangling Relations

When there is a single dangling relation, a red handle is displayed at the midpoint of the dangling segment.To reattach the relations, drag the handle onto a sketch or model entity that is suitable to replace the dangling relation. The cursor displays a "No" symbol if the entity is unsuitable.

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

- if there are multiple dangling relations, or

- if the entity is a sketch point.

Scan Equal

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

To locate equal radii and/or line lengths:

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

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

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

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

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

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

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

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

5 Click **Close**.

Constrain All

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

To solve relations in an imported drawing:

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

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

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

Fully Defining Sketches

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

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

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

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

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

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

With SolidWorks 98 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.
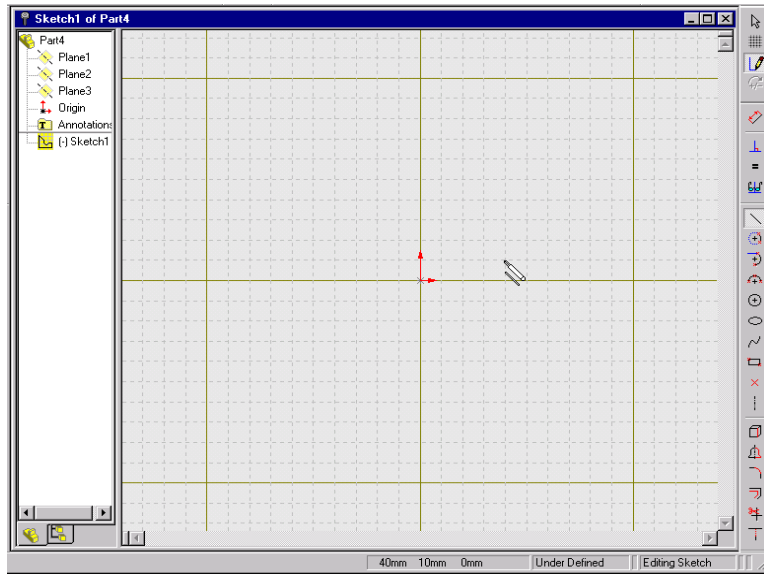
The following information is in the status bar at the bottom of the sketch window.

- 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 2-35 for more information.)

- The text, "Editing Sketch." This makes it obvious that you are in sketch mode even if you choose to turn off the sketch grid while you are working.

- When the cursor points to a menu item or tool button, look for a brief description of the menu item or button on the left side of the status bar.

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

To sketch on Plane2 or Plane3:

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

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

3 Open a sketch.

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

The Sketch Origin

The sketch origin is displayed in red in the active sketch. 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 click **Tools, Options** and select the **Grid/Units** tab.

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

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

To turn grid snap off:

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

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

For more information about additional ways to customize the sketch grid, such as changing the grid spacing, see **Grid/Units** on page A-16.

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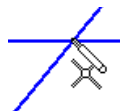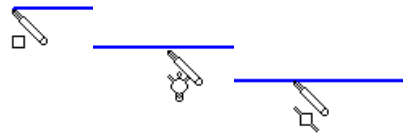
❑ When you sketch an arc, the cursor changes as it moves around the arc.

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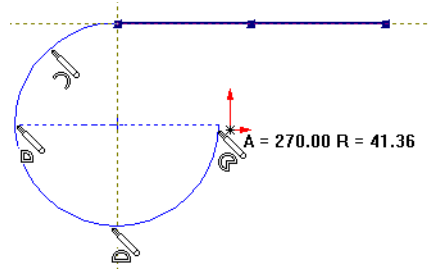
As you sketch, the SolidWorks software monitors your activity and provides *inferencing lines* to help you work more efficiently. While sketching, notice that dashed inferencing lines align your cursor with lines or points that you have already sketched and with existing model geometry.

❑ 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.
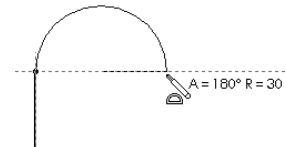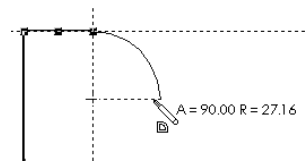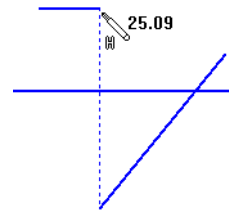
The Right-Mouse Button Menu

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

The right-mouse menu changes depending on the task that you are performing. Only those actions that are appropriate are listed as menu items.
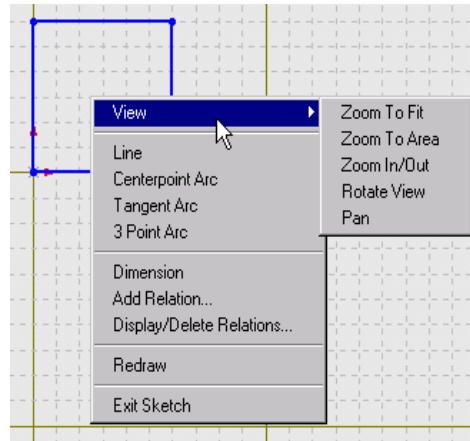
Cutting and Pasting Multiple Sketch Entities

You can cut and paste, or copy and paste, multiple sketch entities from one sketch to another or within the same sketch.

Select the sketch entities and drag while using the following modifier keys:

❑ To *copy* within the same document or different documents, press the **Ctrl** key while dragging.

❑ To *move* within the same sketch, press the **Shift** key.

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

Note: A new sketch has to be open in the target document to do the drag and drop operation.

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 ![sketch icon] or click **Insert**, **Sketch** from the main menu**.**

3 The following happens:

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

- The sketch toolbar is active.

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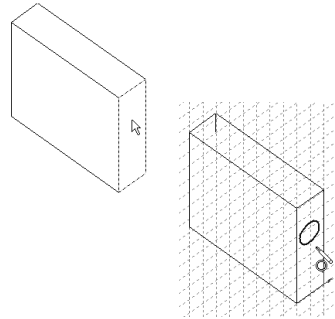
To sketch on a different face, you need to click the new face and open a new sketch. Note how the grid appears on the broad face of the base part in the this illustration.

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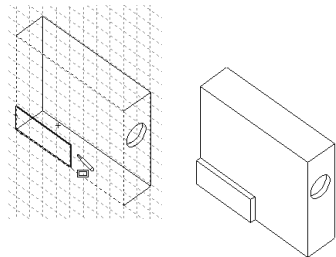
To turn the translucent sketch plane on or off:

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

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

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

Editing a Sketch

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 on

- a face

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

- a sketch name in the FeatureManager design tree.

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

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

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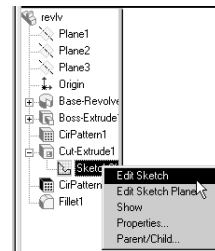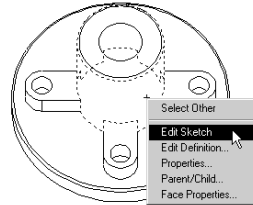
Exiting a Sketch

You exit a sketch when you:

- Create a feature from your sketched profile. For example, when you extrude a 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 Features from Sketches

Sketches are the basis for many different types of features. See Chapter 4, *Creating Features*, for information about using sketches to create solid model geometry; see Chapter 3 for information about creating reference geometry.

Does it matter where I start sketching?

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

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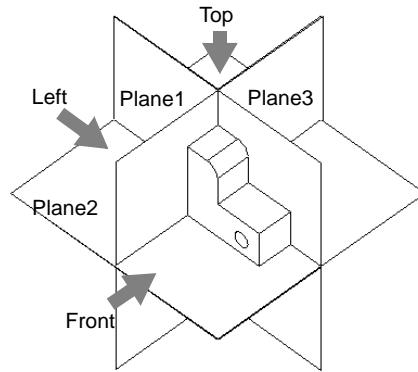
If you open a new part and start sketching *without* selecting a plane, the sketch is on **Plane1** by default and it is a *front* view.
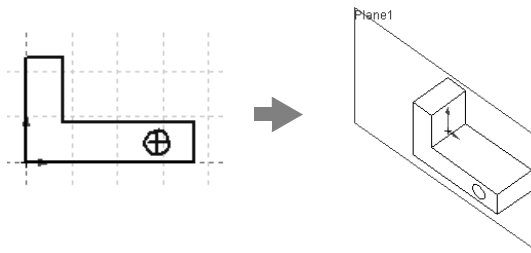
If your first sketch is a top view, you should select **Plane2** in the FeatureManager design tree *before* you click the **Sketch** icon.
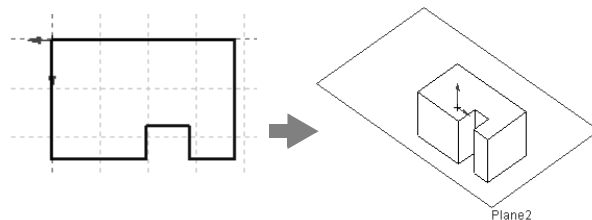
If your first sketch is a left or right view, select **Plane3**.

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

If you make a mistake or change your mind, you can reorient the part (to change "Front" to "Top" for example) with the **Update** button in the **View Orientation** dialog. For more information, see **Using Named Views** on page 1-17.

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
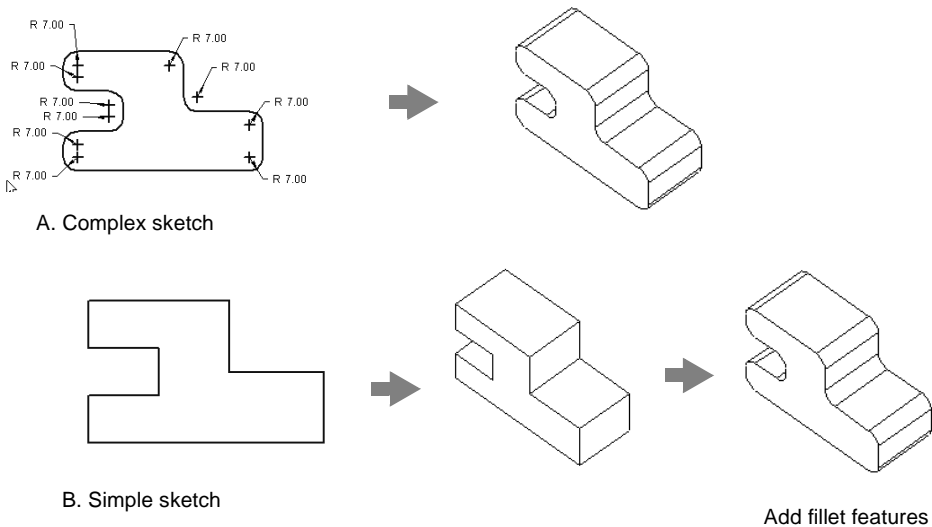
Here are some things to consider:

❑ **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.

3

Reference Geometry

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

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

❑ Creating and using construction planes

❑ Creating an axis

❑ Converting sketch lines to construction geometry

❑ Creating 3D reference curves

❑ Creating and using surfaces

❑ Displaying reference geometry

Creating a Construction Plane

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

To add a plane:

1   Click **View, Planes**.

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

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

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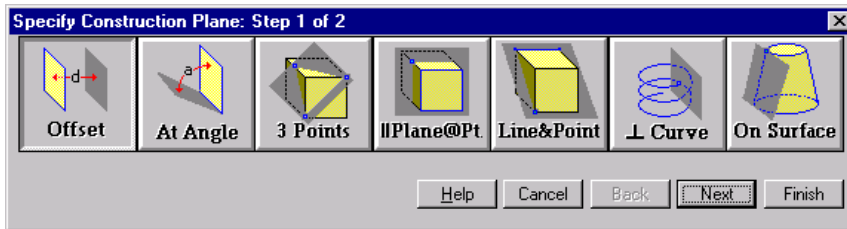
Offset. Creates a plane parallel to a plane or face, offset by a specified distance.

1   Select a plane or a planar face.

2   Specify the offset **Distance**.

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

4   Click **Finish**.

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

1   Select a plane or a planar face.

2   Select an edge, axis, or sketch line.

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

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

5   Click **Finish**.

Three Point Plane. Creates a plane through three points.

1   Select three vertices, points, or midpoints.

2   Click **Finish**.

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

Parallel Plane at Point. Creates a plane through a point parallel to a plane or face.
1. Select a plane or planar face and a vertex, point, or midpoint.
2. Click **Finish**.

Line and Point. Creates a plane through a line, axis, or sketch line and a point.
1. Select an edge, axis or sketch line and a vertex, point, or midpoint.
2. Click **Finish**.

Perpendicular to Curve at Point. Creates a plane through a point and perpendicular to an edge, axis, or sketch curve.
1. Select an edge, axis, or sketch curve and a vertex or point.
2. Click **Finish**.

On Surface. Creates a plane on a conical surface.
1. Select a surface.
2. Select a plane that intersects that surface.
3. If no solution is displayed, you will also need to select an edge on that surface.
4. Click **Other Solutions**, if necessary.
5. Click **Finish**.

Moving, Resizing, and Copying Construction Planes

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

To display the plane's handles:

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

- Click on the edge of the plane.

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

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

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

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

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

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

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

- or -

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

2 Select **Edit Definition**.

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

Changing Construction Plane Names

To change the names for Planes 1, 2, and 3 in the current document, click 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 **Reference Geometry** tab.

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

4 Click **OK**.

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 **Apply** 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 set the default to either hide or show all temporary axes.

To display temporary axes:

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

To turn the display of reference axes on or off:

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

To hide or show an individual axis:

1 Right-click 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 ⬚ or **Insert**, **Reference Geometry, Axis**.

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

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

- **Two Planes.** Select **View, Planes**, then hold the **Ctrl** key and select two construction planes.

- **Two Points.** Select two sketched points or two vertices.

- **One Line.** Select one line or a linear edge.

- **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 Entity, Construction Geometry**. For more information, see **2D Sketching** on page 8-36.

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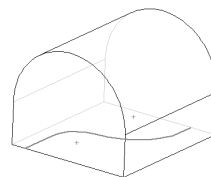
The curve appears in the FeatureManager design tree and on the selected face.
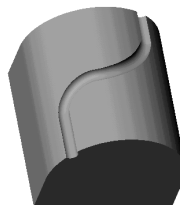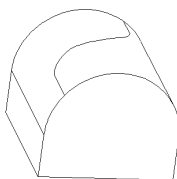
Projected curve used as sweep path

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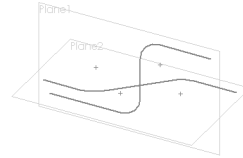
2 **Ctrl**-click to select both sketches.

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

The system creates the projected curve.
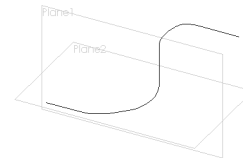
You can select individual segments of a projected curve if the option **Highlight all of selected features** is <u>not </u>checked on the **Tools, Options, Edges** page. Otherwise, the entire reference curve is selected (highlighted) when any part of the curve is selected.

Note: During the creation of features such as lofts, if the reference curve is selectable, the whole reference curve is highlighted whether the **Highlight all of selected features** option is checked or not.

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
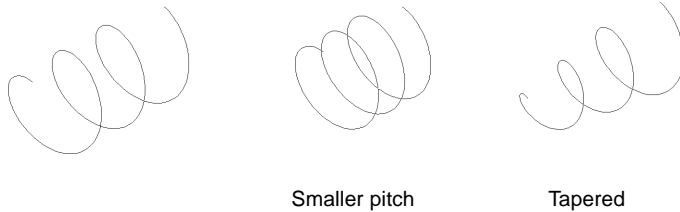
9   Click **OK**.

To create a spiral:

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

2   Close the sketch.

3   Select the circle.

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

5   In the **Defined by** scroll box select **Spiral**.

6   Enter a value for the **Pitch** - the change of the radius for each revolution.

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

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

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

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

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

11 Click **OK**.

Split Line

Split Line projects a sketched curve onto selected model faces. It divides a selected face into multiple separate faces so that each can be selected and modified individually. **Split Line** is used with the **Parting Line** option in the **Draft** feature (see page 4-26). Also, **Split Line** may be used as a hold line for face blend fillets (see **Face Blend with Hold Line** on page 4-22.

To project a split line onto a planar face:

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

2   Exit the sketch.

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

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

5   In the **Project Split Line** dialog, click the **Sketch to Project** box and select the sketch in the FeatureManager design tree.

6   Click the **Faces to Split** box, then select all the faces around the perimeter of the part that you want the split line to pass through.

7   Click **Finish**.
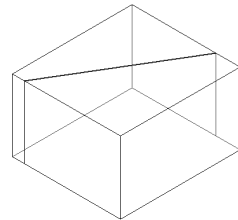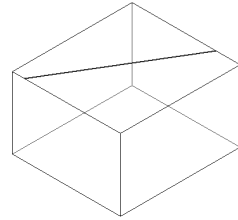
In special cases where there is an interruption in the selected face, you can project the curve in a single direction, and specify the direction of pull.

In this example, the **Face to Split** is cylindrical, and the sketch plane of the **Sketch to Project** lies in the area of a cut.

- Click **Single Direction** to project the curve in one direction only.

- Click **Reverse Direction** if the preview indicates that the curve projects the wrong way.
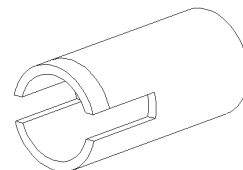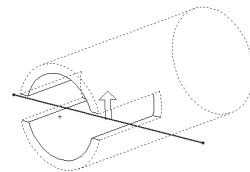
If you do not click **Single Direction** here, the curve projects all the way around the cylinder.

To create a silhouette split line on a part with curved or tangent faces:

1 With a part open, click **Insert, Reference Geometry, 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**.
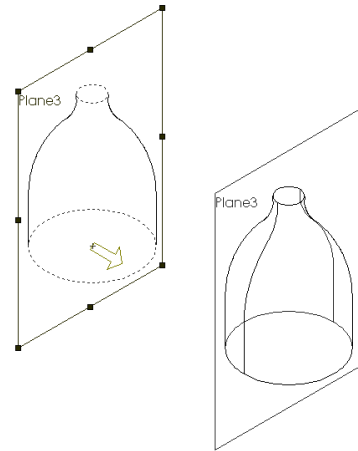
Curve Through Reference Points

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 the first cell in the first row and click **Insert**.

Point	X	Y	Z
1	1.00mm	1.00mm	1.00mm
2	1.50mm	2.00mm	0.50mm
3	2.00mm	3.00mm	0.00mm
4	4.00mm	3.50mm	0.00mm
5	5.50mm	4.00mm	-0.50mm

3 Enter the coordinates for **X**, **Y**, and **Z**. Press the **Tab** key to move from cell to cell, and from row to row.
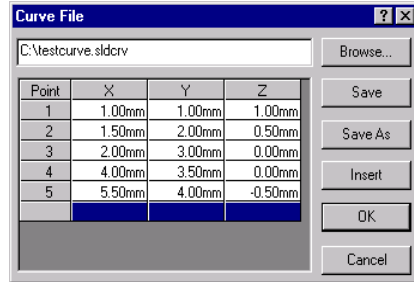
Notice that the value in the first cell in the **Point** column is set to 1. The numbers in the **Point** column define the order in which the points will be connected.

4 Continue adding as many points as needed (one row is one point). The larger the number of points, the greater your control of the resulting curve.

Notice the preview of the curve in the graphics area.

5 If necessary, you can insert a new point between existing points. Click the point number to select the row below where you want the new point to occur, then click **Insert**. A new row is added above the row you clicked. All the points below the selected row move down.

6 Double-click in a previously filled cell to go back and change its value, if necessary.

7 To delete a point, select its row and press the **Delete** key.

8 You can save the curve file for re-use. Click **Save** or **Save As**, navigate to the desired location, and specify the filename. If you do not specify an extension, SolidWorks adds the extension **.sldcrv**.

9 Click **OK** to complete the curve and close the dialog box.

To edit a curve file:

Right-click the curve name in the FeatureManager design tree, or right-click the curve in the graphics area. Select **Edit Definition** from the menu.

To create a curve from 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 13, *Importing and Exporting Files*, for information about importing 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**.
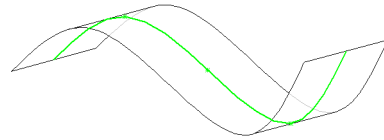
Extruded surface (MidPlane type) from a sketched spline

See **Extrude** on page 4-2 for more information about extruded features.

To revolve a surface:

1 Sketch a profile and a centerline around which to revolve the profile.

2 Click **Insert, Reference Geometry, Revolved Surface**.

3 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**.
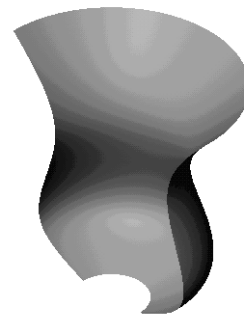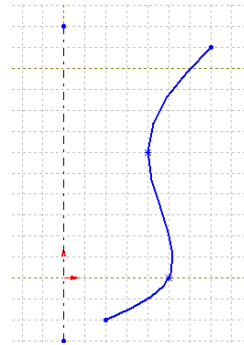
See **Revolve** on page 4-9 for more information about revolved features.

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 Curve** 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**.
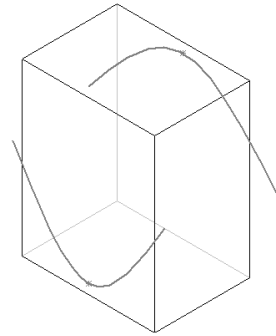
Profiles sketched on model faces
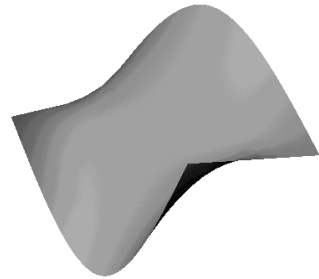
See **Loft** on page 4-15 for more information about lofted features.

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.

Section   Path

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 Curve** box, then click the sketch, curve, or model edge for the guide curve(s).
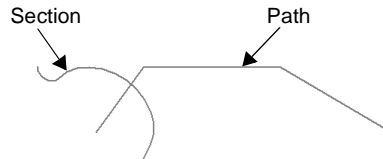
7 Select options as desired under **Orientation/Twist Control**, **Advanced Smoothing**, and **Maintain Tangency**.

8 Click **OK**.
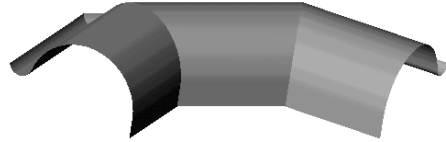
See **Sweep** on page 4-10 for more information about swept features.

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.
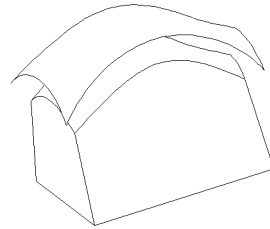
Offset Surface from a lofted model surface

5 Click **OK**.

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.
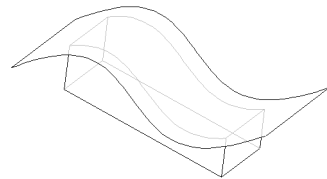
5 Specify the side(s) you want to thicken. Examine the preview to see the effect.

Thicken Side 1   Thicken Side 2   Thicken Both Sides
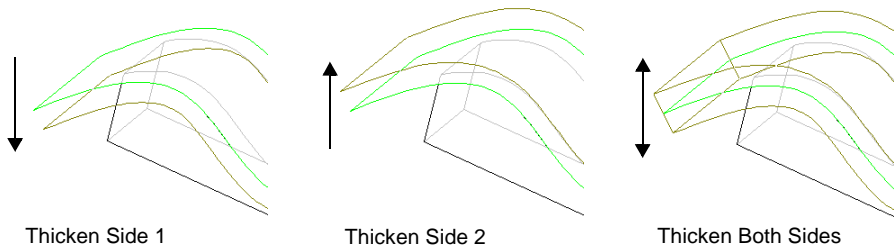
If you are creating a boss, the thickened surface *must* intersect the existing part.

6 Click **OK**.
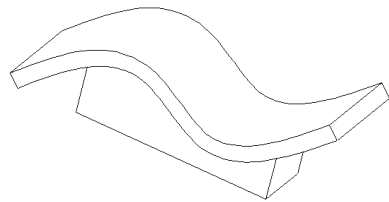
To cut a part with a surface:

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**.

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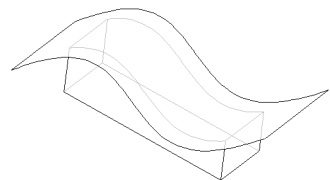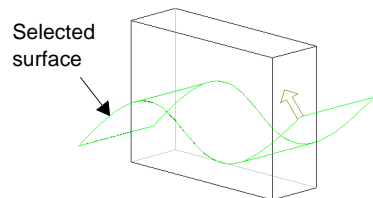
6 To hide the surface if desired, right-click the **Surface** feature in the FeatureManager design tree and select **Hide**.

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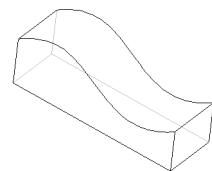
To cut a part by thickening a surface:

1   Create a reference surface in the part model, or import one.

2   Select the surface.

3   Click **Insert**, **Cut**, **Thicken**.

4   In the **Thicken Feature** dialog, enter the desired thickness for the feature.

5   Specify which side to thicken.

6   Click **OK**.

Selected surface
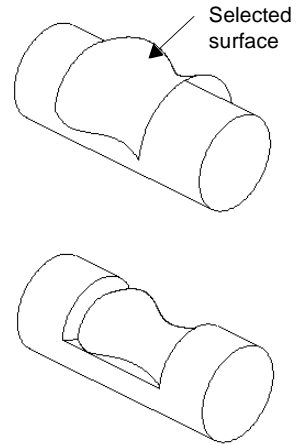
Displaying Reference Geometry

Reference geometry is a necessary tool, but when it is visible, it may make the image of the part appear too complex and hard to understand. You can set system-wide visibility options to filter out various types of reference geometry. Click **Tools**, **Options**, **Reference Geometry** and select the kinds of reference geometry that you want to see, by default. See *Appendix A,* **Reference Geometry** on page A-20 for more information.

To set the visibility of reference geometry in the current document, Use the options under the **View** menu.

By default, reference geometry is hidden in drawing views. For information about importing reference geometry to drawings, see **Reference Geometry in a Drawing** on page 9-15.

4

Creating Features

Features are the individual shapes that, when combined, make up the part. You can also add some types of features to assemblies. Some features originate as sketches; other features, such as shells or fillets, are created when you select the appropriate menu command and define the dimensions or characteristics that you want. This chapter describes the following:

❑ 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

The Features Toolbar

For many of the features described in this chapter, there is an icon on the Features toolbar. For others, an icon may be available, but not displayed by default. You can customize the toolbars to include icons for some of the functions you use most.

Click **Tools, Customize**, and click the **Commands** tab. Choose a category, and examine the available buttons. Drag the buttons you want into any toolbar, delete buttons you 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).

Solid Feature   Thin Feature
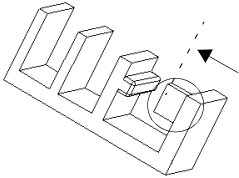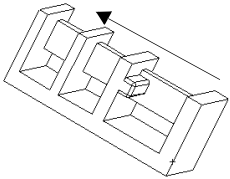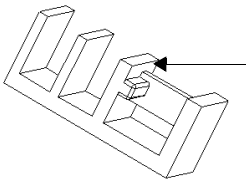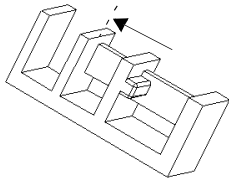
See **Surface** on page 3-13 for information about using **Extrude** to create a surface.
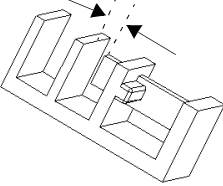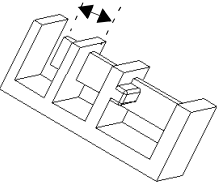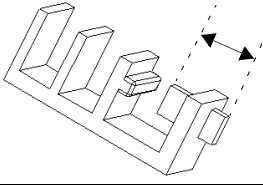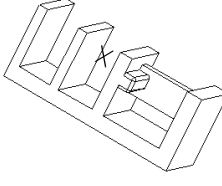
The **End Condition** tab of the **Extrude Feature** dialog lets you define the characteristics of extruded features created with these commands on the **Insert** menu:

- **Base, Extrude**
- **Boss, Extrude**
- **Cut, Extrude**
- **Features, Hole, Simple**
- **Features, Hole, Wizard**
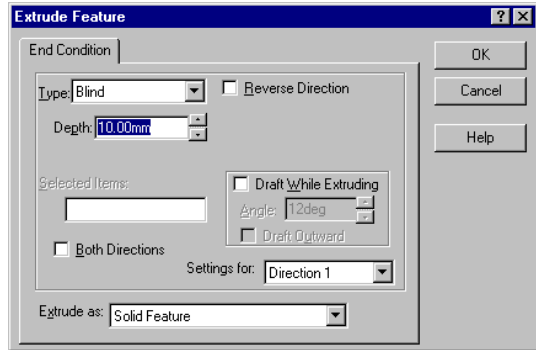
The following table describes the **End Condition** tab of the **Extrude Feature** dialog box and the various options for extruded bosses and cuts.

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.
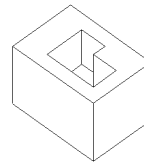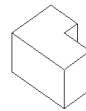
Default cut   Flip side cut

❑ **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.
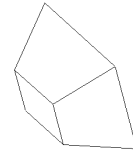
❑ **Selected Items.** If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area. The selection is indicated in the **Selected Items** box.

No draft   25° draft angle – inward

❑ **One Direction** or **Both Directions.**

- **One Direction** extrudes the feature in one direction from the sketch plane. This is the default (**Both Directions** *unchecked*).

- **Both Directions** extrudes the feature in both directions from the sketch plane. Specify all the settings for the first direction (**Direction 1**), then select **Direction 2** from the **Settings for:** box, and specify the settings for the second direction.

Note: Observe the preview to verify the direction and depth of the feature.

❑ **Extrude As:** a **Solid Feature** or a **Thin Feature**.

- **Solid Feature** adds (or removes) solid volumes to the model.

- **Thin Feature** adds (or removes) thin-walled volumes to the model. A **Thin Feature** base can also be used as a basis for a sheet metal part. See Chapter 11, *Sheet Metal*, for more information.
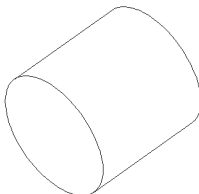
A circle extruded as a solid feature
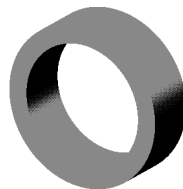
A circle extruded as a thin feature
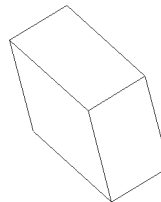
A rectangle extruded as a solid feature

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**).
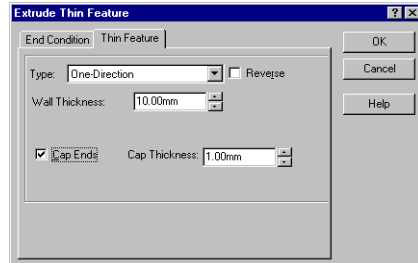
❑ **Wall Thickness** specifies the thickness of the thin feature wall.

❑ **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).

Extruded Base/Boss

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

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

- **Insert**, **Base**, **Extrude**

- **Insert, Boss, Extrude**

- The **Extruded Base/Boss** button on
the Features toolbar

The **Extrude Feature** dialog box appears.

A sketched
rectangle...

...extruded to make a box

2 Select the **Type** of extrusion. (See the table
on page 4-3 for more information.)
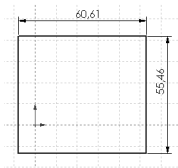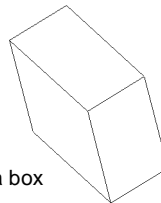
3 Set the desired **Depth** of the extrusion.

For bosses on sheet metal parts, click **Link to Thickness** to link the depth of the
boss to the thickness of the base feature.

4 To add a **Draft** angle, click **Draft While Extruding**. Enter a value for the draft
Angle, and click **Draft Outward**, if desired.

5 If the **Type** you specified relies on the selection of a surface or vertex, click that
item in the graphics area. The selection is indicated in the **Selected Items** box.

6 Examine the preview and click **Reverse Direction** to extrude in the opposite
direction, if necessary.

7 To extrude the feature in two directions from the sketch plane, click **Both
Directions**, click **Direction 2**, and enter the settings for the second direction.

8 Select **Solid Feature** or **Thin Feature** in the **Extrude as:** box.

If you are extruding a **Solid Feature**, skip ahead to Step 10. If you are extruding
a **Thin Feature**, proceed to Step 9.

9 Select the **Thin Feature** tab.
- a) Select a wall thickness **Type**. (See the descriptions on page 4-6.)
- b) Specify the **Wall Thickness**, and if you want to add the wall thickness to the
opposite side of the sketched profile, click **Reverse**.
- c) To add fillets to the corners of an open profile thin base feature, click **Auto
Fillet**, and specify the desired **Fillet Radius**.
- d) To cap the ends of a closed profile base feature, click **Cap Ends**, and
specify the desired **Cap Thickness**.

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

Extruded Cut 🔳

To extrude a solid or thin feature cut:

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

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

The **Extrude Feature** dialog box appears.

2 Select the desired **Type**. (See the table on page 4-3 for more information.)

3 Specify the **Depth**.
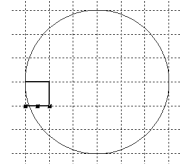
...results in this extruded cut with **Flip Side to Cut** selected.

4 Examine the preview and click **Reverse Direction**, if necessary.

5 Select **Flip Side To Cut**, if desired.

6 To add a **Draft** angle, click **Draft While Extruding**. Enter a value for the draft **Angle**, and click **Draft Outward**, if desired.

7 If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area. The selection is displayed under **Selected Items**.

8 To extrude the feature in two directions, click **Both Directions**, and indicate settings for **Direction 1** and **Direction 2**.

9 Specify **Solid Feature** or **Thin Feature** in the **Extrude as:** box.

If you are extruding the cut as a **Solid Feature**, skip ahead to Step 11. If you are extruding the cut as a **Thin Feature**, proceed to Step 10.

10 Select the **Thin Feature** tab at the top of the dialog box.
   a) Select a wall thickness **Type**. (See the descriptions on page 4-6.)
   b) Specify the **Wall Thickness**.
   c) To extrude in the opposite direction, click **Reverse**.

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

Thin Feature Cut from a sketched line
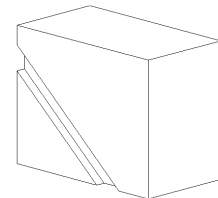
Revolve ⬢ ⬢

Revolve creates a base, boss, or cut by revolving a sketch around a centerline. The default angle is 360°. See **Surface** on page 3-13 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**.
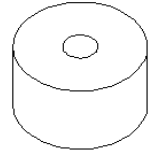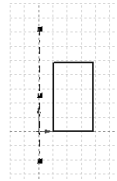
Sweep

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 3-13 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**
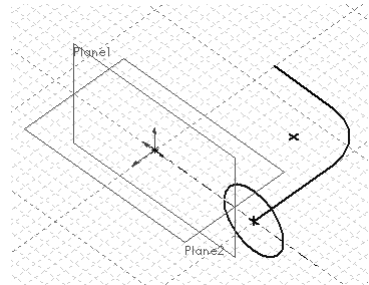
The **Sweep** dialog box appears. For a simple sweep, you only need to use the **Sweep** tab in the dialog box.

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.

Follow Path

- Select **Keep Normal Constant** if you want the section to remain parallel to the beginning section at all times.

7 If you picked an edge as the path, and you want to continue the sweep along all tangent edges, click **Propagate Along Tangent Edges**.

ends remain parallel

Keep Normal Constant

8 If you want to continue the sweep profile up to the last face encountered at the ends of the path, click **Align with End Faces**.

9 Click **OK**.

For examples of creating simple sweep features, refer to the tutorial, *Learning to Use SolidWorks 98*, Chapter 5.
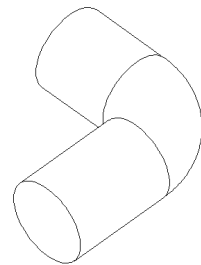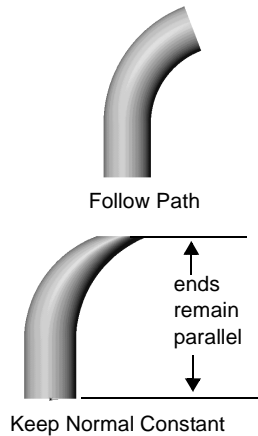
Sweep with Guide Curves

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.

Path

Guide Curve
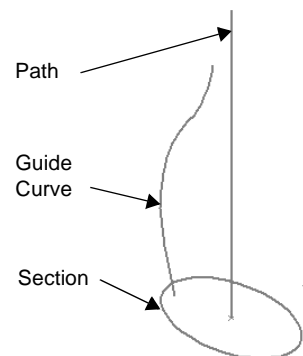
Section

It is important to create the section *after* the path and guide curve(s). The intermediate profiles of the sweep are *dependent* on both the path and guide curves for their definition.

Note: To help you with creating the path and guide curves, you may want to create an auxiliary profile sketch first. Then, after the guide curves and path are done, derive a copy of the sketch, then underive it to break the link to the original sketch. Use this as the section in the remaining steps. See **Derived Sketch** on page 2-43 for more information.

As you sketch the section, pay special attention to any relations that may be added automatically, such as **Horizontal** or **Vertical**. These relations influence the shape of the intermediate sections, and may produce results that are not what you want. If necessary, use **Display/Delete Relations** 🔛 to delete the unwanted relations. This allows the intermediate sections to twist as desired.

4 Add **Pierce** relations in the *profile* sketch, where the guide curve(s) intersect the profile. It is important that the *section* be constrained to the *curves*, and not vice-versa.

5 Click **Insert, Base/Boss/Cut, Sweep**. The **Sweep** dialog box appears.

6 On the **Sweep** tab, click the **Sweep Section** box, then select the profile sketch in the FeatureManager design tree or in the graphics area.

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

8 Under **Orientation/Twist Control**, there are two additional choices besides **Follow Path** and **Keep Normal Constant** (see page 4-11):

   • **Follow Path and 1st Guide Curve -** The twist of the intermediate sections is determined by the vector from the path to the first guide curve. The angle between the horizontal and the vector remains constant in the sketch planes of all of the intermediate sections.

   • **Follow 1st and 2nd Guide Curve -** The twist of the intermediate sections is determined by the vector from the first to the second guide curve. The angle between the horizontal and the vector remains constant in the sketch planes of all of the intermediate sections.

GuideCurve

GuideCurve

Section

Path

Follow path and 1st
guide curve

Follow 1st and 2nd
guide curves (twisted)
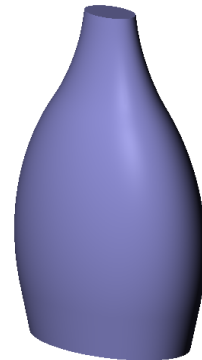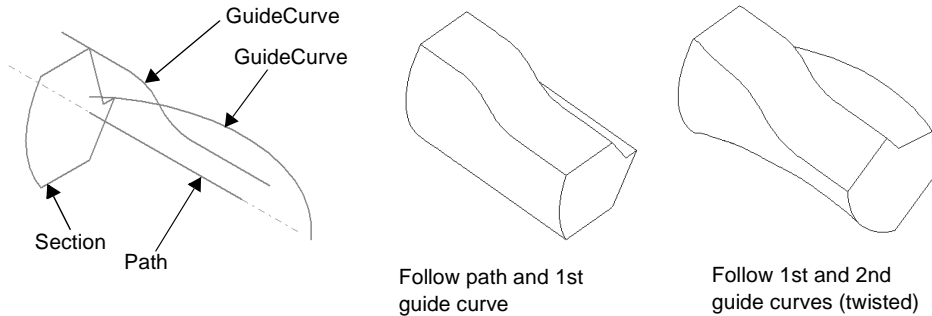
9 On the **Advanced** tab, click the **Guide Curve(s)** box, then select the guide curve(s). Use the **Up** and **Down** buttons to rearrange the order of the guide curves if needed.

10 Under certain conditions, these additional options for sweeps with guide curves are available on the **Advanced** tab:

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

- **Advanced Smoothing** - if the sweep section has circular or elliptical arcs, the sections are approximated, resulting in smoother surfaces. Sketch arcs may be converted to splines.

11 Specify the tangency options, if desired (the default is **None**). See **End Tangency Options for Sweeps** on page 4-14.

12 Click **OK**.

To view the intermediate sections in the sweep:

You can preview the intermediate sections created along the sweep path. This option is only available for sweeps with guide curves.

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

2 On the **Advanced** tab, click **Show Intermediate Profiles**.

3 Use the up and down arrows beside the **Profile Number** box to scroll through the profiles.

Note: You can activate this option before creating the sweep, if desired.

Additional notes on using Guide Curves:

❑ The path and the guide curves may differ in length. The sweep length is determined by the shortest overlapping distance.

  • If the guide curves are longer than the path, the sweep is as long as the path.

  • If the guide curves are shorter than the path, the sweep is as long as the shortest guide curve.

❑ Guide curves may meet at a common point, which is the apex of the swept surface.

❑ You can use any of the following items as a guide curve: sketched curves, model edges, or reference curves. See **Curve** on page 3-7 for information about creating reference curves.

End Tangency Options for Sweeps

With a simple sweep, end tangency is controlled by choosing either **Follow Path** or **Keep Normal Constant** in the **Orientation/Twist Control** list.

When you use guide curves to create a sweep, you have additional options for controlling the tangency at the ends of the path. You specify the **Start Tangency** and **End Tangency** options on the **Advanced** tab of the **Sweep** dialog box. These tangency options are only effective when you use guide curves in the sweep.

Choose from these options:

❑ **None**. No tangency is applied.

❑ **Path Tangent**. The sweep is normal to the path at the start or end.

❑ **Direction Vector**. The sweep is tangent to a selected linear edge or axis, or to the normal of a selected plane. Click the **Direction Vector** button, then click the edge, axis, or plane.

❑ **All Faces**. The sweep is tangent to the adjacent faces of existing geometry at the start or end. This selection is only available when the sweep is attached to existing geometry.

These tangency options are similar to the tangency options for lofts. Refer to the table on page 4-19 for illustrated examples.

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.

A circle on Plane1 and a point on Plane4.

See **Surface** on page 3-13 for information about using **Loft** to create a surface.
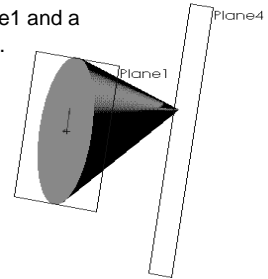
Simple Loft

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**

4 Select the profiles in order, by clicking a corresponding point on each profile.

You do not have to select the vertices precisely; the vertex closest to the selection point is used. A preview curve connecting the selected entities is displayed.
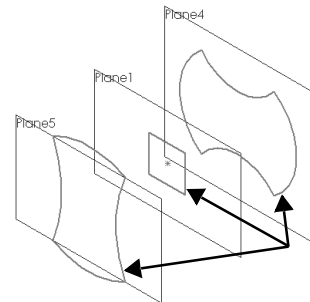
Click corresponding points

5 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 with the wrong vertex connected to deselect it, then click again to select a different point on the profile.

- To clear all selections and start over, right-click in the graphics area, select **Clear Selections**, and try again.

6 Choose from these options as needed:

- Click **Close Along Loft Direction** - to create a closed body along the loft direction. This connects the last sketch and the first sketch automatically.

- Click **Maintain Tangency** to cause the corresponding surfaces in the resulting loft to be tangent if the corresponding lofting segments are tangent. Faces that can be represented as a plane, cylinder, or cone are maintained. Other adjacent faces are merged, and the sections are approximated. Sketch arcs may be converted to splines.

- Click **Advanced Smoothing** to obtain smoother surfaces. This option is available only if the loft sections have circular or elliptical arcs. The sections are approximated, and sketch arcs may be converted to splines.

7 Specify the tangency options, if desired (the default is **None**). See **End Tangency Options for Lofts** on page 4-18.

8 Click **OK**.

For examples of creating simple loft features, refer to the tutorial, *Learning to Use SolidWorks 98*, Chapter 6.
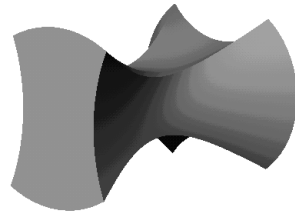
Loft with Guide Curves

You can create a *guide curve loft* by sketching two or more profiles and one or more guide curves to connect the profiles. Guide curves help you control the intermediate profiles that are generated.

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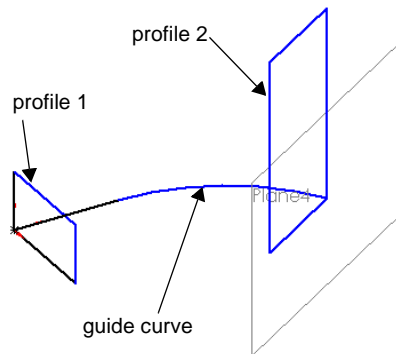
It is recommended that these relations be added to the profile sketches.

4 Click **Insert, Base** (or **Boss**), **Loft.**

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

6 Click the **Guide Curve(s)** box, then select the guide curves.

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

8 Click **Close Along Loft Direction**, **Advanced Smoothing**, and **Maintain Tangency** if needed. See **Simple Loft** on page 4-15 for more information about these options.

9 Specify the tangency options, if desired (the default is **None**). See **End Tangency Options for Lofts** on page 4-18.

10 Click **OK**.
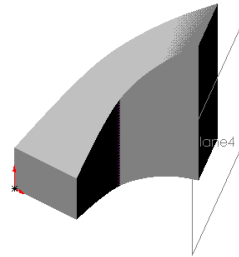
Notes on Creating Lofts with Guide Curves:

❑ Guide curves must intersect all the sketched profiles.

❑ There is no limit to the number of guide curves you may use.

❑ Guide curves can intersect at a point.

❑ You can use any of the following items as a guide curve: sketched curves, model edges, or reference curves of any kind. See **Curve** on page 3-7 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 3-11.

❑ Guide curves can be longer than the resulting loft. The loft stops at the end of the smallest guide curve.

❑ You can further control the behavior of the loft by creating the same number of segments on all the guide curves. The end points of each segment mark corresponding points for transition of the profiles.

Center Line Loft

You can create a loft that uses a variation of a guide curve that acts as a center line. The sketch planes of all the intermediate sections are normal to the center line. The center line may be a sketched curve or a reference curve.

To use a center line to guide a loft:

1 Create the loft profiles and guide curves if any.

2 Sketch a curve, or create a reference curve. The center line curve must intersect the area inside each closed profile.

3 Click **Insert, Base** (or **Boss**)**, Loft**, then select the profiles and guide curve(s) as described above.

4 Click the **Advanced** tab, click the **Center Line** box, then click the sketch or curve in the FeatureManager design tree or the graphics area.

   If you want to preview the effect of the center line, click **Show Sections**. Using the slider, you can adjust the **Number of Sections** that are displayed. You can view the sections in order by clicking the up and down arrows beside the **Section Number**.

| Sketches | Center Line OFF | Center Line ON |
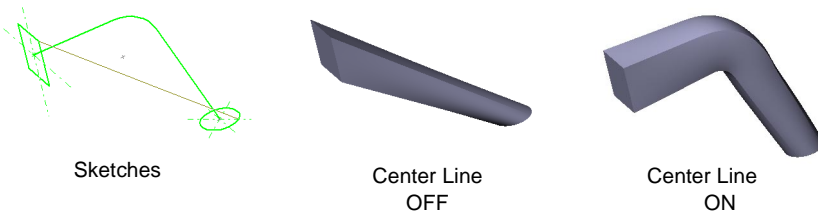
End Tangency Options for Lofts
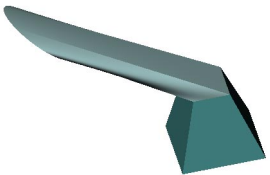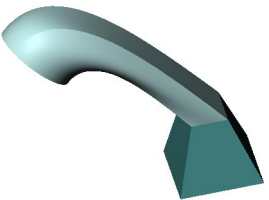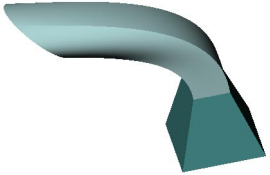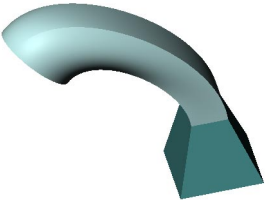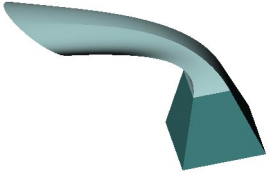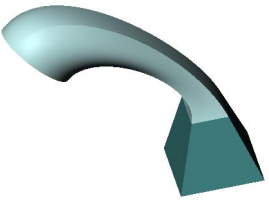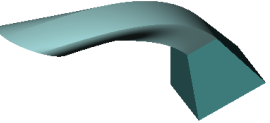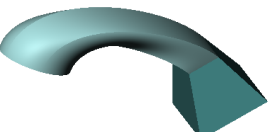
You have several options for controlling the tangency of a loft at the starting and ending profiles.

You specify the **Start Tangency** and **End Tangency** options on the **Advanced** tab of the **Loft** dialog box. Choose from these options:

❑ **None**. No tangency is applied.

❑ **Normal to Profile**. The loft is normal to the sketch plane of the profile at the start or end.

❑ **Direction Vector**. The loft is tangent to a selected linear edge or axis, or to the normal of a selected plane. Click the **Direction Vector** button, then click the edge, axis, or plane.

❑ **All Faces**. The loft is tangent to the adjacent faces of existing geometry at the start or end. This selection is only available when the loft is attached to existing geometry.

The examples in the table are created from this base feature and these sketches.

Direction Vector

End Profile   Start Profile

Start Tangency None End Tangency None		Start Tangency None End Tangency Normal to Profile	
Start Tangency Normal to Profile End Tangency None		Start Tangency Normal to Profile End Tangency Normal to Profile	
Start Tangency All Faces End Tangency None		Start Tangency All Faces End Tangency Normal to Profile	
Start Tangency Direction Vector End Tangency None		Start Tangency Direction Vector End Tangency Normal to Profile	

Fillet/Round 🖼️

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

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

❑ Add larger fillets before smaller ones. When several fillets converge at a vertex, create the larger fillets first.

❑ Add drafts before fillets. If you are creating a molded or cast part with many filleted edges and drafted surfaces, in most cases you should add the draft features before the fillets.

❑ Save cosmetic fillets for last. Try to add cosmetic fillets after most other geometry is in place. If you add them earlier, it takes longer to rebuild the part.

❑ To enable a part to rebuild more rapidly, use a single **Fillet** operation to treat several edges that require equal radius fillets. Be aware however, that when you change the radius of that fillet, all the fillets created in the same operation change.

Constant Radius

To fillet multiple model edges and/or faces:

1 Click **Fillet** 🖼️ on the Features toolbar, or **Insert**, **Features, Fillet/Round**.

2 Select the faces and edges that you want to fillet or round.

3 Specify the fillet **Radius**.

4 Select **Fillet Type** of **Constant Radius**.

5 Verify that the correct number of edges and faces are shown in the **Items to Fillet, Edge Fillet Items** list.

Selected face

Before and after creating a fillet feature
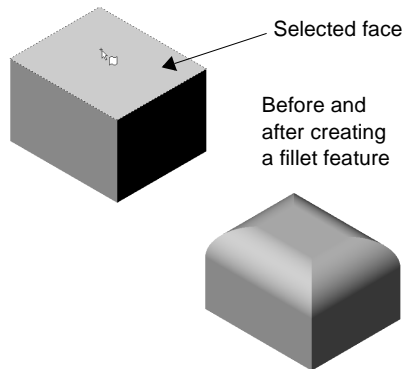
6 By default, the **Propagate to Tangent Faces** option is selected. This extends the fillet to all faces that are tangent to the selected face or edge. If you do not want the fillet to extend along tangencies, *deselect* the checkbox.

Selected edge

7 Select the **Overflow Type**. See **Overflow Type** on page 4-24 for more information.

8 Click **OK**.

Propagate to Tangent Edges OFF

Propagate to Tangent Edges ON
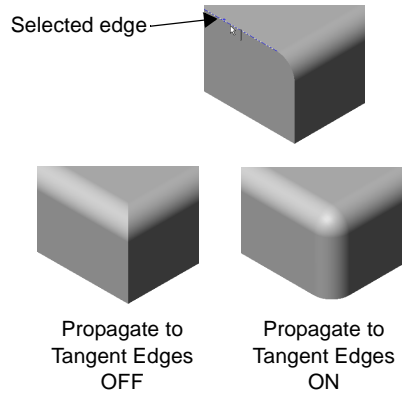
Variable Radius

You can also make variable-radius fillets. You specify a different radius for each vertex of the edge being filleted or rounded.

To insert a variable radius fillet:

1 Click **Fillet** on the Features toolbar, or **Insert**, **Features, Fillet/Round**.

2 Change the **Fillet Type** to **Variable Radius**.

3 Select the edges that you want to round.

4 Choose a transition type:

- **Smooth transition** creates a fillet that changes smoothly from one radius to another when matching a fillet edge on an adjacent face.

- **Straight transition** creates a fillet that changes from one radius to another more abruptly without matching edge tangency with an adjacent fillet.

5 In the **Vertex List** under **Items to Fillet**, highlight **Vertex1**. Notice that the current radius value appears next to the selected vertex.

6 Change the **Radius** to the desired value. Repeat with each vertex until you have changed all the radii as desired.

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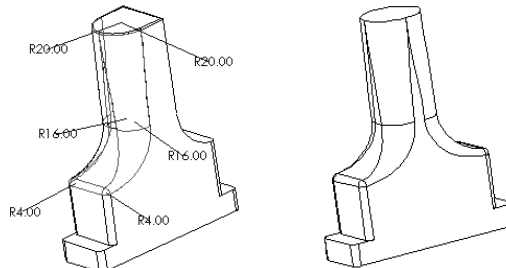
To change the radii of the fillets after they are inserted, double-click the fillet feature, then double-click and modify the dimension displayed on each vertex.

For an example of creating variable radius fillets, refer to the tutorial, *Learning to Use SolidWorks 98*, Chapter 9.

Face Blend

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

To insert a face blend fillet:

1   Click **Fillet** 🔲 on the Features toolbar, or **Insert**, **Features, Fillet/Round**.

2   Change the **Fillet Type** to **Face Blend.**

3   Click the first face or set of faces to be blended. Under **Items to Fillet**, the selection is indicated in **Face Set 1**.

4   Click the **Face Set 2** box, and click the face(s) to blend with.

5   Specify the **Radius**.

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**.
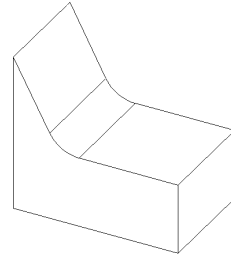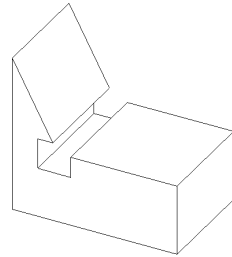
Face Blend with Hold Line

You can determine the radius and shape of a face blend fillet by setting a boundary, or hold line, for the fillet. The hold line can be an edge on the part or a split line projected on a face to be filleted.

In a fillet of this type, the radius of the fillet is driven by the distance between the hold line and the edge to be filleted.

To insert a face blend fillet using an edge:

1 Click **Fillet** 🔲 on the Features toolbar, or **Insert**, **Features**, **Fillet/Round**.

2 From the **Fillet Type** box, select **Face Blend**.

3 Click the **Face Set 1** box, then click a model face (or faces) to fillet.

4 Click the **Face Set 2** box, then click the *adjacent* model face (or faces) to fillet.

5 Click the **Advanced Face Fillet** tab and check the **Set Fillet Boundary** checkbox.

6 Click the model edge or edges that mark the end of the fillet. The names are listed in the **Hold Lines** box.

7 Click **OK**. The faces that share an edge with the **Hold Lines** are removed by the fillet.
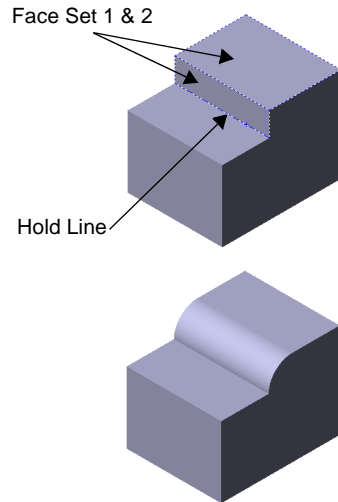
To insert a face blend fillet using a split line:

1 Create a split line on a model face. See **Split Line** on page 3-10 for information about creating a split line.

2 Click **Fillet** 🔲 on the Features toolbar, or **Insert**, **Features**, **Fillet/Round**.

3 Select **Insert**, **Features**, **Fillet/Round**.

4 Under **Fillet Type**, select **Face Blend**.

Do not enter a radius. The radius is determined by the distance between the **Hold Line** and the edge to be filleted.

5 Click one model face to be blended. Under **Items to Fillet** the selection is indicated in **Face Set 1**.

6 Click the **Face Set 2** box, and click an *adjacent* model face.

7 Click the **Advanced Face Fillet** tab, and check the **Set Fillet Boundary** checkbox.

8 Click the split line. The name is listed in **Hold Lines** box.

9 Click **OK**.
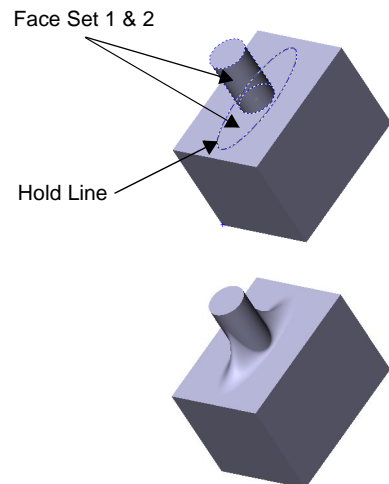
Overflow Type

One of the options on the **Fillet Feature** dialog box is **Overflow Type**. The following describes the available types and the results obtained with each type.

- **Default.** The system chooses one of the following options, depending on the geometry conditions (convexity of edges being filleted and the adjacent edges, and so on).

- **Keep Edge.** Maintains the integrity of adjacent linear edges. However, the fillet surface is broken into separate surfaces, and in many cases the top edge of the fillet may have a dip in it.

- **Keep Surface.** Uses the adjacent surface to trim the fillet. As a result, the fillet edge is continuous and smooth, but the adjacent edge is disturbed.
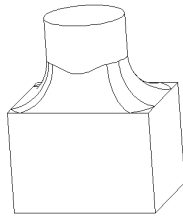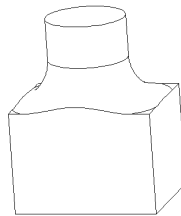
Keep Edge   Keep Surface

Chamfer

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

To create a chamfer:

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

2 On the model, select the edges, faces, loops, or vertex to chamfer. (You can also do this as the first step.)

3 Look for an arrow on the selected edge; it indicates the direction of the chamfer. If the direction is not what you want, click **Flip Direction**.

4 Examine the **Items to Chamfer** list. If you selected an incorrect item, click it again to deselect, or right-click and select **Clear Selections**.

5 Select a chamfer type from the list, and specify the necessary parameters:

   • If you selected **Distance-Distance**, enter values for the distances on either side of the chamfer edges, or click **Equal Distance** and specify one value.

   • If you selected **Angle-Distance**, enter values for distance and angle. An arrow indicates the direction in which the distance is measured. If necessary, click **Flip Direction**.

   • If you selected **Vertex-Chamfer**, enter distance values for the three chamfer edges, or click **Equal Distance** and specify a single value.

6 Click **OK**.
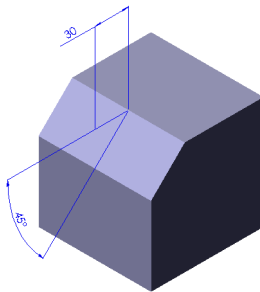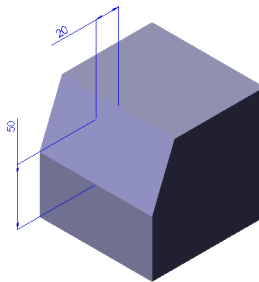
Angle-Distance   Distance-Distance   Vertex-Chamfer
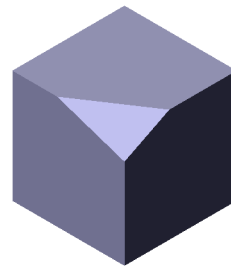
Draft ▣

Draft 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*.

If the **Draft** icon does not appear on your toolbar, you can add it. Click **View, Toolbars, Customize**, and follow the instructions on the **Commands** tab.

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 ▣ or **Insert, Features, Draft**.

2 In the **Type of Draft** box, select **Neutral Plane**, click in the **Neutral Plane** box, and select the neutral plane in the graphics area.

3 Set the **Draft Angle**. If you want the draft to slant in the opposite direction, click **Reverse Direction**.

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

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

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

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

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

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

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

6 When you are satisfied with the results, click **OK**.
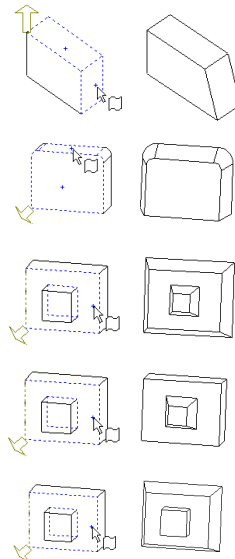
Parting Line

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

To draft on a parting line, you may first divide the faces to be drafted by inserting a **Split Line** (see page 3-10), 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 3-10.

3 Click 🔲 or **Insert, Features, Draft**.

4 In the **Type of Draft** box, select **Parting Line**.

5 Specify the **Draft Angle.**

6 Click the **Direction of Pull** box, and select an edge or face in the graphics area to indicate the direction of pull.

 Note the arrow direction, and click **Reverse Direction** if 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**.
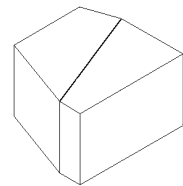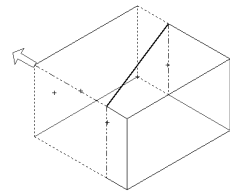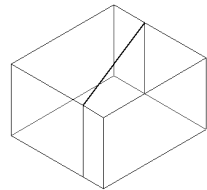
Hole 🔲

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

4 Specify the **Diameter** of the hole.

5 If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area. The selection is indicated in the **Selected Items** box.

6 Examine the preview and click **Reverse Direction**, if necessary.

7 To add a draft, click **Draft While Extruding**. Enter a draft **Angle** and click **Draft Outward** if necessary.

8 To create a hole in both directions, select the **Both Directions** checkbox. Indicate settings for **Direction 1** and **Direction 2**.

9 Click **OK**.

10 To position the hole:
   a) Right-click the hole feature in the model or the FeatureManager design tree and select **Edit Sketch**.
   b) Add dimensions to position the hole. You can also modify the hole diameter in the sketch.
   c) Exit the sketch or click **Rebuild**.
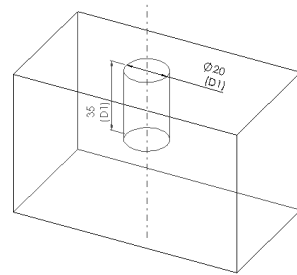
To change the diameter, depth, or type of the hole, right-click the hole feature in the model or the FeatureManager design tree, and select **Edit Definition**. Make the necessary changes, and click **OK**.

To insert a hole using the Hole Wizard:

1   Select a planar face on which to create the hole.

2   Click **Insert, Features, Hole, Wizard** or click **Hole** 🔟 on the **Features** toolbar.

3   Select the **Hole type** from the pull-down list (countersunk, counterbore, and so on).

4   In the **Section dimensions** area, click-pause-click a dimension value to select it. Enter the new dimension.

5   Select the **End Condition Type** from the pull-down list.

6   If the **Type** you specified relies on the selection of a surface or vertex, click that item now in the graphics area.

7   Click **Next**. The **Hole Placement** dialog appears.

8   Drag the centerpoint of the hole, or add dimensions to place the hole at the desired location. You can use the dimension tool 🖉 while the **Hole Placement** dialog is displayed.

9   Click **Finish**.
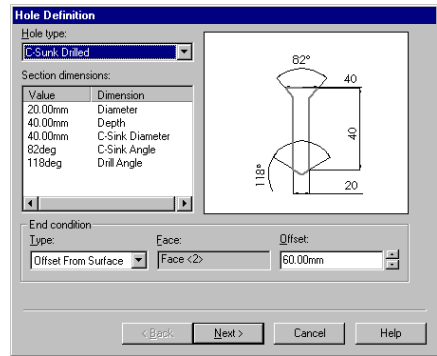
Shell

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

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**.
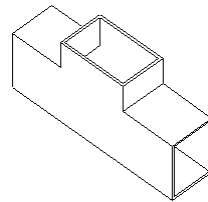
To set a different shell thickness for each face:

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**.

MultiThickness Faces

Face to Remove
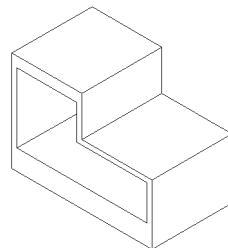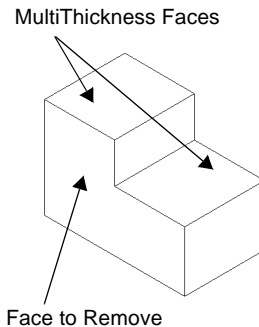
Rib

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.

Note: For better visibility, use **Isometric** and either **Hidden Lines Removed** or **Hidden in Gray** view.

4 Enter the **Thickness** of the rib and click **Next.**

5 Note the direction of the arrow in the preview. If necessary select **Flip Side of material** to reverse the direction in which material is added.

6 To add draft, select **Enable Draft** and enter the **Angle**. Select **Draft Outward** if necessary.

7 Click **Finish**.
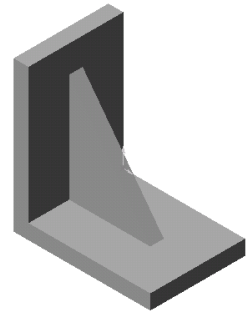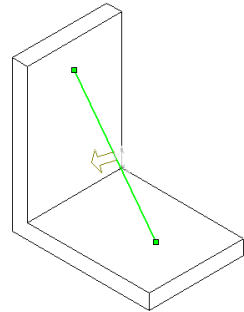
To insert a rib with multiple draft angles:

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.
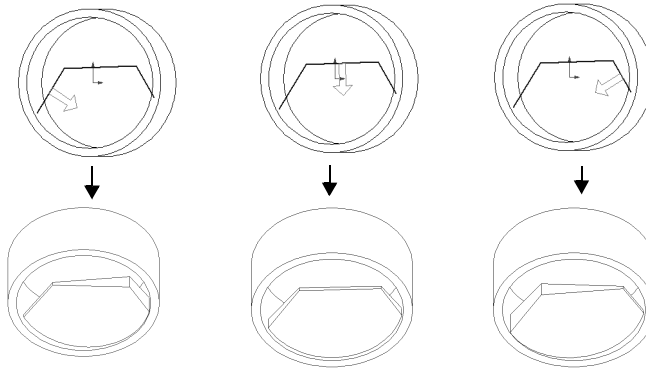
6   Click **Finish**.

Dome

You can add a *dome* feature to any planar model face.

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 **Elliptical Dome**. This creates a dome whose shape is a half ellipsoid, with a height equal to one of the ellipsoid radii.

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

6   Click **OK**.
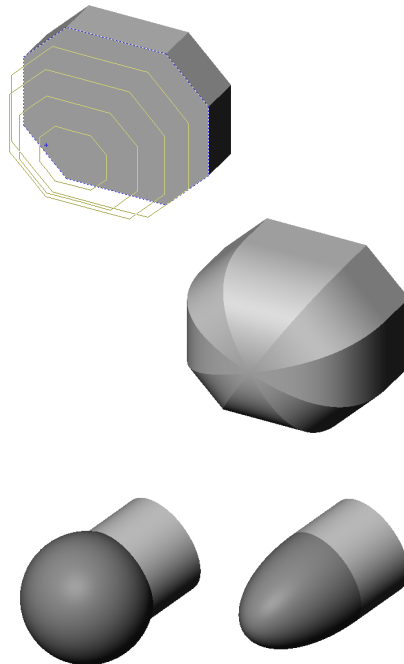
Elliptical Dome OFF   Elliptical Dome ON

Pattern/Mirror

Pattern repeats the selected feature(s) in a linear or circular array.

Mirror copies the selected feature(s) or all features, mirroring them about the selected plane or face.

❑ For a **Linear Pattern**, you select the feature(s), then specify the direction, the linear spacing, and the total number of instances.

❑ For a **Circular Pattern**, you select the feature(s) and an edge or axis as the center of rotation, then specify the angular spacing and the total number of instances.

❑ For a **Mirror Feature**, you select the feature(s) to copy and a plane about which to mirror them.

❑ To **Mirror All**, you select a planar face on the model, then mirror the entire model (base and all other features) about the selected face.

You can also create patterns of patterns, and mirrored copies of patterns.

For information about using a pattern of components in an assembly, see **Adding a Component Pattern** on page 7-9.

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 Select **Insert**, **Pattern/Mirror**, **Linear Pattern**.

3 With the **Items to copy** box selected, click the feature(s) to pattern.

4 Click a model edge or dimension to indicate the pattern direction or driving dimension for the **First Direction**. Notice the direction arrow on the model.

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

6 Specify the values for **Spacing** and **Total Instances** of the pattern.
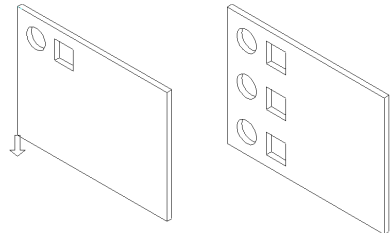
Spacing is the distance from a point on one feature to the corresponding point on the adjacent copy (*not* edge-to-edge spacing).

Total Instances includes the original feature(s) on which the pattern is based.

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

7   Click **Vary Sketch** if you want the pattern to change as it is repeated. See **Controlling and Modifying Patterns** on page 4-38 for more information about using **Vary Sketch**.

8   Click **Geometry Pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-38.

9   Click **OK** to create the pattern in one direction, or continue with Step 10 to create the pattern in two directions.

10   Select **Second Direction** in the scroll box, and click a different model edge to indicate this direction. Again, you may need to click **Reverse Direction**.

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

12   Click **OK**.
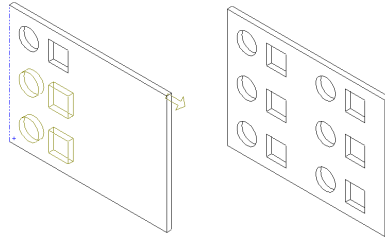
Circular Pattern

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

To create a circular pattern:

1   Create a base part and on the base part, create one or more cut, hole, or boss feature(s) that you want to repeat.

2   Create an axis or use an existing linear edge or axis around which to pattern the feature.

   Note: For any circular object in SolidWorks, there is a temporary axis. If axes are not displayed, click **View, Axes** (for user-defined axes) or **View, Temporary Axes** (for axes created implicitly by the model).

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

4   With the **Items to copy** box selected, click the feature(s) to pattern.

5   Select an axis or angular dimension to define the pattern.

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

7 Click **Reverse** if you want to create the pattern in a counter-clockwise direction. Otherwise, the pattern goes in a clockwise direction.

As you modify the values, a preview of the resulting pattern is displayed.
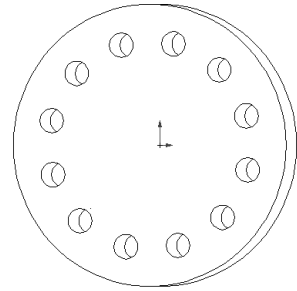
8 Click **Vary Sketch** if you want the pattern to change as it is repeated. (Refer to **Controlling and Modifying Patterns** on page 4-38 for more information on using **Vary Sketch**.)

9 Click **Geometry Pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-38.

10 Click **OK**.

You can use an equation to calculate the spacing angle used in a circular pattern.

To use an equation to control a circular pattern:

1 Create a circular pattern as described in **Circular Pattern**, above. Leave the default values in the **Spacing** and **Total Instances** boxes. It is not important what numbers you use at this time.

2 In the FeatureManager design tree, double-click the circular pattern feature. Two values appear in the model: **Total Instances** and **Spacing** angle.

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

4 Click the **Spacing** angle dimension in the model. The dimension name is entered in the **New Equation** dialog box.
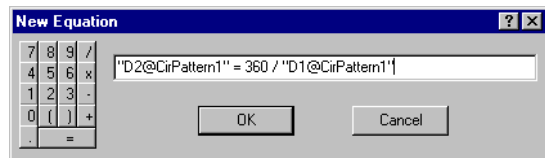
5 Using the calculator keypad in the dialog box, click the equals sign (=) and click **360 /**.

6 Click the **Total Instances** value in the model. Its full name is added to the end of the equation.

7 Click **OK** to complete the equation, and click **OK** again to close the **Equations** dialog box. Note the **Equations** folder ▣ in the FeatureManager design tree.

If you change the number of **Instances** in the pattern, the **Spacing** angle is re-calculated, so that the instances are always regularly spaced around the center.

8 Click **Rebuild** 🚦 , or click **Edit, Rebuild**.

For more information about using equations, see **Equations** on page 5-7.

Mirror Feature

Mirror Feature creates a copy of a feature (or features), mirrored about a plane. You can either use an existing plane or create a new one. If you modify the original feature, the mirrored copy is updated to reflect the changes.

To mirror a feature (or features):

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

2   With the **Mirror Plane** box selected, click on a plane or face.

3   With the **Items to copy** box selected, click a feature or features in the model or in the FeatureManager design tree.

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

4   Click **Geometry Pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-38.

5   Click **OK**.

Note: You can also mirror a feature about a face that is perpendicular to the sketch plane. Mirroring will be about the selected *face*, not about the feature itself.
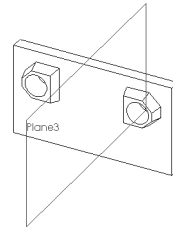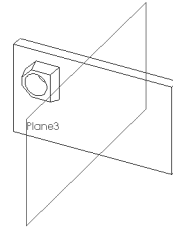
Mirror Pattern

To create a mirrored copy of a pattern:

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

2   With the **Mirror Plane** box selected, select a plane about which to mirror the pattern.

3   With the **Items to copy** box selected, click the *pattern* to be mirrored. You only need to select *one instance* of one element of the pattern in the model.

    A preview of the mirrored pattern is displayed.

4   Click **Geometry Pattern** if you want to make an exact, geometric copy of the faces and edges of the original feature. See **Geometry Pattern** on page 4-38.

5   Click **OK**.

Mirror All

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

To mirror a part around a planar face:

1   Establish a plane corresponding to the plane of symmetry of the complete part. This can be an existing plane or one you create.

2   Create the features for one half of the part on one side of this plane.

3   Select the face of the part half on the plane of symmetry.

4   Click **Insert, Pattern/Mirror, Mirror All.**

A mirror image of the original part half is joined to it at the selected face to make a complete, symmetrical part.
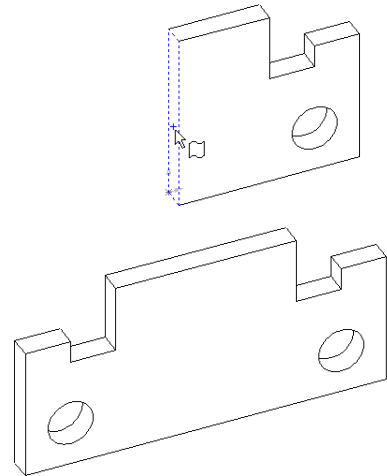
Pattern of Patterns

To create a pattern of patterns:

1   Select **Insert**, **Pattern/Mirror**, then either **Linear** or **Circular** pattern.

2   Select the pattern feature in the FeatureManager design tree, or select a face on a patterned feature in the model. You only need to select one instance of one element of the pattern in the model.

3   Proceed as described in **Linear Pattern** on page 4-33 or **Circular Pattern** on page 4-34.

Controlling and Modifying Patterns

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

❑ Use the **Geometry Pattern** option to make exact mirrored/patterned copies of features, and to improve system performance when using mirrored/patterned features.

❑ Use the **Vary Sketch** option to adjust the profile of the patterned feature based on its dimensions and relations to other features.

❑ Use **Delete Instance** to remove individual instances of the patterned features.

❑ Use a mathematical **Equation** to calculate values that define the pattern.

Geometry Pattern

You can make mirrored or patterned copies of features using the **Geometry Pattern** option. **Geometry Pattern** (a checkbox on the **Mirror/Pattern** dialog box) copies the faces and edges of the geometry being mirrored or patterned. Any references to other model geometry (such as **Up to Surface**, **Offset from Surface**) used to create the original feature are ignored (not solved).

The **Geometry Pattern** option also improves system performance when using complex mirrored or patterned features.

In the example below, the cut is extruded using the **Offset from Surface** end condition. When the **Geometry Pattern** option is *off*, the copies retain this design parameter. The end of the cut is offset the same distance from the selected face in each instance.

When the **Geometry Pattern** option is *on*, the copies are exact copies of the faces and edges of the original feature. The end condition is ignored, and the distance from the end of the cut to the selected face in not recalculated.

Item to Copy
Offset From Surface
End Condition

Geometry Pattern OFF

Geometry Pattern ON
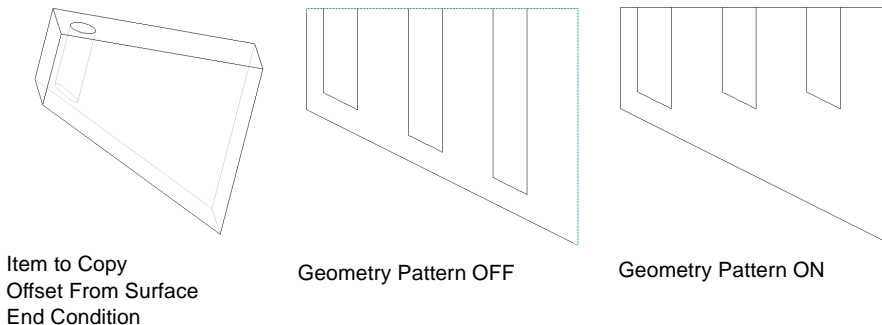
Note: You cannot use the **Geometry Pattern** option with any feature that has a face in common with another face on the model.
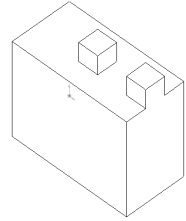
Vary Sketch

Use the **Vary Sketch** option if you want the pattern to change its dimensions as it is repeated. For example, you may want to maintain a specific distance between the edges of the base part and the patterned features.

In this example, the first feature (the 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.

Linear pattern *without* Vary Sketch

first feature

Linear pattern *with* Vary Sketch
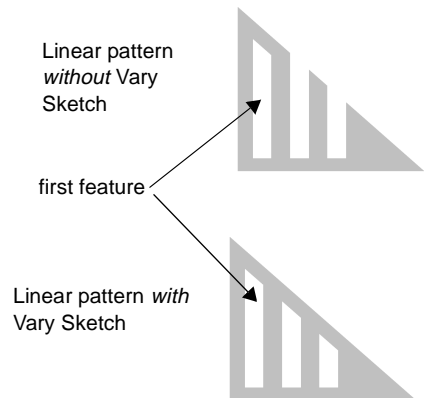
To create a variable pattern:

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

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

- The feature sketch should be *fully defined*.

2 Double-click one of the feature dimensions and click the spinbox arrow to change the dimension.

This gives you a preview of the way this dimension will drive the feature when you create the pattern. Try a different dimension for a different result.
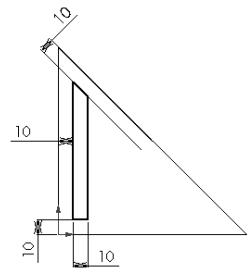
3 When you are satisfied that the dimensions are correct and the pattern will repeat as you wish, click **Insert, Cut, Extrude** (or **Insert, Boss, Extrude**) to create the first feature.

4 In the FeatureManager design tree, select the feature to repeat.

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

6 Click the dimension that you want to use to drive the pattern. Notice the preview arrow.

7 Click **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.
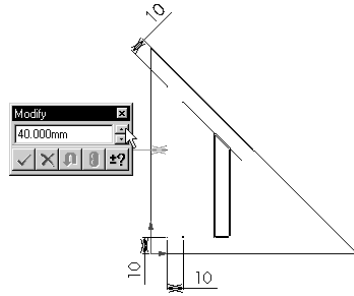
Delete Instance

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).

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

❑ Editing, moving, and copying features

❑ Using equations

❑ Dependency editing using Rollback, Suppress and Unsuppress

❑ Working with part configurations

❑ Using a design table

❑ Displaying a section view of the model

❑ Annotating parts

❑ Specifying lighting characteristics and material properties

The FeatureManager Design Tree

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 is constructed. The features are listed in the order in which they are created.
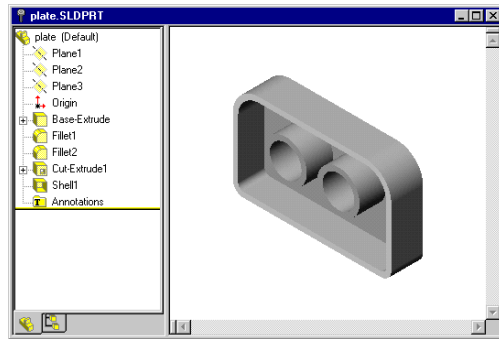
In the FeatureManager design tree of an active part, you can:

❑ Click the ⊞ sign beside an item's icon, or double-click an item's name, to expand the item and display its contents.

❑ Right-click a feature to edit its definition, sketch, or properties, or to view its parent/child relationships. See **Feature Editing** on page 5-3.

❑ Double-click a feature to expose its dimensions and drag handles.

❑ Click a feature name twice slowly (click, *pause*, click), type in a new name, and press **Enter** to change its name.

❑ Select a feature for dependency editing operations (**Rollback**, **Suppress, Unsuppress**). See **Dependency Editing** on page 5-10.

❑ 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.
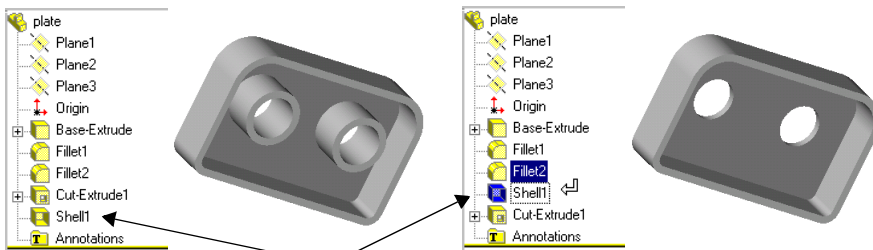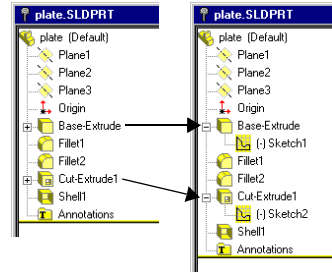
Changing feature order changes the part

Feature Editing

This section describes feature editing functions. These functions are accessible using either main menu selections, or right-mouse menu selections on FeatureManager design tree icons.

Edit Definition

You can edit the definition of a feature to change its parameters. For example, you can edit the depth of an extruded feature, or the instance count in a pattern.

To edit the definition of a feature:

1   Click a feature either in the graphics area or in the FeatureManager design tree, then click **Edit, Definition,** or right-mouse click the feature and select **Edit Definition**. Depending on the selected feature type, the appropriate dialog box appears.

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

3   Click **Apply** or **OK** to accept the changes; click **Cancel** to discard the changes.

Edit Sketch

For features that are based on sketches, you can edit the sketches to change the geometry of the features.

To edit the sketch of a feature:

1   For features made from *single* sketch (extruded features, for example), click the feature either in the graphics area or in the FeatureManager design tree, then click **Edit, Sketch**. Alternatively, you can right-mouse click the feature, and select **Edit Sketch**.

   For features made from *multiple* sketches (loft or sweep features, for example), expand the feature in the FeatureManager design tree (click the ⊞ beside the feature name) in order to see the sketches. Right-mouse click the sketch, and select **Edit Sketch**.

2   Edit the sketch as needed.

3   Click ▨ to exit the sketch, or right-click anywhere in the graphics area, and select **Exit Sketch, or click Rebuild** ▨ . The part rebuilds with your changes.

Edit Sketch Plane

You can change the plane of a sketch to change the location of a feature on the model. Sketches can be on the standard planes, on reference planes you create (see **Creating a Construction Plane** on page 3-2), or on planar model faces.

To change the sketch plane:

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

2 Select a new sketch plane by clicking either a plane in the FeatureManager design tree, or a planar model face.

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

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 edit feature properties:

1 Click a feature either in the graphics area or in the FeatureManager design tree, then click **Edit, Properties,** or right-mouse click the feature and select **Properties**. The **Feature Properties** dialog appears.

2 Edit the color, or click **Suppressed** (see **Dependency Editing** on page 5-10 for more information about suppressing features).

3 Click **OK**.

To change feature colors:

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

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

3 In the **Entity Property** dialog, click the **Change Color** button. Select a color from the color palette or define a custom color.

4 Click **Advanced** to change lighting properties such as shininess and transparency.
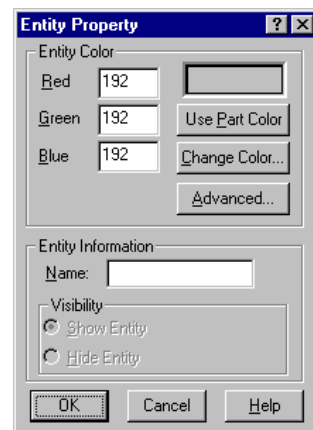
5 To restore the feature to the color of the part, click **Use Part Color**.

6 Click **OK**.

To change the color of a feature face:

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

2 In the **Entity Property** dialog, click the **Change Color** button. Select a color from the color palette or define a custom color.

3 Click **Advanced** to change lighting properties such as shininess and transparency.

4 To restore the face to the color of the feature it belongs to, click **Use Part Color**. (Face colors override colors specified for the *feature*.)

5 Click **OK**.

Parent/Child Relationships

When features are built upon other features, their existence depends upon the existence of the previously-built feature. The new feature is called a *child* feature. For example, a hole is the child of the solid in which it is cut.

A *parent* feature is an existing feature upon which others depend. For example, a boss is the parent feature to a fillet that rounds its edges.

To view Parent/Child relationships:

1 In the FeatureManager design tree or in the graphics area, right-mouse click the feature whose relationships you want to see.

2 Select **Parent/Child** from the menu. You can only *view* the relationships; you cannot edit them.
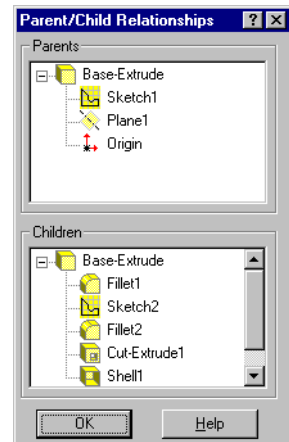
Feature Handles

Extruded and revolved features have handles that allow you to move, rotate, and resize the features quickly.

To display and use the feature handles:

1   Click the **Move/size feature** tool 🔷 on the Features toolbar. (If this button is not displayed on your toolbar, click **View, Toolbars, Customize**, and follow the instructions on the **Commands** tab to add it to a toolbar.) This button stays enabled while you continue working in the part.

2   Click the feature you want to work with, either in the FeatureManager design tree or in the graphics area. Double-click the feature to display both the handles and the dimensions of the feature.

3   To rotate or resize a feature, drag the appropriate handle.

4   To move a feature, you can either:

   • Drag the feature to a new location. If the feature has locating dimensions or relations that restrict its movement, you are notified that the dimensions will be recalculated, and relations may be deleted. You have the choice to continue or not.

   • Hold the **Alt** key as you drag to maintain the dimensions and relations on the feature. The feature is allowed to move only within unspecified degrees of freedom.

5   When you are finished, click 🔷 again to disable the handle display.
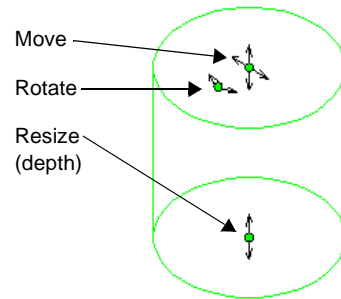
With the **Move/size feature** tool 🔷 disabled, you can still move a feature, by holding the **Shift** key and dragging one of its faces. The behavior is the same as using the move handles, and you can use the **Alt** key the same way.

When you *drag a feature with the move handle*, any child feature moves with its parent. When you **Shift**-drag, an attempt is made to maintain the relative position of any child feature, based on the child's dimensions and relations. However, in some cases (for example, when the child feature is underdefined, or dimensioned to something other than its parent), the child may not move with the parent as expected.

To move more than one feature at a time, press the **Ctrl** key as you select the features. This is useful in the case of underdefined child features.

Copying Features

To copy a feature:

1 Press the **Ctrl** key, then drag and drop the feature where needed. You can drop the feature on the same face, or on a different one. To copy more than one feature at a time, press the **Ctrl** key as you select the features.

Locating dimensions are left dangling, and the feature and sketch are marked with a red exclamation point indicating rebuild errors.

2 Either delete or re-attach the dangling dimensions. Edit the sketch, select the dangling dimension, and drag the red handle to a new attachment point. As you drag, the ⊘ cursor is displayed until you are on an edge or vertex that is suitable to re-attach the dimension.

To copy features from one part to another part:

Tile the windows, then drag and drop features from one window to another, or use the **Copy** 🖺 and **Paste** 🖺 tools on the Standard toolbar.

Equations

You can use equations to define mathematical relations between parameters or dimensions. Equations can include sketch dimensions and feature parameters (such as the depth of an extruded feature, or the instance count in a pattern). Equations can also be used to relate the dimensions of different parts in an assembly, or the values (distances or angles) used in mates.

To add an equation:

1 Click **Tools, Equations**, and in the **Equations** dialog box, click **Add**.

2 Enter the equation by clicking the dimensions or parameters in the model, and the numbers and operators in the calculator keypad, either in the **New Equation** dialog box, or on the keyboard.

Equations are solved left to right (the expression on the left is driven by the expression on the right), *in the order in which they appear in the list.* Therefore, the order in which you enter equations can affect the resulting part.

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

3 Click **OK**. The equation is listed in the **Equations** dialog box, and the resulting value is displayed in the **Evaluates To** column.

4 If you want to add another equation, click **Add** again; otherwise, click **OK** to close the **Equations** dialog box. Notice the **Equations** folder **E** in the FeatureManager design tree.

5 Click **Rebuild** **⊟** to apply any changes caused by solving the equation.

To add more equations:

1 Click **Tools, Equations, Add,** or right-mouse click the **Equations** folder **E** , and select **Add equation**.

2 Repeat Steps 2 through 5 above.

To edit an equation:

1 Click **Tools, Equations, Edit All,** or right-mouse click the **Equations** folder **E** , and select **Edit equation**.

2 In the **Edit Equations** dialog box, edit the text of the equations as needed, and click **OK**. The revised equation is displayed in the **Equations** dialog box.

3 Click **OK**, and **Rebuild** **⊟** the model.

To delete an equation:

1 Click **Tools, Equations,** or right-mouse click the **Equations** folder **E** , and select **Delete equation**.

2 Click the equation number or the status marker (see the next section) for the equation to delete, and click the **Delete** button in the **Equations** dialog box.

3 Click **OK**, and **Rebuild** **⊟** the model.

Equation Status

When you delete a feature or a dimension that is used in an equation, the system asks you if you want to delete the equation also. Because the order of solving equations is important, you may prefer to keep the equation, and edit it later. (You can cut, copy and paste equations in the **Edit Equations** dialog box, if necessary, to modify the order.)

If you choose to keep the equation, it has **Not Solved** status, and is marked with a red exclamation point **!** in the **Equations** dialog box. Equations with **Solved** status are marked with green checkmark **✓**.

Measuring Sizes and Distances

You can measure the size of items, and the distance and angle between lines, points, surfaces and planes in sketches, part or assembly models, or drawings.

To measure the distance between surfaces, edges, or vertices:

1 Click **Tools, Measure**.To keep the dialog in place while you are working, click the push pin icon ![pin].

2 Click the entities to measure. (You could also pre-select entities by **Ctrl**-clicking them before opening the dialog.)

3 In the **Projection On** area, select **Screen** to measure the projection to the screen; click **Plane/Face** to measure the projection to a selected plane or planar face.

Measurement between these points

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

4 New measurements update dynamically when you change selections. To select different items to measure:

 • Click a selected item again to deselect it, then click a new item.

 • Click an item in the **Selected Items** list and press the **Delete** key.

 • Right-click in the graphics area, and select **Clear Selections** to start again.

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

Dependency Editing

The **Dependency Editing Toolbar** contains tools to rebuild the model, and to suppress and unsuppress features of the model.

To display the Dependency Editing Toolbar:

Select **View**, **Toolbars**, **Dependency Editing**, or right-mouse click in any toolbar area, and select **Dependency Editing** from the menu.

Suppress/Unsuppress

You can suppress a feature so you can work on the model with the selected feature temporarily removed from the model. Suppressing a feature not only removes it from the display, but also from any calculations in which it may be involved. Regeneration of complex models occurs faster when features with a large amount of detail are suppressed.

Suppression is governed by the parent/child relationships of the features. See **Parent/Child Relationships** on page 5-5 for more information. Examine the illustrations below:

❑ In the first example, the **Tooth Cut** feature (the parent) and the **Tooth Copies** feature (the child, a circular pattern of **Tooth Cut** features) are unsuppressed.

❑ In the second example, **Tooth Cut** (the parent) is suppressed, causing **Tooth Copies** (the child) to be suppressed also (the child is *dependent* on the parent).

❑ In the third example, unsuppressing **Tooth Cut** (the parent) does not automatically unsuppress **Tooth Copies** (the child). However, unsuppressing a child feature causes its parent feature to be unsuppressed.

Tooth Cut	Unsuppressed	Suppressed	Unsuppressed
Tooth Copies	Unsuppressed	Suppressed	Suppressed

To suppress a feature:

   1 Select the feature(s) in the FeatureManager tree.

   2 Click **Edit, Suppress**, or 🔳 on the Dependency Editing toolbar.

   The selected feature is removed from the model (but *not* deleted). The feature disappears from the model view and is grayed out on the FeatureManager design tree.

To unsuppress a previously-suppressed feature:

   1 Select the suppressed feature on the FeatureManager tree.

   2 Click **Edit, Unsuppress**, or 🔳 on the Dependency Editing toolbar.

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

To unsuppress a previously-suppressed feature and its dependents:

   1 Select the suppressed feature on the FeatureManager design tree.

   2 Click **Edit, Unsuppress with Dependents**, or 🔳 on the Dependency Editing toolbar.

   - or -

   1 Select the child feature.

   2 Click **Edit, Unsuppress** 🔳 .

   The selected feature and any features that are dependent on it are returned to the model.

To suppress a feature using Properties:

   1 In the FeatureManager design tree, right-click a feature, and select **Properties**.

   2 In the **Feature Properties** dialog box, click **Suppressed**.

   3 Click **OK**.

   4 To unsuppress the feature, repeat the procedure and click **Suppressed** again (*deselect* it).

Rollback

Rollback reverts the model to the state it was in before a selected feature was created. When a part is rolled back, new features are inserted at the point in the FeatureManager design tree where you rolled back, instead of at the bottom of the tree. Use this function to change a part at an earlier point of its development.

To rollback with the rollback bar:

1 Place the cursor over the rollback bar in the FeatureManager design tree. The cursor changes to a hand.

2 Click to select the rollback bar. The bar changes color from yellow to blue.

3 Drag the rollback bar up the FeatureManager design tree until it is above the feature you want rolled back,

    - or -

Use the up and down arrow keys on the keyboard to move the rollback bar up or down. (Check **Arrow key navigation** in **Tools**, **Options**, **General** to enable this functionality.)

Note: The rolled back icons are grey, and cannot be selected.
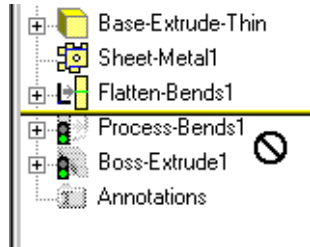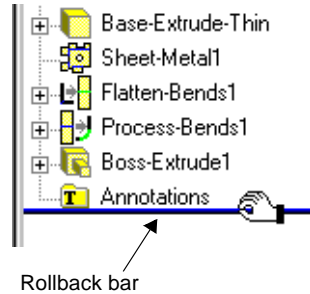
To roll forward:

Move the rollback bar down the FeatureManager design tree. You can:

• Roll forward all the way by dragging to the bottom of the tree, or a few steps at a time, by dragging the rollback bar to the desired location.

• Roll forward one feature at a time. Select the rollback bar, then use the up and down arrow keys to walk through the regeneration of the features one by one. (Check **Arrow key navigation** in **Tools**, **Options**, **General** to enable this functionality.)

Derived Parts

You can create a new part directly from another part. The new part, called a *derived part*, has the original part as its first feature, and it is linked to that part by means of an *external reference*. This means that any changes you make to the original part will be reflected in the derived part.

When a part has an external reference, its name in the FeatureManager design tree is followed by an arrow **->**. To see the name, location, and status of the externally referenced part, right-mouse click the derived part, and select **List External References.**

You can *add* features to a derived part. You cannot delete or modify the features of the original part in the derived part document.

You can only edit the original part in its own window. To edit the original part, right-mouse click the derived part, and select **Open *partname*.sldprt**. Any changes you make are applied to the original part. In addition, the changes are reflected in any derived parts that have an external reference to the original part.

To create a derived part from another part document:

1 Open a new part document.

2 Click **Insert, Base Part.**

3 Browse to the original part file (**.sldprt**) you want to use as an external reference, and click **Open** (or double-click the part file).

   The derived part is opened in a new part window. Notice the arrow after the name, indicating an external reference to the original part.

To create a derived part from an assembly component:

1 In an assembly document, select a component.

2 Click **File, Derive Component Part**.

   The derived part is opened in a new part window.

   This method is especially useful when the assembly component has been edited within the assembly. For example, you can create a cavity in a mold assembly, then derive the pieces of the mold. The derived parts maintain the necessary references to the cavity component in the assembly. See **Creating Molds** on page 7-15 for more information.

Working with Part Configurations

Part configurations allow you to create versions of a part with certain features suppressed or with different feature parameter values. You can create several configurations of a part in a single document. When you use the part in an assembly, you can choose which configuration to use.

Creating a Part Configuration

You can 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 tab at the bottom of the FeatureManager design tree to change to the Configuration Manager.

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

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

4   Click **Advanced** to specify how a part configuration is referenced when it is used in a Bill of Materials. Under **Part No. displayed when used in a Bill of Materials**, select one of the following:

   • **Use document name.** Specifies that the part name is used.

   • **Use name below.** When selected, a text box opens, containing a name to be used in the Bill of Materials. The configuration name is filled in by default. Alternatively, you may enter a name (such as a part number) of your own choosing.

5   Click **OK**. The new configuration name appears in the tree.

6   Return to the FeatureManager design tree view by clicking the FeatureManager tab.

7   Suppress or unsuppress features to create the desired configuration.
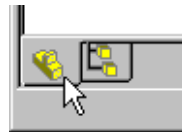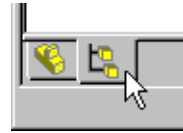
Viewing a Part Configuration

You can switch between the various part configurations that you have created.

To switch to a different configuration:

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

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

   - or -
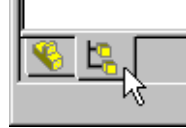
Double-click the configuration name.

A confirmation message box appears, notifying you that some configurations may take time to display. Click **OK** to continue; click **Cancel** to abort the action. If you do not want the message to display again during this session, click **Don't ask for confirmation again during this session**.

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

Editing a Configuration

To edit a part configuration:

1 View the desired configuration.

2 Change to the FeatureManager view, and change the suppression as needed.

To edit the configuration properties:

1 Right-click the configuration name, and select **Properties**. The **Configuration Properties** dialog box appears.

2 Edit the name, 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 you open a part file with more than one configuration, you can specify which configuration to display. Otherwise, the part is opened in the configuration in use when it was last saved.

To open a configuration of a part:

1 Click **File**, **Open**.

2 Navigate to the desired part, select the **Configure** check box, and click **OK**.

The **Configure Document** dialog box appears, with a list of available configurations.

3 Select the desired configuration.

4 Click **OK**. The part opens in the selected configuration.

Deleting a Configuration

To delete a part configuration:

1 In the Configuration Manager, click the name of the configuration you want to delete.

2 Press the **Delete** key (or click **Edit, Delete**), and click **Yes** to confirm.

Note: Deleting a configuration *does not* delete any features. You cannot delete the configuration that is currently active.

Design Table

A design table allows you to build multiple configurations of parts by 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.

To use design tables, you must have Microsoft Excel 97 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.

2 In the first *column* (row header cells **A2, A3,** etc.), enter the names of the design configurations that you want to create.

3 In the first *row*, (column header cells **B1, C1,** etc.), enter the full names of the dimensions or the features that you want to control. There are two ways to do this:

- Copy and paste (using **Ctrl-C** and **Ctrl-V**) dimension names from the **Properties** dialog box into the appropriate cells in the spreadsheet. To copy the name of a dimension, right-click the dimension, and select **Properties**. Select the name in the **Full Name** box, then copy and paste.

  You can also use the **Properties** command to assign more meaningful names to dimensions. See **Dimension Properties** on page 9-11.

- Type the names into the spreadsheet. To display the dimension names for all the dimensions, click **Tools, Options,** and on the **General** tab, under **Model,** click **Show dimension names**.

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

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

	D1@Base-Extrude	D2@Cut-Extrude1
rev 1	45	88
rev 2	48	94

You also can use a design table to suppress and unsuppress features. To unsuppress (include) a feature, put the feature name in the first row (as a column header) and enter **yes** in the appropriate cell. To suppress (remove) the feature, leave the related cell *blank*. For example, the following design table suppresses the feature **Cut-Extrude1** in the configuration named **rev 2**:

	D1@Base-Extrude	Cut-Extrude1
rev 1	45	yes
rev 2	48	

5 Save the spreadsheet.

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

7 Click **Insert, Design Table**. In the **Open** dialog box, navigate to the spreadsheet file and click **OK**.

The spreadsheet appears in the part document, the Excel menus and toolbars are displayed, and a message is posted, indicating the names of any new configurations that were generated by the design table.

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

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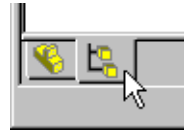
To edit a design table:

1   Click **Edit, Design Table**. The table appears, and the Excel menus and toolbars are displayed.

2   Edit the table. You can change the dimension values in the cells, add rows for additional configurations, or add columns to control additional dimensions.

3   Click anywhere outside of the design table to close it.

To delete a design table:

1   Click **Edit, Delete Design Table**.

2   Confirm the deletion and click **OK**.

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

Sending Part Documents

You can send a part document to another system using electronic mail. This function uses the electronic mail application installed on your system.

To mail a part document to another computer:

1   With a part document open, click **File, Send To**.

2   Enter your mail password, the mailing address of the recipient, and any other information requested by your mail application.

The open document is attached to the mail message.

Custom File Properties

You can use *custom file properties* to add more information to a part or subassembly document. These properties can be used:

- In a bill of materials (see **Bill of Materials** on page 9-28)

- As advanced selection criteria for managing assembly configurations (see **Component Selection by Properties** on page 6-31)

- As text in notes (see **Note** on page 9-18)

To add custom properties to a part or subassembly document:

1 Click **File, Properties**, then click the **Custom** tab.

2 Enter a **Name** for the property, or choose one from the list.

3 Select the **Type** of value (text, number, etc.) the custom property will use.

4 Enter a **Value** for the custom property that is compatible with the selection in the **Type** box.

5 Click **Add.** The **Properties** box displays the name, value and type of the custom properties.

To modify custom properties:

1 In the **Name** box, click the property that you want to change.

2 Edit the **Type** or **Value** as needed.

3 Click **Modify**.

To delete custom properties:

In the **Properties** box, click the property that you want to delete, then click **Delete**.

Displaying Annotations in Parts

You can add annotations to your model to further clarify the information the part document provides. You can add much of the necessary detailing of your models in the part document itself. This includes dimensions, notes, symbols, etc. Then you can import the dimensions and annotations from the model into a drawing.

See Chapter 9, *Detailing*, for more information about dimensions and annotations.

You can control the display of annotations in the part window using right-mouse menu selections on the **Annotations** folder in the FeatureManager design tree.

To select the types of annotations to display:

1   In the FeatureManager design tree, right-mouse click **Annotations** and select **Details**. The **Annotation Properties** dialog box appears.

2   Under **Display filter**, select the annotation types to display (**Notes**, **Datums**, etc.), or click **Display All Types** to show all types of annotations available for the part.

3   Change the values in the **Text Scale** boxes to modify the scale of the text used in the annotation.

4   Select from the following options:

   • **Always display text at the same size.** When checked, annotations and dimensions are displayed at the same size regardless of zoom.

   • **Display items only in the view in which they are created.** When checked, any annotation is only displayed when the part is in the same view orientation as when the annotation was inserted. Rotating the part or selecting a different view orientation removes the annotation from the display.

   • **Display Annotations.** When checked, all annotation types that are selected in the display filter are displayed.

5   Click **OK** to accept your changes; click **Cancel** to exit the dialog without saving the changes.

To globally toggle the display of annotations:

Right-mouse click **Annotations** and select **Display Annotations**. All annotation types that are selected in the **Annotation Properties** dialog are displayed. This has the same effect as checking the **Display Annotations** box in the **Annotation Properties** dialog box.

To toggle the display of feature dimensions:

Right-mouse click **Annotations** and select **Show Feature Dimensions.** This has the same effect as checking the **Feature Dimensions** box in the **Display filter** of the **Annotation Properties** dialog box.

To toggle the display of reference dimensions:

Right-mouse click **Annotations** and select **Show Reference Dimensions.** This has the same effect as checking the **Reference Dimensions** box in the **Display filter** of the **Annotation Properties** dialog box.

Section View of a Model

You can display a view of the model cut through the part or assembly. You can use any combination of standard planes, reference planes, and planar model faces as section planes.

You may find it useful to view both the section view and the complete view of the model at the same time by splitting your screen into two windows. (Drag the split controls on the bottom or right side of the window frame.)

To display a section view of the model:

1   Click **View, Display, Section View,** or click 🔲 on the View toolbar.

2   Click on one or more planes in the FeatureManager design tree, or click one or more planar faces on the model.

Alternatively, you can pre-select planes or planar faces before clicking **View, Display, Section View**. If nothing is selected, **Plane1** is used by default. To pre-select multiple planes or faces, hold the **Ctrl** key as you select.

The selected planes and faces are listed in the **Section Plane(s)/Face(s)** box.

3   Click a plane or face in the **Section Plane(s)/Face(s)** box, then enter an offset value (or use the spinbox arrows to change the value) in the **Section Position** box to move the section plane to the desired position for the cut.

4   Click **Flip the Side to View** to reverse the viewing direction of the section cut.

5   Select a **Preview/Display** mode:

- When **Preview** is checked, the section view is updated whenever any item in the dialog box is changed. If you need to adjust more than one setting, especially with a large part or assembly, this option may not be efficient.

- When **Preview** is not checked, click the **Display** button to update the section view. You can make multiple changes to the settings, then click **Display** again to update the view with all the changes at once.

6   Click **OK**.
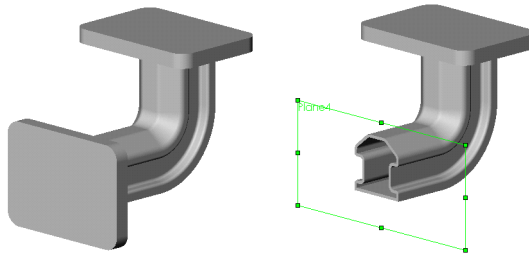
To use the viewing plane to cut a section view:

Instead of model faces and planes, you can use a *viewing plane* to cut the section view. The viewing plane is an invisible plane that is parallel to the screen, and normal to your line of sight. The initial position for this plane is the center of the part body bounding box. As you rotate the part, the angle of the viewing plane relative to the part body changes, cutting the part at a different angle.

To use the viewing plane to make a section cut, click **Use Viewing Plane** in the **Section View** dialog box.

You can offset the **Section Position** of the viewing plane.The **Flip the Side to View** option is not available when you use the viewing plane.

To change the section view:

Click **View, Modify, Section View** and change the settings in the **Edit Section View** dialog box and/or select a different plane.

To return to a normal view of the model:

Click **View, Display, Section View** to remove the checkmark beside the option in the menu.

Mass Properties and Section Properties

You can display the density, mass, volume, surface area, center of mass, inertia tensor, and principal axes of inertia of a part or assembly model.

To display mass properties:

1 Click **Tools, Mass Properties.**

   To display the mass properties of an assembly component, select the component first.

   If no assembly component is selected, the mass properties for the entire assembly are calculated.

   The results are displayed in the **Mass Properties** dialog box, and the principal axes and center of mass are displayed graphically on the model.
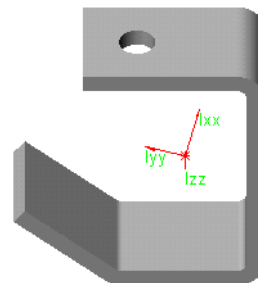
2 Click **Print** if you want to print the results directly from this dialog box.

3 Click **Copy** if you want to copy the information to the clipboard, then paste it into another document.

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

Section Properties

You can display certain mass properties for a planar face, sketch, or section face in a drawing view. The properties reported depend on the entity selected.

To display section properties:

1 Select a planar model face in any document, or the crosshatch section face in a section view in a drawing, or select a sketch (either click the sketch in the FeatureManager design tree, or right-click a feature, and select **Edit Sketch**).

   Note: Section properties of sketches can only be calculated for *closed* profiles.

2 Click **Tools, Section Properties.** The results are displayed in the **Sectional Mass Properties** dialog box, and the principal axes and center of mass are displayed graphically on the model.

3 Click **Print** if you want to print the results directly from this dialog box.

4 Click **Copy** if you want to copy the information to the clipboard, then paste it into another document.

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

Part Color and Lighting

You can change the color of a part and the properties that define the way the material reacts to light. This section describes changing the color of the *whole part*; for information about changing the color of an individual feature, see **Feature Properties** on page 5-4.

To change the color of a part:

1 Click **Tools, Options,** and select the **Color** tab.

2 Under **System**, select **Shading**, then click the **Edit** button.

3 Choose a color from the palette, or click **Define Custom Colors** to specify a custom color.

4 Click **OK** to close the **Color** dialog.

5 Make sure that **Apply To:** is set to **Active Document**, then click **OK** to close the **Options** dialog.

To change the way a part reacts to light:

1 Click **Tools, Options,** and select the **Color** tab.

2 Under **System**, select **Shading**, then click the **Advanced** button.

3 Use the slider controls to change the display properties.

- **Ambient**. Light reflected and scattered by other objects

- **Diffuse**. Light scattered equally in all directions on the surface

- **Specularity**. Ability of a surface to exhibit highlights

- **Shininess**. A glossy, reflective surface or a dull, matte surface

- **Transparency**. Ability to pass light through the surface

- **Emission**. Ability to project light from the surface

4 Click **OK** when done.

Light Source Types

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

There are three types of light source:

❑ **Ambient Light.** Light that illuminates the part evenly from all directions. For example, the light in a room with white walls has a high degree of ambient light because the light reflects off the walls and other objects; a spot-lighted area has little ambient light because the light is focused in a narrow beam.

❑ **Directional Light.** Light that comes from a source that is infinitely far away from the surface. This is a columnated light source consisting of parallel rays arriving from a single direction. (The sun is a real-world example of directional light.) The direction of the light source is apparent to the viewer.

❑ **Spot Light.** A restricted, focused, point-source light that emits a cone-shaped beam that is brightest at its center.

To adjust the lighting in a shaded view:

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

The **Light Sources** dialog box appears, with a tab for each light source. By default, there is an **Ambient Light** source and one **Directional Light** source.
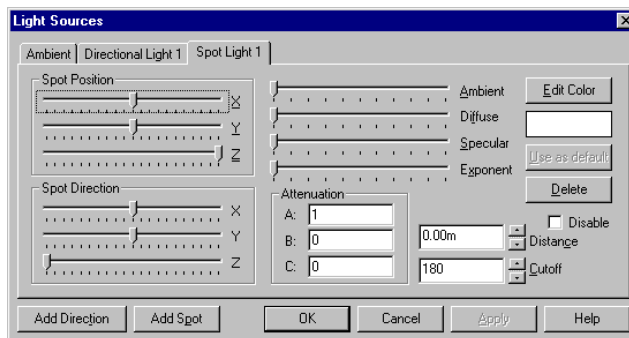
2 On the tab for each light source, select the color, direction, intensity, and other characteristics of the light source, as described below. Experiment until you get the results that you want.

3 Choose from these options as needed:

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

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

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

- **Disable.** Turns off the light source defined on the current page. To turn the light source back on, deselect this checkbox.

4 To add more light sources, click the **Add Direction** or **Add Spot** buttons at the bottom of the dialog box. You may use a maximum of eight light sources in addition to the ambient light source. For each new light source, click the tab and specify the characteristics of that light source.

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

Light Source Characteristics

The light source characteristics vary depending on the type of light source. Move the slider to the right to increase the effect of each light source characteristic.

Ambient Light Characteristics

- **Ambient.** Light that has been scattered by other objects in the environment.

Directional Light Characteristics

- **Position.** Adjust the sliders to specify the position of the directional light source relative to the center of the scene.

- **Ambient.** Light that has been scattered by other objects in the environment.

- **Diffuse**. Light that comes from one direction, but once it strikes the surface the light scatters equally in all directions so it appears equally bright no matter where the observing eye is located.

- **Specular**. Light that comes from one direction and bounces off the surface. Specularity indicates the ability of a surface to exhibit highlights. A matte surface has no specularity; a glossy surface has high specularity.

Spot Light Characteristics

These additional characteristics are unique to the **Spot Light**:

- **Spot Position.** Specifies the position from which the light arrives. Adjust the sliders to specify the position of the spot light relative to the center of the scene.

- **Spot Direction**. Specifies where the light is pointing in the scene. Adjust the sliders to specify where the spot light strikes the scene.

    Think of these controls in the same terms as a spotlight in a theater. The position of the light is usually above and behind the audience; the direction the light points is toward a specific part of the stage.

- **Exponent.** The property that results in a more or less concentrated beam of light. A higher spot exponent results in a more focused light source. Move the slider to the right to increase the light concentration.

- **Attenuation.** The property that decreases the intensity of light as the distance from the light source increases. The three boxes (**a**, **b**, and **c**) hold values that are used in an equation to arrive at an attenuation factor.

$$\text{attenuation factor} = \frac{1}{a + b * d + c * d^2}$$

- **Distance.** The distance from the light source to the surface of the model. Use the spinbox arrows to modify the distance. (This value is represented by **d** in the attenuation factor equation.)

- **Cutoff**. The property that describes the spread of the beam, restricting the shape of the light cone. The maximum value is 180; reduce the value to reduce the spread of the light cone.

Curvature

You can display the model with the surfaces rendered in different colors according to their local radius of curvature. Curvature is measured in units equal to 1 divided by the radius of the curve, in meters. By default, the greatest radius of curvature shown is 1000.000, and the smallest radius is 1.000.

As the value increases, the corresponding color changes from black (1.000), through blue, green, and red (1000.00). You specify the color gradations associated with the radii of curvature on the **Color** tab of the **Options** dialog box.

To render curved surfaces in colors:

Click **View, Display, Curvature.** The curvature of the model is displayed in color as long as the menu item remains checked. When you point to a model surface, the curvature value is shown beside the cursor.

To remove the color, click **View, Display, Curvature** again.

To render an individual model face in colors:

Right-mouse click the model face and select **Face Curvature**. The curvature of the selected face is displayed in color as long as the menu item remains checked. When you point to a model surface, the curvature value is shown beside the cursor.

To remove the color, right-mouse click the model face and click **Face Curvature** again.

To set the curvature radius values:

1 Click **Tools, Options**, and select the **Color** tab.

2 Click the **Curvature** button.

3 Select the radius value edit boxes that you want to change and enter new values. You can modify the values at five intervals on the color scale; values are interpolated for the intermediate colors in the range.

4 Click the **Apply** button to see how the color scale changes in response to the new radius values.

5 Click **OK** to save the new settings.

6

Working with Assemblies

You can build complex assemblies consisting of many parts and sub-assemblies.

This chapter describes the basics of creating an assembly from parts you have already built, and general information about working with assemblies of any kind. The following topics are discussed:

❑ Design methods

❑ Creating an assembly

❑ Assembly mating

❑ Exploding an assembly view

❑ Simplifying large assemblies

❑ Using assembly configurations

❑ Using advanced component selection methods

❑ Customizing the appearance of an assembly

❑ Mailing an assembly file

Design Methods

You can create an assembly using bottom-up design, top-down design, or a combination of both methods. This chapter explains the bottom-up method. See also the next chapter, *Working with Parts within an Assembly,* for information about the top-down method.

Bottom-Up Design

Bottom-up design is the traditional method. In bottom-up design, you create parts, insert them into an assembly, and mate them as required by your design. Bottom-up design is the preferred technique when you are using previously constructed, off-the-shelf parts.

1. Create parts

2. Create assembly

3. Insert and mate parts
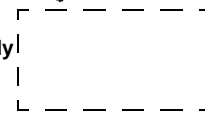
An advantage of bottom-up design is that because components are designed independently, their relationships and regeneration behavior are simpler than in top-down design. Working bottom-up allows you to focus on the individual parts, and it is a good method to use if you do not need to create references between parts.
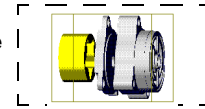
Top-down Design

Top-down design is different because you start your work in the assembly. You can use the geometry of one component to help define other components, or to create cut or hole features that are added only after the components are assembled. Or you can start with a layout sketch, define fixed component locations, planes, etc., then design the parts referencing these definitions.

1. Create assembly

2. Create parts in place
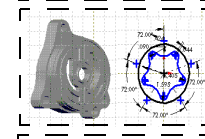
3. Create (or insert) other parts

For example, you can insert a part in an assembly, then build a fixture based on this part. Working top-down, creating the fixture *in context*, allows you to reference model geometry, so you can control the dimensions of the fixture by creating geometric relations to the original part. That way, if you change a dimension of the part, the fixture automatically is updated.
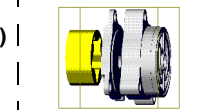
Understanding Assemblies

When a part is inserted into an assembly, it is referred to as a *component*. An assembly can be a *sub-assembly* of another assembly. Components are *linked* to the assembly file. Assembly files have the .**sldasm** extension.

The Assembly Window

This is a typical assembly window.
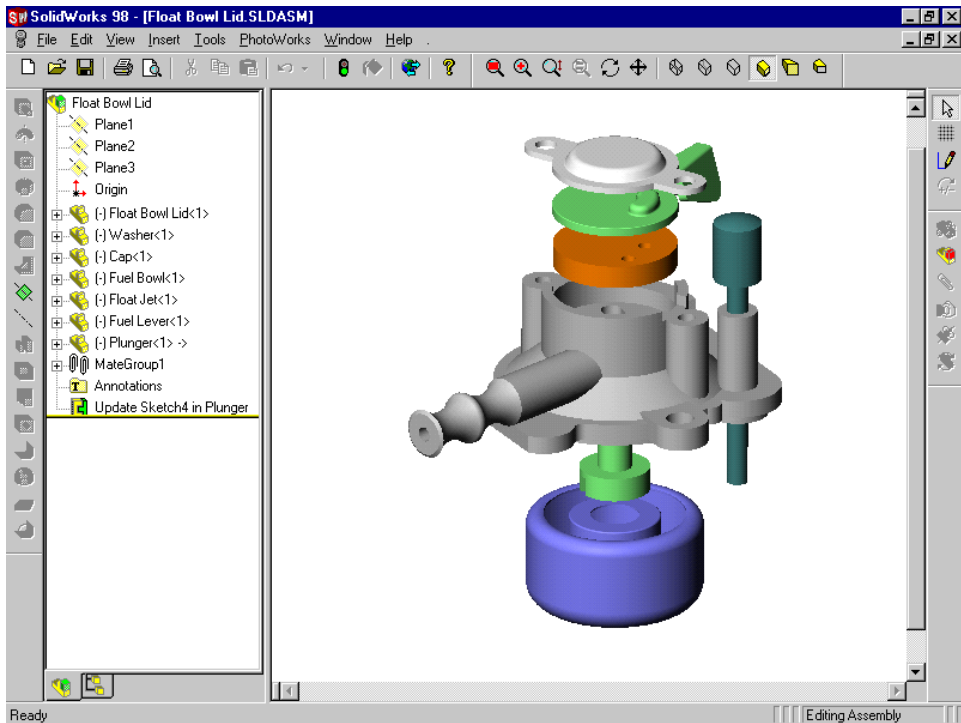
The FeatureManager design tree displays the names of the:

- Assembly, sub-assemblies and parts

- MateGroups and mating relations

- Assembly planes, origin, and annotations

- Assembly features (cuts or holes), and component patterns

- Part features built in the context of the assembly

The first item in the FeatureManager design tree is the name of the assembly. You can expand or collapse each sub-assembly or component to view its detail by clicking the ⊞ beside the component or sub-assembly name.

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

In the FeatureManager design tree, a part or sub-assembly name may have a prefix, that provides information about the state of its relationships to other components. The prefixes are:

(–)   underdefined

(+)   overdefined

(f)   fixed

(?)   not solved

The absence of a prefix indicates that the component's position is fully defined.

The Assembly Toolbar

The Assembly toolbar gives you quick access to these frequently used assembly tools.

Move Component

Rotate Component Around Centerpoint

Rotate Component Around Axis

Mate

Edit Part

Hide/Show Component

For information about moving and rotating components in an assembly, see **Positioning Components in an Assembly** on page 6-11.

For information about mating components, see **Assembly Mating** on page 6-14.

For information about editing a part in an assembly, see **Editing Parts in the Assembly Context** on page 7-4.

For information about hiding and showing components, see **Simplifying Large Assemblies** on page 6-23.

The FeatureManager Design Tree in an Assembly

There are times when you want to focus on the structure or hierarchy of the design rather than the details of the sketches and features. You can view an assembly's hierarchy in the FeatureManager design tree.

You may want to focus on the design of the assembly without all of the features of the components. By changing the FeatureManager design tree's display mode, you can view the assembly reference planes, components, sub-assemblies, assembly features, and mating relationships. The image of the assembly, including all its components (other than any suppressed or hidden ones), still appears in the SolidWorks window.

Each of these ways of viewing the assembly affects only the level of detail displayed in the FeatureManager design tree. The assembly itself is not affected.

To display the hierarchy or feature detail of an assembly:

❑ To display the hierarchy, right-mouse click the assembly name in the FeatureManager design tree, and select **Show Hierarchy Only**.

The FeatureManager design tree displays only the parts and sub-assemblies, but no lower level detail.

❑ To display the detail again, repeat the procedure, selecting **Show Feature Detail**. To display the detail for a single component, double-click the component in the FeatureManager design tree.

To view an assembly by dependencies or by features:

❑ To display the dependencies, right-mouse click the assembly name in the FeatureManager design tree, and select **View Dependencies**, *or* click **View, FeatureManager Tree, By Dependencies**.

Dependent items (instead of features) are listed under each component. Dependent items include mates and component patterns.

❑ To display the features again, right-mouse click the assembly name in the FeatureManager design tree, and select **View Features**, *or* click **View, FeatureManager Tree, By Features**.

Creating an Assembly

An assembly can consist of parts you have previously designed as well as components that you create within the assembly. This section describes how to add, replace, copy, delete, verify, and save components in an assembly.

Adding Components to an Assembly

When you place a component in an assembly, the part file is linked to the assembly file. The component appears in the assembly; however, the data for the component remains in the source part file. Any changes you make to the part file will update the assembly.

There are several ways to add components to a new or existing assembly.

To add components by using the menu:

1   With an assembly open, click **Insert, Component, From File**. The **Open** dialog box appears.

2   Browse to the directory that contains the component (or sub-assembly) you want to insert into the assembly.

3   Click the component name, then click **Open**. If **Preview** is checked, you can see the selected component when you select it. You can also double-click the component (or sub-assembly) name to select it, and close the dialog box in one step. The cursor changes to a cross ✛.

4   Click the cursor in the area of the assembly window where you want to place the component.

To add components by dragging and dropping from an open 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   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, then release the mouse button.

To add another component instance by dragging and dropping:

1 With an assembly open, press the **Ctrl** key, then click a component in the FeatureManager design tree.

2 Drag the component into the graphics area, then release the mouse button.

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

Inferencing to the Assembly Origin

When you insert a component by dragging it from an open part window, or by clicking **Insert, Component, From File**, you can inference the assembly origin. Look for the cursor shown here.

When you inference the assembly origin:

- the component is fixed **(f)**

- the component origin is coincident with the assembly origin

- the planes of the component and the assembly are aligned

You can also drag a part from an open part window and drop it in the FeatureManager design tree to inference the origin.

Adding Patterns of Components

You can add a pattern of components in an assembly based on a feature pattern of an existing component. You can also define a pattern for placing components in an assembly in much the same way as you define a feature pattern in a part. Component patterns are *time-dependent*; they depend on an existing mating relation. See **Time-Dependent Features** on page 7-7 and **Adding a Component Pattern** on page 7-9 for more information.

Deleting a Component from an Assembly

To delete a component from an assembly:

1 Click the component in the display or in the FeatureManager design tree.

2 Press the **Delete** key, or click **Edit**, **Delete**.

3 Click **Yes** to confirm the deletion. The component and all its dependent items (mates, component patterns, explode steps, etc.) are removed.

Managing the Part Files of an Assembly

Verifying Which Part Files Are Used in an Assembly

You can check which part and sub-assembly files an assembly uses. This is useful if you have several versions of a part file.

To list the part files:

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 sub-assembly 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. You can also use this method to back up the assembly and all the referenced files.

To copy the part files:

1 Click **File**, **Find References**.

2 In the **Search Results** dialog box, click **Copy Files**.

3 In the **Choose Directory** dialog box, browse to the destination directory.

4 Click **OK**. The assembly file and all the referenced files are copied to the new directory. This operation does not change the open assembly or any of its components; it simply makes a copy of all the files.

Saving an Assembly and Its Parts

When you save an assembly, by default, you save the assembly and all referenced parts which have been changed.

If you do not want to overwrite a part that you have modified and used in an assembly, save the part with a different name (**Save As**) and replace the part in your assembly.

You can also use **Save As Copy** to save the document to a new filename without replacing the name in the active session.

You can save selected assembly parts to a new location or filename by editing the referenced file locations.

To edit referenced file locations:

1   Click **File, Save As**, and click the **References** button.

2   Select the filename that you want to save with a new name or path, or to change the path for all the entries on the list, click **Select All**.

3   Edit the path or filename:

   • Click-pause-click the filename, then type a new path and/or filename.

   • Click a filename, then click **Browse** to change the path.

4   Click **Replace** and enter the text strings to find and replace. Click **Match case** if the case is significant in the search.

5   Click **OK** to make the changes and exit the dialog. Click **Cancel** to exit without saving your changes.

Note: When you save an assembly or its components this way, you change the reference itself. The part document referenced in the assembly is the one stored in the new pathname, not the original pathname.

Replacing a Part in an Assembly

You can replace a part in an assembly with another part. If the parts are similar, mates used in the original part are applied to the replacement part. See **Assembly Mating** on page 6-14 for more information about mates.

Preservation of the mates depends on the entities used in the mates being the same. Therefore, if you replace one part with another part created by deriving or copying, the entities used in the mates are likely to match (depending on the differences between the components).

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.
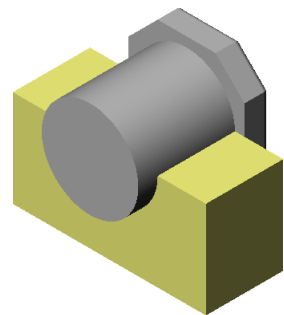
4   Click **Open**.

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. In this example, the assembly maintains the concentric mate between the cylindrical faces, and the distance mate between the flat faces.
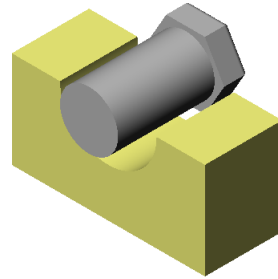
You can rename the corresponding edges and/or faces on a replacement component to match the edge/face names on the original part, to insure that the mates are preserved.

To name the entities used in a mate:

1   Right-click the entity, and select **Face Properties** (or **Edge Properties**).

2   Under **Entity Information**, enter a new **Name**, and click **OK**.

3   Repeat for the replacement part, using the same **Name**. Now when you replace the component, the mate can be preserved.

To reload an assembly component:

1   Right-mouse click the component you want to reload or replace.

2   Click **Open as read-only** if you do not plan to save any changes to the part file.

3   Select either **Reload** or **Replace**.

- **Reload** refreshes shared documents. For example, if you open a part file for read-only access and use that part while another user is making changes to the same part, you can reload the part to get the updated version.

- **Replace** lets you load a new file into the SolidWorks session and replace each open reference to the current file with a reference to the new file instead. Use the **Browse** button to locate the new file.

4   Click **OK** to accept the change, or click **Cancel**.

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, balloons, cosmetic threads, and geometric tolerances. See Chapter 9, *Detailing*, for complete information about creating and using annotations.

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:

❑ If the assembly is displayed in **Wireframe, Hidden In Gray,** or **Hidden Lines Removed** mode, the component is enclosed in a *bounding box*.

❑ If the assembly is displayed in **Shaded** mode, the component is displayed in the highlight color (green by default).
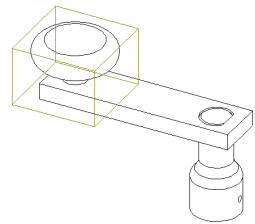
Fixing the Position of a Component

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.

It is recommended that at least one assembly component is either fixed, or mated to the assembly planes or origin. This gives a frame of reference for all other mates, and may prevent unexpected movement of components when mates are added.

To fix a component:

In the FeatureManager design tree, right-mouse click the component (part or sub-assembly) icon, and select **Fix**.

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

To float (unfix) a component:

In the FeatureManager design tree, right-mouse click the component (part or sub-assembly) icon, and select **Float**.

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

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 within the component's own mategroup are maintained.

To rotate a component freely:

1   Click the desired component in the graphics area or the FeatureManager design tree.

2   Click **Tools**, **Component**, **Rotate** or **Rotate Component Around Centerpoint** on the Assembly toolbar. You can only use this rotate tool on components that have not been mated.

3   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 Component Around Axis** on the Assembly toolbar.

4   Click and drag to rotate the component around the axis. Dragging the mouse left-to-right rotates about the axis in one direction; right-to-left rotates in the opposite direction.

Moving a Component

When you move a component, it retains any mating relationships (within its own mategroup) with other components. You cannot move a component whose position is fixed or fully defined.

To move a component:

1   Click 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*.

Measuring Distances Between Components

You can measure the size of items, and the distance and angle between lines, points, surfaces and planes in sketches, part or assembly models, or drawings. See **Measuring Sizes and Distances** on page 5-9 for more information.

Detecting Interference Between Components

In a complex assembly, it may be difficult to visually determine whether components interfere with each other. You can determine the interference between components and visually browse the resulting interference volumes.

To check for interference between components in an assembly:

1 Click **Tools**, **Interference Detection**.

2 Select two or more components in the assembly, or click the assembly icon (top level or sub-assembly) in the FeatureManager design tree.

3 Click **Treat coincidence as interference** if you want coincident entities (overlapping faces, edges, or vertices) to be reported as interferences. Otherwise, overlapping volumes are considered.

4 Click **Check**.

If there is interference, the **Interference Results** box lists the interference occurrences (one occurrence is reported for each pair of interfering components). When you click an item in the list, the related interference volume is highlighted in the graphics display area, and the names of the components involved are listed.
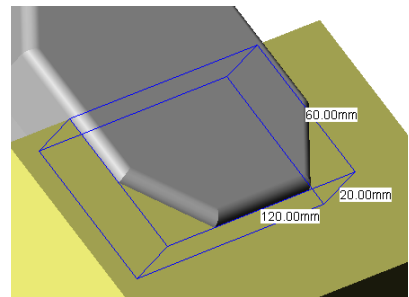
The volume of the interference is reported in the form of length, width, and height of the bounding box around the area of interference. These numbers are displayed on the graphic display of the components.

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

6 Click **OK** to dismiss the dialog box. When the dialog box is dismissed, the interference volumes are dismissed also.

Note: If detecting interference is important in your design work, check for interference each time you move or rotate a component.

Assembly Mating

Mating Relationships

Mating relationships let you precisely position the components with respect to each other in an assembly. They let you define how the components move/rotate with respect to other parts. By adding mating relationships successively, you can move the components into the desired positions.

Mating creates geometric relationships, such as coincident, perpendicular, tangent, and so on. Each mating relationship is valid for specific combinations of geometry. The following table shows the mating relationships that are supported between the various types of geometry.

	Plane	Cylinder	Line	Cone	Point
Plane (planar face or plane)	coincident distance parallel perpendicular angle				
Cylinder (cylindrical face)	tangent	tangent concentric			
Line (linear edge, axis, or sketch line)	coincident parallel perpendicular distance	coincident tangent concentric	coincident parallel perpendicular distance angle		
Cone	---	concentric	concentric	concentric	
Point (vertex or sketchpoint)	coincident distance	coincident concentric	coincident distance	concentric	coincident distance

Mategroups

When you create a new assembly, an empty mategroup is automatically created. This mategroup (which appears with a default name of **MateGroup1**) is shown in the FeatureManager design tree with a double paperclip icon.

When you create a mating relationship, the mating relationship icon (a single paperclip) and name appears in the FeatureManager design tree under the **MateGroup<n>** mategroup. Each assembly has at least one mategroup. Each mategroup consists of mates that are solved together.

For example, if you have two parts in an assembly, and add a concentric and a coincident mate between the parts, **MateGroup1** will contain mates called **Concentric1 (*part1, part2*)** and **Coincident1 (*part1, part2*)**.

☐ 📎📎 MateGroup1
   📎 Concentric1 (bracket<1>,Yoke_male<1>)
   📎 Coincident1 (bracket<1>,Yoke_male<1>)
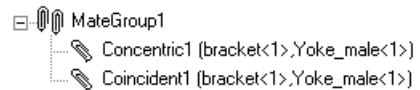
Assemblies can have multiple mategroups. Additional mategroups are automatically added when needed. This occurs when performing certain operations, such as adding a mate to a time-dependent feature in the assembly. See **Time-Dependent Features** on page 7-7 for more information.

You can rename mategroups and mates. Click-pause-click on the name in the FeatureManager design tree, type the new name, and press **Enter**.

Creating a Mating Relationship

To mate components in an assembly:

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

2 Select the desired faces, edges, vertices, or reference planes on the components. The selected entity and component names are listed in the **Items Selected** box.

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

   In the **Assembly Mating** dialog box, you can click the pushpin 📌 to keep the dialog box open. This allows you to apply multiple mates before closing the dialog box.
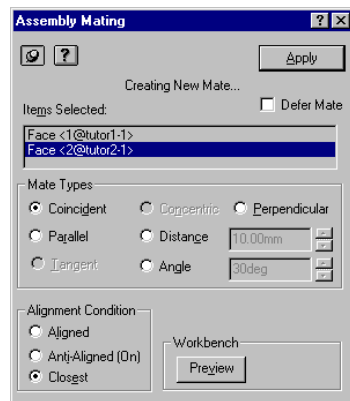
3 Select the desired **Mate Type**. (Only the mating types that are valid for the selected entities are available.)

4 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 are either aligned or anti-aligned, depending on which condition can be satisfied with the least part movement.

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

6 Click **Defer Mate** if you want to define several mates, then solve them all at once. Mates defined while this option is checked are solved when you either close the dialog box or uncheck the option.

7 If you are satisfied with the mate, click **Apply**. If you are not satisfied with the mate, click **Undo** or click a different alignment option and **Preview** again.

8 Click **Apply** when you are satisfied with the mate. When you close the **Assembly Mating** dialog box, any deferred mates are solved, and the components move into the specified positions.
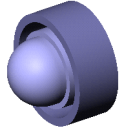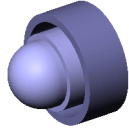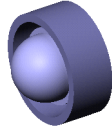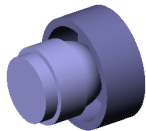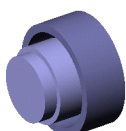
Modifying a Mating Relationship

You can change these characteristics of a mating relationship:

- **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

The examples in the table show the effect of the changing the alignment and flipping the direction for a distance mate. All the examples use the same face selections.

Selected faces
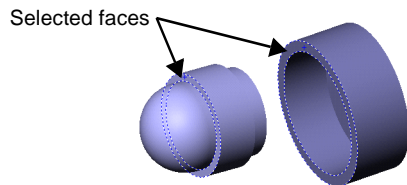
Offset Distance	Component Alignment	Flip Dimension To Other Side	Example
0 (coincident)	aligned	off	
20mm	aligned	off	
20mm	aligned	on	
20mm	anti-aligned	on	
20mm	anti-aligned	off	

To modify a mating relationship:

1   Right-click the desired mating relationship in the FeatureManager design tree to select it.

The related geometry in the graphics display is highlighted.

2   Select **Edit Definition** from the right mouse menu.

3   In the **Assembly Mating** dialog box, change the desired options.

4   To replace one component with another, click the entity to replace in the **Items Selected** box, press the **Delete** key, then select the new entity.

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

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

Deleting a Mating Relationship

You can delete mating relationships when necessary.

To delete a mating relationship:

1   Click the desired mating relationship in the FeatureManager design tree.

2   Press the **Delete** key, or click **Edit**, **Delete**.

3   Click **Yes** at the prompt to confirm the deletion.

Suppressing a Mating Relationship

You can suppress mating relationships to prevent them from being solved. This allows you to experiment with different types of mates without overdefining the assembly.

To suppress a mating relationship:

1   Right-click the mating relationship in the FeatureManager design tree, and select **Properties**.

2   Click **Suppressed**, and click **OK**.

   To unsuppress the mate again, repeat the process, and *deselect* **Suppressed**.

   You can also select one or more mates (hold the **Ctrl** key to select more than one), and click **Edit, Suppress** (or **Unsuppress**).

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 in 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 since the last **Apply**

   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.
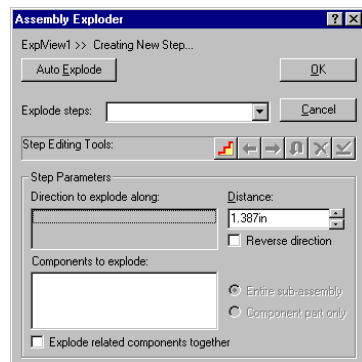
3 Click a component edge or face that is parallel to the direction you want to explode.

Direction to explode along

A preview arrow appears and a description of what you have selected appears in the **Direction to explode along** box.

Component to explode

If the preview arrow is pointing in the wrong direction, click **Reverse direction**.
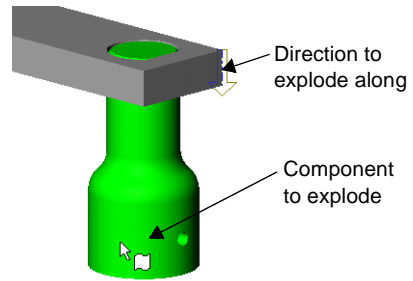
4 Click the component you want to explode, either in the FeatureManager design tree or in the graphics area.

The name of the component appears in the **Components to explode** box.

5 Click **Apply** ☑.

The component explodes from the assembly. Note the drag handles.

6 Drag the arrow-shaped handle to position the component as desired.

drag handle

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**.
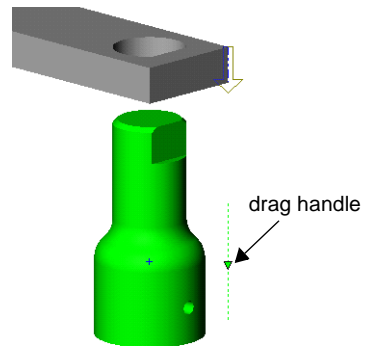
8 If you are satisfied with this exploded view, click **OK**.

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, right-click anywhere and select **Collapse**.
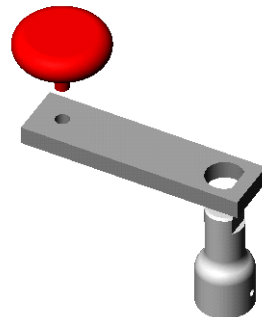
Exploding and Collapsing an Exploded View

To explode and collapse an exploded view:

1   Click the Configuration Manager tab at the bottom of the window.

2   Click the ⊞ beside the desired configuration, and beside the **ExplView** feature to see the **Explode Steps**.

3   Double-click the **ExplView** feature. The assembly explodes.

4   To collapse the exploded view, right-click anywhere and select **Collapse**.
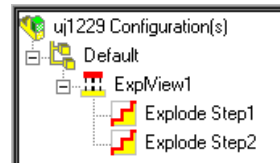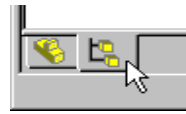
> **Note:** To change the name of an exploded view or explode step, click-pause-click the name, type the new name and press **Enter**.

Editing an Exploded View

To edit an exploded view (method 1):

1   In the Configuration Manager, expand the desired configuration, and expand the exploded view to see the steps,

> *- or -*

Right-click the **ExplView** feature for the desired configuration, and select **Edit Definition**.

2   Click the **Explode Step** feature you want to edit, or use the **Next Step** and **Previous Step** buttons to examine and edit each step in turn.

As you select each step, the drag handle appears and the components involved are highlighted (**Shaded** view mode) or enclosed in bounding boxes (**Wireframe, Hidden In Gray,** or **Hidden Lines Removed** view mode).

3   Edit the explode step until the component is positioned as desired.

4   Click **Apply** after editing each step.

5   Repeat for each step as needed, then click **OK** to finish the view.

To edit an exploded view (method 2):

1   Double-click the **ExplView** feature to explode the view.

2   Right-click the component you want to reposition in the graphics area, and select **Show Explode Steps**.

The drag handle(s) for the explode step(s) on the selected component are displayed.

3   Drag the component by the green drag handle to the new position.

4   Repeat as needed for each component, then collapse the assembly.

To edit an exploded view (method 3):

For complex assemblies, you may find it easier to edit the explode steps for a component by selecting the component from the FeatureManager design tree.

1   Double-click the **ExplView** feature to explode the view.

2   Switch from the Configuration Manager to the FeatureManager by clicking the FeatureManager tab at the bottom of the window.

3   Right-click the component to reposition, and select **Show Explode Steps**.

4   Proceed as described above.

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.

Simplifying Large Assemblies

Large assemblies can be complex, consisting of many components. Some assemblies have sub-assemblies 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, and you can switch between configurations as desired. For more information about configurations, see **Using Assembly Configurations** on page 6-26.

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 sub-assembly:

1   In the FeatureManager design tree or the graphics area, right-click the desired part or sub-assembly, and select **Component Properties**.

2   In the **Component Properties** dialog box, select the **Suppress** check box, and click **OK**.

The selected part or sub-assembly 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 sub-assembly, repeat this procedure, *deselecting* the **Suppress** check box.

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

You can also:

• Use the **Suppress** and **Unsuppress** icons on the Dependency Editing toolbar.

• Select one or more components (hold the **Ctrl** key to select more than one), and click **Edit, Suppress** (or **Unsuppress**).
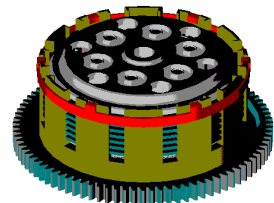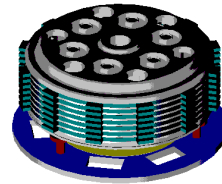
Hiding Components

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 sub-assembly:

1 In the FeatureManager design tree, right-click the desired part or sub-assembly, and select **Component Properties**.

2 In the **Component Properties** dialog box, deselect the **Show Model** check box, and click **OK**.

The selected part or sub-assembly 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 sub-assembly is turned off.

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

> **Note:** If you rebuild the assembly, the system still uses the undisplayed parts to generate the assembly.
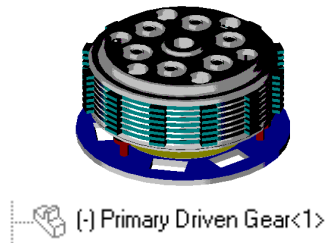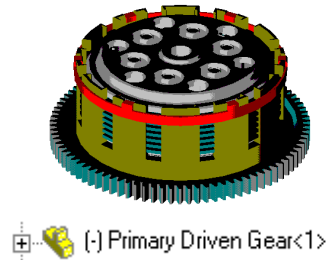
You can also:

- Use the **Hide/Show Component** icon on the Assembly toolbar to toggle the hidden and shown states of a component.

- Select one or more components (hold the **Ctrl** key to select more than one), and click **Edit, Hide Component** (or **Show Component**).

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, modify, and delete assembly configurations.

Creating an Assembly Configuration

You create a configuration that has suppressed/hidden parts or sub-assemblies, and name the configuration. You can then view the assembly with the suppressed/hidden parts by selecting the named configuration.

To create a configuration:

1 Click the Configuration Manager tab at the lower left of the window.

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

3 Enter a **Configuration Name**, 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 sub-assembly components. All sub-assembly child components will inherit the properties specified in the configuration of the sub-assembly.

- Click **Apply properties to ALL sub-assembly components** to apply the properties selected in Step 4 to all added components. For example, if you want all components of newly added sub-assemblies to be suppressed, regardless of the properties specified in the sub-assembly configuration, choose this option.
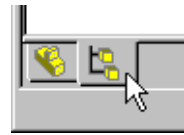
6 Click **Advanced** to specify how the assembly configuration is referenced when it is used in a Bill of Materials. Under **Part No. displayed when used in a Bill of Materials**, select one of the following:

- **Use the document name.** Specifies that the assembly name is used.

- **Use name below.** When selected, a text box opens, containing a name to be used in the Bill of Materials. The configuration name is filled in by default. Alternatively, you may enter a name (such as a part number) of you own choosing.

7 Click **Don't show child components in BOM when used as sub-assembly** if you *always* want the sub-assembly listed as a single item in the bill of materials. When unchecked, the child components may be listed in the bill of materials when certain options are selected. See **Bill of Materials** on page 9-28.

8 Click **OK**.

The new configuration name appears in the tree.

9 Return to the FeatureManager view by clicking the FeatureManager tab.

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

11 **Save** the assembly. The new named configuration is created.
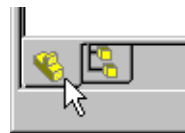
Viewing an Assembly Configuration

To switch to a different configuration:

1 Click the Configuration Manager tab.

2 Either double-click the name of the configuration you want to view, or right-mouse click the name and select **Show Configuration**.

A message appears, notifying you that some configurations may take time to display. Click **OK** to continue; click **Cancel** if you want to abort the action.

If you do not want the message to display again during this session, click **Don't ask for confirmation again during this session.**

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

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.

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, type the new name, and press **Enter**.

Deleting an Assembly Configuration

To delete a configuration:

   1   In the Configuration Manager, click the name of the configuration you want to delete.

   2   Press the **Delete** key (or click **Edit, Delete**), and click **Yes** to confirm.

   Note: You cannot delete the active configuration.

Assembly Envelopes

An *assembly envelope* is a special type of assembly component. An envelope is not a *functional* component of the assembly. It is a *reference* component, and it is ignored in global assembly operations (Bill of Materials, Mass Properties, etc.).

You can use an envelope by itself to quickly change the visibility of assembly components. You can also use an envelope (either alone or in combination with file properties), to select components for other editing operations, such as suppress, unsuppress, copy, or delete.

This functionality makes it easy to create and manage complex configurations. See **Simplifying Large Assemblies** on page 6-23 for more information about hidden and suppressed components in assembly configurations.

There are two ways to insert an envelope component:

- Insert an existing part file to use as the envelope.
- Create a new envelope component in the context of the assembly.

In either case, the envelope component must be a *solid* volume (not shelled). This is because selection with envelopes is based on *interference* between the volumes of the functional assembly components and the envelope component. It is *not* based on the *containment* of components within the bounding box of the envelope component.

Envelope components are displayed in a light blue transparent color when you use **Shaded** view mode. An **Envelope** feature is added to the Configuration Manager for each envelope component that you create. Also, envelope components are identified in the FeatureManager design tree.

To create an assembly envelope from a file:

1 Click **Insert, Envelope, From File**.

2 In the **Open Reference** dialog box, select the part file to use as an envelope.

3 Click in the assembly window where you want to place the envelope component. You can mate the envelope component to the other components in the assembly to position it precisely.

To create an assembly envelope in the assembly context:

1 Click **Insert, Envelope, New**.

2 In the **Save As** dialog box, enter a name for the new envelope component. A part document with this name is created.

3 Click a face or plane in the assembly window where you want to begin sketching the envelope component. **Plane1** of the new envelope component is mated to the selected face or plane with an **Inplace** mating relation.

4 Create the base feature and any additional features of the envelope component.

You can reference the geometry of the assembly components as you create the envelope component. If any of the referenced entities changes, the envelope updates accordingly.

5 When you are done defining the new envelope component, right-click anywhere in the graphics area, and select **Edit Assembly:*assemblyname***.

To remove an assembly envelope:

1 Select the envelope component in either the FeatureManager design tree or the graphics area. You can also select the **Envelope** feature in the Configuration Manager.

2 Press the **Delete** key. Both the envelope component and the **Envelope** feature are deleted.

To use the envelope for selection:

1 In the Configuration Manager, right-mouse click the **Envelope** feature, and choose **Select using envelope**.

2 Choose one or more criteria for selecting the components:

- **Inside envelope.** Components that are completely inside the envelope are selected.
- **Outside envelope.** Components that are completely outside the envelope are selected.
- **Crossing envelope.** Components that cross the boundary of the envelope are selected.

3 Click **OK**. Components that meet the criteria are selected, and you can perform the desired operation (**Suppress**, **Hide**, **Delete**, etc.).

To use the envelope for component visibility:

1 In the Configuration Manager, right-mouse click the **Envelope** feature, and choose **Show/Hide using envelope**.

2 Specify the envelope criteria for selecting the components (see above).

3 Choose whether to **Show** or **Hide** the components that meet the criteria.

4 Specify what to do with any component that does not meet the criteria:

- If you clicked **Show** in Step 3, you can either **Hide it** (regardless of current display state), or **Leave its show/hide state as is**.
- If you clicked **Hide** in Step 3, you can either **Show it** (regardless of current display state), or **Leave its show/hide state as is**.

5 Click **OK**. The display is updated to hide or show components according to your selections.

Component Selection by Properties

You can select components for editing operations based on properties. Properties include those specified under **File, Properties, Custom,** as well as some properties that are specific to SolidWorks documents, including **Part volume**, **Configuration name, Document name,** and **Part Mass.**

You can specify a set of properties and values to be used as selection criteria. You can combine properties using the logical operators **And** and **Or**.

For example, you assign the custom property *System* to each component in an assembly. You specify the name of the system the component belongs to as the value. Then you select all components that have the custom property *System* whose value is *Cooling* **Or** *Heating*. This selects all the components that belong to either the cooling system or the heating system of the assembly.

You can save any set of selection criteria as a *query*. Then you can reload it whenever you want to apply the same set of criteria again. The query (**.sqy**) is an ASCII text file that you can read and edit using a text editor such as Notepad.

Advanced Component Selection

You define the selection criteria by specifying the properties, conditions, and property values of components you want to select.

To define advanced selection criteria:

1 Click **Tools, Advanced Select**.

2 Under **Define additional criteria**, select a **Property** from the dropdown list. Properties include those listed under **File, Properties**, on the **Custom** tab, plus several special SolidWorks properties.

3 Select a **Condition**, and enter a **Value**.

- For most properties, conditions include standard mathematical operators.

- For some properties, conditions include **is (exactly)**, **is not,** and **contains**. You use **Contains** to specify a text string which is part of the text of the value; for example, the value "simplified" *contains* the string "sim".

- For the SolidWorks special property **Part volume**, conditions include **is inside**, **is crossing**, and **is outside**. The values that are available are the names of any envelopes that you have defined. See **Assembly Envelopes** on page 6-29.

4   Click the **Add** button to include the selection criteria in the **Criteria** list.

5   To add another criteria to the list, click **And** or **Or** as a logical operator for combining the criteria, then specify another **Property**, **Condition**, and **Value**. Click **Add** after specifying each new criteria.

6   To delete a criteria from the list, click in the *first column* of the **Criteria** list box, *beside* the criteria to delete. A blue box appears beside the selection. Click the **Delete** button.

7   When you have specified all the necessary criteria, click **Apply**. The selected components are highlighted. If needed, you can modify the selection, and click **Apply** again.

8   If you want, you can **Save Criteria** for future use, or **Load Criteria** that you saved earlier. See **Saving and Reloading Selection Criteria** on page 6-33.

9   Click the pushpin 🔲 if you want to keep the dialog box open for further selections and operations.

10 Perform the desired editing operation, such as **Suppress, Hide,** or **Copy**.

Advanced Show/Hide Components

For show and hide operations, you can also control the visibility of components that *do not meet* the specified criteria. You use the same methods to select the components as described in the previous section, **Component Selection by Properties** on page 6-31.

To define advanced visibility criteria:

1   In the Configuration Manager, right-mouse click a configuration, and select **Advanced Show/Hide**.

2   Define the selection criteria by specifying the properties, conditions, and values of components you want to select.

3 Under **Operations**, choose whether to **Show** or **Hide** the components that meet the criteria.

4 Specify what to do with any component that does not meet the criteria:

- If you clicked **Show** in Step 3, you can either **Hide it** (regardless of current display state), or **Leave its show/hide state as is**.

- If you clicked **Hide** in Step 3, you can either **Show it** (regardless of current display state), or **Leave its show/hide state as is**.

5 Click **Apply**. If needed, you can modify the selection, and click **Apply** again.

6 If you want, you can **Save Criteria** for future use, or **Load Criteria** that you saved earlier.

7 Click **Close** when you are satisfied with the selections.

Saving and Reloading Selection Criteria

When you use **Advanced Selection** or **Advanced Show/Hide**, you can save the selection criteria to a file. Then you can load the criteria later, to quickly apply the same set of selection criteria whenever you want.

To save selection criteria:

1 In the **Advanced Selection** or **Advanced Show/Hide** dialog box, specify a set of selection criteria.

2 Click **Save Criteria**.

3 In the **Save Criteria** dialog box, browse to the directory where you want the file to be, enter a file name, and click **Save**.

The query is saved with the extension **.sqy**

To reload selection criteria:

1 In the **Advanced Selection** or **Advanced Show/Hide** dialog box, click **Load Criteria**.

2 In the **Load Criteria** dialog box, browse to and select the query file (**.sqy**), and click **Open**.

The query is loaded, and the criteria are listed in the dialog box.

Opening Assemblies

When you open an assembly document, the application accesses the information in all the part files that the assembly references. It then builds the assembly using this information.

By default, when you open an assembly, it is opened in the configuration that was in use the last time it was saved.

You can open a different version of the assembly (a named configuration), that displays only the parts you want to see. This allows you to focus on specific parts or features and see how they fit in an assembly.

You can use simplified versions of parts, or *part configurations* (see **Part Configurations in an Assembly** on page 6-36) when opening an assembly. You can also specify sub-assembly configurations when you open an assembly (see **Sub-assembly Configurations** on page 6-37).

Note: Whenever you open an assembly, if a part or sub-assembly file cannot be found, you are asked to browse for it. You can also substitute a different part file, or the same part file with a new name.

Opening a Named Configuration

You can open a named assembly configuration, that is, an assembly showing only the specific components or sub-assemblies you want to work on. Refer to the section, **Using Assembly Configurations** on page 6-26 for information on configurations.

You can also open an assembly using a simplified version of each part, that is, a parts configuration in which you have suppressed certain features. See **Suppress/ Unsuppress** on page 5-10 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 the part configuration name.

Note: If the referenced configuration of any component specifies **Use Named Configuration**, this option does not override the named configuration.

7 Click **OK**. The selected configuration opens.

You can change to another configuration of the assembly while the assembly is open. See **Viewing an Assembly Configuration** on page 6-27 for information.

Suppressing/Unsuppressing All Components

You can quickly create new configurations, by globally suppressing or unsuppressing the components. The easiest way to do this is to open the assembly in a new configuration, with one of these options:

• structure only shown (all components suppressed)

• all referenced models shown (all components unsuppressed)

To open the structure of an assembly:

1 Click **File**, **Open**. In the **Open** dialog box, set **Files of Type** to **Assembly Files** or **All Files**.

2 Select an assembly, click **Configure**, and click **Open**.

3 In the **Configure Document** dialog box, select either:

• **New configuration showing assembly structure only.** This opens the assembly structure only. The components are all suppressed, regardless of the state they were in when last saved. No components appear in the assembly model until you unsuppress them.

• **New configuration showing all referenced models.** This opens the complete assembly. The components are all unsuppressed, regardless of the state they were in when last saved. All components appear in the assembly model.

4 Enter a name, and click **OK**.

Part Configurations in an Assembly

When you open an assembly, all the components are loaded into memory, even if they are not currently displayed in their own windows. If you right-click an assembly component, and select **Open *partname*.sldprt**, you display the previously loaded part in its own window. If you change the configuration in use in the part window, the newly displayed configuration is placed in memory for that part.

If a part occurs more than once in an assembly, you can use a different named configuration for each instance. The component *body* of each instance reflects the selected configuration. However, you can only see the *feature detail* for instances that use the part configuration *currently in memory*.

To change the configuration of a component instance:

1   Right-click the component name in the FeatureManager design tree, or right-click on a component face, and select **Component Properties**.

2   Under **Referenced Configuration**, select one of the following:

- **Use component's in-use or last saved configuration**. When you change the configuration in the part window, any instances that use this option will update when you return to the assembly.

- **Use named configuration**. To always use a specific configuration, regardless of the configuration currently in memory, select this option, and choose a configuration from the pull-down list.

   It is a good design practice to use named configurations in assemblies when parts have multiple configurations, to prevent the in-use configuration from affecting the assembly in unexpected ways.

3   If the selected component is part of a sub-assembly, choose which configuration properties to use:

- **Use properties specified in configuration *<configuration name>* of *<sub-assembly name>*.** The **Visibility** options (**Hidden, Suppressed, Color**) specified for this component in the selected configuration in the sub-assembly document are used.

- **Override properties specified in configuration *<configuration name>* of *<sub-assembly name>*.** The **Visibility** options specified for this component in *this* assembly are used.

4   Click **OK**.

Sub-assembly Configurations

When you open a top-level assembly, any sub-assemblies it contains are loaded into memory, even if they are not displayed in their own windows.

Sub-assemblies *always* use the configuration *currently in memory*. This means that, if a sub-assembly is used more than once in a top-level assembly, you can only use one configuration of that sub-assembly.

By default, the sub-assembly's last saved configuration is loaded when you open the top-level assembly. You can change the sub-assembly configuration, either immediately, or for future use.

To change the configuration of a sub-assembly (for immediate use):

1   In order to change the sub-assembly configuration used in the current top-level assembly, the sub-assembly document must be open. If the sub-assembly document is not already open, right-click the sub-assembly name in the FeatureManager design tree, and select **Open *assemblyname*.sldasm**.

2   In the sub-assembly document, select the desired configuration. When you return to the top-level assembly, all instances of the sub-assembly update.

To change the configuration of a sub-assembly (for future use):

1   Right-click the sub-assembly in the FeatureManager design tree, and select **Component Properties**.

2   Under **Referenced Configuration**, click **Open with named configuration**, and select the configuration name from the list.

3   Click **OK**.

   Important notes:

   • *This not an active switch.* Notice the use of the term **Open with** (next time), not **Use** (now). No change is made at this time. However, the *next* time you open the top-level assembly, the specified sub-assembly configuration will be used.

   • In order for the named configuration to be used the next time you open the top-level assembly, you must be sure that the sub-assembly is *not already open* (and therefore already loaded in memory).

   • Once the top-level assembly is loaded, if you open the sub-assembly, the specified configuration is displayed, because that is the configuration *currently in memory*.

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 **Component Properties.**

2   Click the **Color** button.

3   Specify the values for **Red**, **Green**, and **Blue**, and click **OK**,

   - or -

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

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

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

6   Click **OK**.

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

Sending Assembly Documents

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

This function uses the electronic mail application installed on your system.

To mail an assembly file:

1 With the assembly open, click **File, Send To**. The **Send Mail** dialog box appears.

2 Click either ***<assemblyname>*.sldasm only** or ***<assemblyname>*.sldasm and other documents it references**.

If you click ***<assemblyname>*.sldasm and other documents it references**, a list of the referenced documents, the number of documents, and the combined size of the documents is displayed.

Note: The size of the combined documents may be important to you because mailing large size documents may cause transmission problems.

You can selectively delete 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** to send your document(s).

4 Enter your mail password, the mailing address, and any other information requested by your mail application.

7

Working with Parts within an Assembly

In this chapter, you will learn about *top-down* assembly design, including several techniques for building and modifying components within the context of an assembly.

These techniques are based on *external references* and *time-dependent features.*

❑ You can use the geometry of one component to help define another. When you create references between components in an assembly, the models are fully associative. Changes you make to a *referenced component* cause the others to update accordingly.

❑ You can create cut or hole features that are added only after the components are assembled. These *assembly features* exist only in the context of the assembly.

❑ You can start with a layout sketch, define fixed component locations, planes, etc., then design the parts referencing these definitions.

❑ You can place component patterns, create molds, and join part volumes.

External References

An *external reference* is created when a 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 **Editing Parts in the Assembly Context** on page 7-4 for more information.

An example of an external reference is a mold. The cavity in the mold has an external reference to the design part you use to create the cavity. If you change the design part, the cavity changes to match it. See **Creating Molds** on page 7-15.

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

A component or feature with an external reference has the suffix **->** if the external reference is in-context. If the external reference is out-of-context, the suffix is **->?**.

To list the external references on a part or feature:

Right-mouse click the part or the feature with the **->**, and select **List External References.** The referenced components, features, and entities are listed, as well as the status of each reference.

Setting External Reference Options

To set external reference behavior:

1 Click **Tools, Options**, and select the **External References** tab.

2 Choose from these options:

- **Open referenced documents with read-only access.** Specifies that all referenced documents will be opened for read-only access by default.

- **Don't prompt to save read-only referenced documents (discard changes)**. Specifies that when this part is saved or closed, no attempt will be made to save its referenced documents that are opened for read-only access.

- **Allow multiple contexts for parts when editing in assembly.** Allows you to create external references for a single part in more than one assembly context. However, any individual *feature* or *sketch* may only have external references in *one context*.

3 Choose from these options for assemblies:

- **Automatically generate names for referenced geometry.** When this option is *off* (the default), you can mate to parts for which you have read/only access because you are using the internal face or edge IDs of the parts. Unless you plan to replace components, leave this option *off*, especially in a multi-user environment.

    When this option is *on*, you automatically create entity identifiers (for example: **Face1**, **Edge2**) when you mate the part, so you need write access to the part. Turn this option *on* if you intend to replace components using the same entity identifiers, remembering that you need write access to the parts.

    Note: You can also rename the corresponding edges and/or faces on replacement components to match the edge/face names on the original part. See **Replacing a Part in an Assembly** on page 6-9.

- **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.

4 Click **OK**.

Searching for Referenced Documents

When you open any parent document, all other documents that are referenced in the parent document are opened also. By default, the software searches for referenced documents in the following order:

1 The directory paths specified in the **Folders** list on the **External References** tab of the **Options** dialog box. This list is only used if the **Search document folder list for external references** option is checked.

2 The last path you specified to open a document.

3 The last path the system used to open a document (in the case of the system opening a referenced document last).

4 The path where the referenced document was located when the parent document was last saved.

5 If a referenced file still is not found, you are given the option to browse for it.

> **Note:** All updated reference paths in the parent document are saved when you save the parent document.

You can define the folder list, then enable or disable the search behavior as needed, without affecting the list.

To use a document folder list to search for external references:

1   Click **Tools, Options**, and select the **External References** tab.

2   Click the check box **Search document folder list for external references** to enable the search behavior.

3   In the **Folders** section, choose from these options:

- To *add* a directory path to search order, click the **Add** button. The **Choose Directory** dialog box appears. Browse to the desired directory, and click **OK** to add the path to the search list.

- To *delete* a directory path from the search list, select the path in the list box, and click the **Delete** button.

- To *change* the search order, select a directory path in the list box, and click the **Move Up** button or **Move Down** button to change the path's position in the search list.

4   Click **OK**.

Editing Parts in the Assembly Context

While editing an assembly, you can:

❑ Create a part in the assembly window

❑ Edit a part in the assembly window

❑ Open a part file in its own window

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 6-18.

Editing a Part in an Assembly

Editing a part while in an assembly allows you to modify a component without leaving the assembly. You can also reference surrounding geometry while creating new features. The sketches that you create while editing a part in the assembly may use any edge or can be dimensioned to any edge of any part. Features may use other end conditions, such as **Up to Surface**, to another component. See **Specifying End Conditions** on page 4-4.

To edit a part while in an assembly:

1 Right-click the component in the FeatureManager design tree or the graphics area and select **Edit Part**, or click **Edit Part** 🔧 on the Assembly toolbar.

The title bar shows the name of the part in the assembly that is open for editing for example, *partname* in *assembly*.**sldasm**. Note that the message in the status bar indicates that you are now editing the *part* document even though the entire

assembly is visible. 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. Under **Apply to**, select **System Defaults** or **All Possible**, and click **OK**.

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*, or click the **Edit Part** icon 🛠 again to return to editing the assembly.

Opening a Part in its Own Window

While in an assembly, you can open a part file in its own window and make the desired modifications with the part isolated. The changes automatically update the assembly.

To open a part within an assembly:

1 Right-click the component in the FeatureManager design tree or graphics 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.

Time-Dependent Features

SolidWorks gives you the ability to work with time-dependent features in an assembly. Time-dependent features are the basis of top-down design.

Time-dependent features include:

❑ Assembly features (assembly cuts, holes, and weld beads)

❑ Assembly planes, axes, and sketches.

❑ Assembly component patterns

When a time-dependent feature references components that have already been positioned by mates, the features become dependent on both the components and the mategroup(s) that position the components. You can view these dependencies by right-mouse clicking the feature and selecting **Parent/Child**.

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

In this example, coincident mates are used to align the faces of the components.
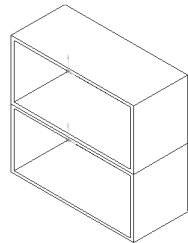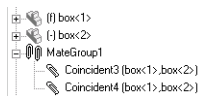
Then a cut is extruded through both components. This is an *assembly feature*, not a cut feature in the part document.
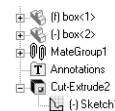
Now add a component (a pin, for example) to the assembly.
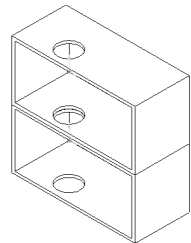
If you mate the pin to the assembly feature cut (which is time-dependent), an additional mate group is required. This is because the new component can only be positioned after the time-dependent feature has been updated.
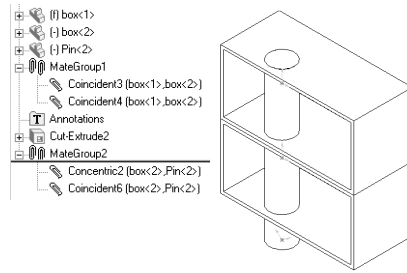
The position of a component in an assembly is fully defined in the first mate group which contains mates to it. If it is referenced in later mate groups, it is treated as a fixed component.

Additional mate groups are added automatically when required and removed automatically when the last mate they contain is deleted. When a new mate is added, it is put in the earliest mate group possible, based on parent-child relationships.

Note: It is good design practice to only reference time-dependent features for mating when that is the only way you can achieve the design intent of the assembly. You can edit the positions of the components with more flexibility when time-dependent features are not involved because the order of evaluation does not matter.

It is possible to have a situation where two components have been partially positioned in **MateGroup1,** and in order to complete the positioning, you need to add a mate which references a time-dependent feature which is dependent on **MateGroup1**.

You could do this by mating the bottom face of the pin coincident to the top face of the upper part first. Then add the assembly hole, and add the concentric mate between the pin and the assembly hole. The coincident mate is moved to the second mate group for a successful solution. This is because it must be solved simultaneously with the concentric mate to position the pin, and the concentric mate can only be solved after the assembly hole is updated.

In this example, the software detects that the coincident mate can move forward, and moves the mate when the concentric mate is performed. However, in more complicated cases you may need to determine which mates to move forward. To do this, delete the mates which position the component you wish to move, then add the mate to the time-dependent feature (to create the additional mate group), then add mates as needed to fully position the component.

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

Adding a Component Pattern

You can 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 the assembly a component that has a pattern feature, and a component that you can mate with the pattern.

2 Mate the component to be copied to the *original* feature used to create the pattern on the first component (not one of the copied instances).

3 Click **Insert, Component Pattern**.

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

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

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

7 Click **Finish**.
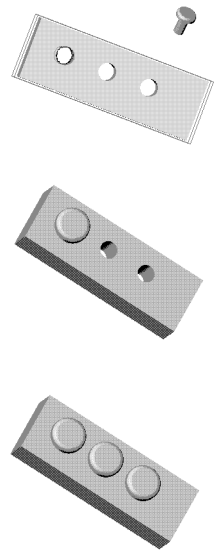
To define a new pattern for placing components in an assembly:

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** (the component to copy).

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**.

See **Linear Pattern** on page 4-33 and **Circular Pattern** on page 4-34 for more information about patterns.

Reorder and Rollback

Reorder

Reordering of assembly items in the FeatureManager design tree is dictated by parent-child relationships, just as it is with features in parts.

You can reorder mates within a mate group, assembly planes, axes, or sketches.

You cannot reorder the first mate group, assembly patterns, or in-context part features. You cannot reorder 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.

If you rollback to layout features before the components, all the components are rolled back as well.

Using an Assembly Layout Sketch

You can design an assembly from the top-down using layout sketches. You can construct a rough sketch showing where each assembly component belongs. Then, you can create and modify the design before you create any parts. In addition, you can use the layout sketch to make changes in the assembly at any time.

The major advantage of designing an assembly using a layout sketch is that if you change the layout sketch, the assembly and its parts are automatically updated.

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 you create each component. Use the layout sketch to define the component size, shape, and location within the assembly; make sure that each part references the layout sketch.

To design an assembly using the layout sketch technique:

The following example is typical of an automobile engine accessory drive belt system. The system has several pulleys and idlers, connected by a serpentine belt.

1 Create an assembly sketch. Use circles to represent the outer rims of the pulleys. Use tangent lines between the circles to represent the serpentine belt. Dimension the size and location of each pulley.

2 For *each* pulley, insert a new part, sketch a circle, add a **Coradial** relation with the corresponding circle in the layout sketch, then extrude the sketch. Add other features to the pulley parts as needed.

3 Using the layout sketch as a guide, create the belt around the pulleys as another new part.
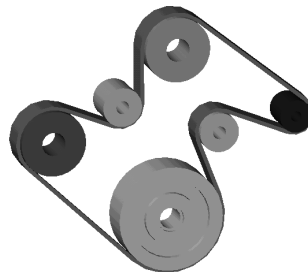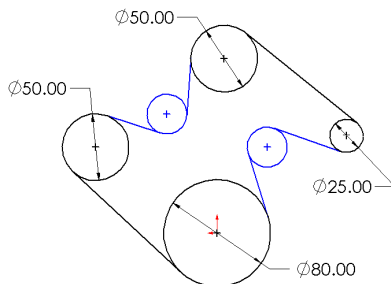
You can change the location of the pulleys in the assembly, or change the size of the pulleys either in the part documents or in the assembly.

To make changes to your design:

The assembly now consists of a belt and a set of pulleys whose locations and sizes are dictated by the layout sketch. The advantage of this type of design is clear when you have to make changes to the design.

For example, working *in the layout sketch,* you can:

- Rearrange the pulleys so that a slipping pulley has more of its surface covered by the belt

- Make a pulley larger so less torque is required to turn it

- Drag the pulleys and belt to dynamically experiment with different solutions to your problem

- Establish relationships within the sketch that would be difficult or impossible to create within the assembly

- Set an angle dimension to specify how much of each pulley must be covered by the belt

When you exit the layout sketch, the assembly and the parts are updated.
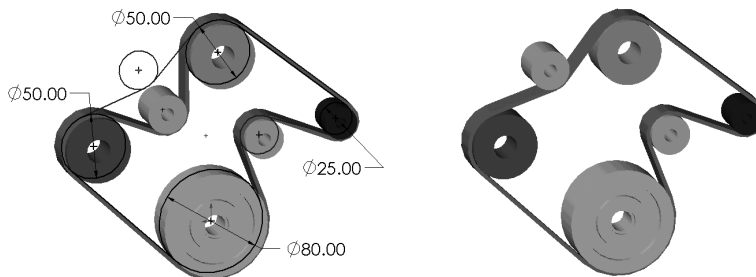
If you designed this assembly without a layout sketch, you would need to change many dimensions, or move components in the assembly, then rebuild each time.

The layout sketch does not have to be the master plan for a design. If you had a model of an engine with some fixed pulley locations, you could relate the layout sketch to the model. Then, to design the belt system, you could make the circles in the layout sketch coincident with the known locations of the pulleys in the model. You would still have the freedom to drag the other pulleys to different locations, and the assembly would update automatically when you made changes.

Working with Assembly Features

The **Assembly Feature** menu has three selections: **Cut**, **Hole,** and **Weld Bead**. Weld beads are described in Chapter 12, *Welding*.

While in an assembly, you can create cut or hole features that exist *in the assembly only.* You determine which parts you want the feature to affect by setting the scope.

This is useful for creating cuts or holes that are added *after* the parts are actually assembled, and that affect more than one component. When you want to add a cut or hole to a single component in an assembly, it is better to edit the part in context than to use an assembly feature.

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, click the component again in the graphics area or in the FeatureManager design tree.

Creating an Assembly Feature

Assembly feature cuts and holes affect the assembly only; the part files are not affected. You must set the scope before you can create an assembly feature.

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 extruded cut:

1   Open a sketch on a face or plane, and sketch a profile of the cut.

2   Click **Insert**, **Assembly Feature**, **Cut**, **Extrude**.

3   Set the **Depth**, **Type**, and other options as needed in the **Extrude Feature** dialog box.

4   Click **OK**.

To create an assembly feature revolved cut:

1   Open a sketch on a face or plane, and sketch a profile of the cut and a centerline.

2   Click **Insert**, **Assembly Feature**, **Cut**, **Revolve**.

3   Set the **Type** and **Angle** in the **Revolve Feature** dialog box.

4   Click **OK**.

To create an assembly feature hole:

1   Click the planar face where you want to create the hole.

2   Click **Insert**, **Assembly Feature**, **Hole**, **Simple** and set the **Diameter** and **Type**. If you choose the **Blind** type, specify the **Depth**.

3   Click **OK**.

To edit an assembly feature:

Right-click the assembly feature in the FeatureManager design tree, and select either **Edit Sketch**, **Edit Definition**, or **Feature Scope**.

Creating Molds

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

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

*cavity size = part size * (1 + 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(s). You may insert multiple design parts in a single mold base to create multiple cavities.

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

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

- or -

Click the mold base and click **Edit Part** .
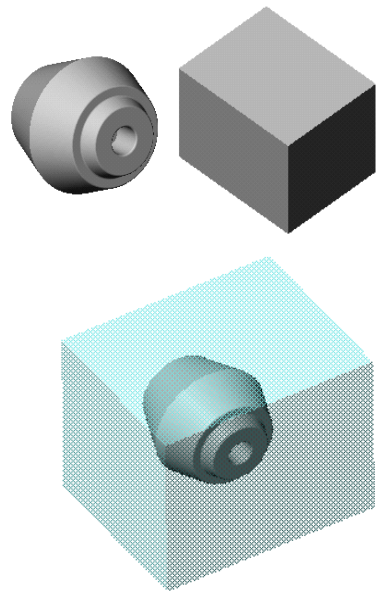
Note: It is important to be aware that you are editing the *part*, not the *assembly*, because the changes you are about to make will be reflected in the original part file of the mold base.

If you use standard mold base sizes, and do not want the originals to be affected by this operation, use the **Save As** command to save the mold base with a different name for use in each new mold assembly. Otherwise, the original mold base will include the cavity you are about to insert.

6 In the FeatureManager design tree, select the design part(s).

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

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

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

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

- **About Component Centroids.** Scales the cavity for each part about its own centroid.

- **About Component Origins.** Scales the cavity for each part about its own origin.

- **About Mold Base Origin.** Scales the cavity for each part about the origin of the mold base part.

10 Click **OK**.

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.
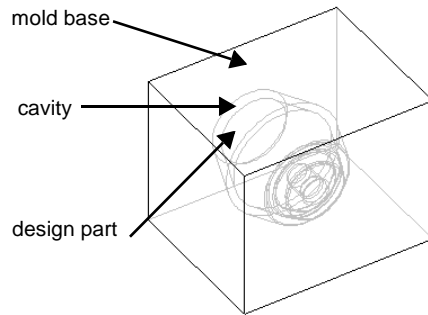
To cut the mold base to make two mold halves:

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, when projected, divides the mold base as desired (the parting plane). You can also select a model edge, and click **Convert Entities** ⬚ to project the edge as a parting line.

4 Click **Insert**, **Cut**, **Extrude**. Leave the **Flip Side to Cut** check box unchecked in the **Extrude Cut Feature** dialog box.
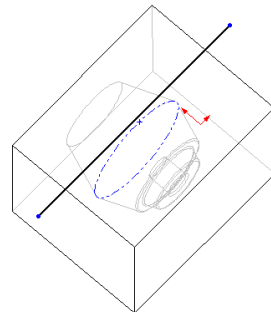
5 Click **OK**. The mold base is cut, exposing the cavity within the mold base. This is one half of the mold.

6 Save this part.

7 To create the other half of the mold, 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.
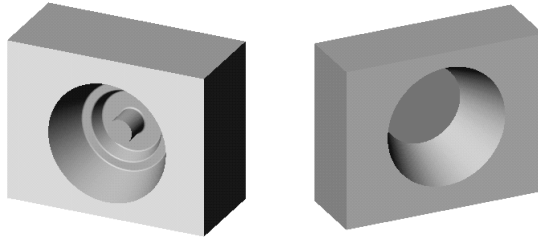
An additional mold example is found in Chapter 14, *Creating a Mold*, in the *Learning to Use SolidWorks 98 Tutorial*.

Creating Drafts for Molds

You may wish to add draft angles to molded design parts, to make it easier to remove the finished part from the mold base. Refer to **Split Line** on page 3-10 and **Draft** on page 4-26 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. Joining makes the model less confusing by removing overlapping solids.

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 or plane 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.
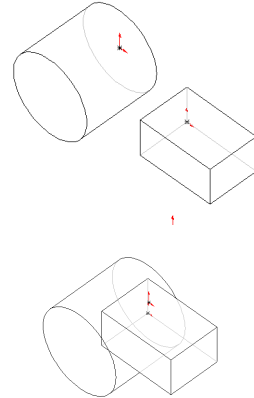
The system finds all abutting faces that touch but do not intrude into each other's space. It then finds the affected face with the smallest area, extends it into the other component, and fills in the resultant gap.

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**.
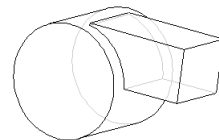
To identify which parts are used to create a joined part, either:

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

❑ Select the joined part, and click **File, Find References.**

❑ Expand the component containing the joined components, then expand the **Join** feature.

To edit the definition of the joined part to add or remove components:

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

The **Join** dialog box appears. Joined parts are listed in the **Design Component** box.

2 In the FeatureManager design tree, click the part you want to remove from the joined component, or select new components to be joined to the existing component.

3 Click **OK**. The joined component updates to reflect the changes.

Note: The joined part is fully associative to the original parts. Any changes made to the original parts are reflected in the joined part.

8

Drawings

You can create 2D drawings of the 3D solid parts and assemblies you design. Parts, assemblies, and drawings are linked documents; any changes that you make to the part or assembly change the drawing document.

Note: One-way associativity can be set between drawings and models during the software installation. This permits changes to the drawing that do not change the model. This option can only be reset with a new installation of the software.

Generally, a drawing consists of several views generated from the model. Views can also be created from existing views. For example, a section view is created from an existing drawing view.

This chapter describes:

❑ Setting default drawing and page set-up options

❑ Working with drawing templates and sheets

❑ Creating drawings with standard views

❑ Creating other views, such as detail, section, projected, broken, etc.

❑ Aligning and displaying views

❑ 2D sketching

Dimensions, notes, datums, geometric tolerances, and other notations complete the drawing. Dimensions and other annotations are discussed in the next chapter, **Detailing**.

Drawing and Page Setup Options

You have a variety of options to customize your drawings to your company's standards and to conform to the requirements of your printer or plotter.

To customize your SolidWorks 98 drawings, you can:

❑ Select default drawing options by clicking **Tools, Options** and then reviewing the options available on these tabs:

- **Drawing**. Specify the scale and display of views.

- **Crosshatch**. Specify the crosshatch patterns used for section views.

- **Line Font**. Specify the line weights and styles used for various edge types in the drawing.

  Detailing options are discussed in the next chapter, **Detailing**. Also see Appendix A for more information about the SolidWorks 98 options.

❑ Set the page properties to meet your printer or plotter requirements. Click **File, Page Setup** to specify:

- Paper margins and orientation

- Line weights

- Scale of printed drawing

- Custom headers and footers

  For more information, refer to **Page Setup** on page 1-33.

❑ Customize your drawing templates to conform to the standards of your workplace. You can:

- Customize information blocks and text (See **Customizing a Drawing Template** on page 8-8.)

- Change the sheet size and orientation (See **Modifying the Sheet Setup** on page 8-7.)

Drawings Options

You can set defaults for new drawings and views.

To specify the default drawing options:

1 Click **Tools, Options**, and select the **Drawings** tab.

2 Set any of the following as needed:

- **Sheet Scale.** Specify the scale to use for all views (except **Detail** views) on the sheet.

- **Type of Projection**. Select **1st Angle** or **3rd Angle.**

- **Default Display for New Drawing Views.**

  Select **Wireframe**, **HLG**, or **HLR**.

  For **Tangent Edge Display**, select **Visible**, **With Font** (as specified on the **Line Font** tab), or **Removed**.

- **Automatic placement of imported dimensions from model.** When checked, dimensions imported from the model are placed at a reasonable distance from the model in the drawing view.

- **Display drawing view borders.** When checked, the boundary of each view is displayed as a gray box.

- **Automatic scaling of 3 view drawings.** When checked, all the views of a **Standard 3 View** are scaled to fit on the drawing sheet, regardless of the paper size selected.

- **Automatic update of BOM.** When checked, the bill of materials in an assembly drawing is updated automatically to reflect component changes in the assembly. See **Bill of Materials** on page 9-28 for more information.

- **Automatic drawing view update when model changes.** When checked, all views in a new drawing are updated automatically when the model changes. For information about manual update of selected views, see **Updating Views** on page 8-29.

- **Show contents while dragging drawing view**. When checked, the model is displayed as you drag a view. When unchecked, only the view boundary is shown (as an empty box) while dragging. This is similar to the **Show window contents while dragging** behavior in Microsoft Windows.

- **Detail View Scaling.** Specify the scale (relative to sheet scale) for **Detail** views.

- **Detail Item Snapping.** When dragging detail items, you can infer to other detail items. Specify whether to infer to the centers, the corners, or both

(depending on how the item is dragged). For more information, see Chapter 9, *Detailing*.

You can modify these options for an individual sheet or view if necessary. For more information about these options, see Appendix A, *SolidWorks 98 Options*.

Crosshatch Options

You can specify the appearance of the crosshatch pattern used in section views.

Note: You need to specify the crosshatch pattern in the *part* document for which you will create a section view. By specifying the crosshatch pattern for *each part*, a section view of an assembly can include the appropriate crosshatch pattern for each of its parts.

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 is displayed in the **Type** box.

4   Specify the scope in the **Apply To:** box.

- **Active Document** – the document that you are currently working on

- **System Defaults** – all new documents

- **All Possible** – the current document as well as all new documents

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.

Line Font Options

You can specify the style and weight of lines for selected types of edges in drawings.

To specify the default line fonts:

1   Click **Tools, Options,** and click the **Line Font** tab.

2   Select an edge type from the **Type of Edge** list: **Visible Edges, Detail Circle,** etc.

3   Choose a line style to apply to the selected edge type from the **Line Style** list: **Solid, Dashed, Phantom**, etc.

4   Choose a line weight to apply from the **Line Weight** list: **Thin, Normal**, or one of six **Thick** weights.

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

   • **All Possible** – the current document as well as all new documents

   Note: If the active document is a *part* or *assembly*, the only choice is **System Defaults**; the changes become effective for *new* documents, but current documents are not affected.

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.

Note: To define the actual line width used for plotting and printing, click **File, Page Setup**. Enter the values for each **Line Weight** in the boxes on the **Printer** tab of the **Page Setup** dialog box.

Creating a Drawing

Drawings consist of one or more views generated from a part or assembly. The part or assembly associated with the drawing must be open and saved before you can create the drawing.

Drawing files have the **.slddrw** extension.

To create a new drawing:

1   Click **File**, **New**, or click ⬜ on the **Standard** toolbar. The **New** dialog box appears.

2   Select **Drawing** and click **OK**. The **Template To Use** dialog box appears.

3   Click one of the following to select a template:

- **Standard Template.** Select a standard sheet size template (for example, **C-Landscape**) from the list.

- **Custom Template.** Click **Browse,** navigate to a custom template on your system or the network, and click **Open**. See **Customizing a Drawing Template** on page 8-8 for information about creating a custom template.

- **No Template**. Select a blank, standard sheet from the **Paper Size** list, or choose the **User Defined** option. If you choose **User Defined**, you must specify the paper size in the **Width** and **Height** boxes.
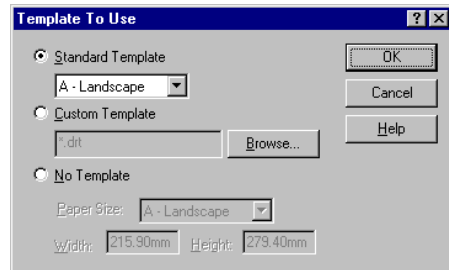
4   Click **OK**.

A new drawing document is opened, using the selected template, and the current sheet scale is displayed in the status bar at the bottom of the SolidWorks window.

Modifying the Sheet Setup

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

To specify sheet details:

1   Right-mouse click the sheet icon in the FeatureManager design tree, any blank area of the drawing sheet, or the sheet tab at the bottom of the drawing window, and select **Properties**. If the drawing is composed of several sheets, select the sheet icon or tab that corresponds to the sheet you want to modify.

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

   • **Name**. Enter a title in the box.

   • **Paper Size**. Select a standard size from the list, or select **User Defined** to specify a custom paper size. If you selected **User Defined**, specify a **Height** and **Width** for the paper.

   • **Scale**. Specify the default scale for all views on the sheet (default is 1:2).

   • **Template**. Select a standard template from the list, or select **Custom** or **None**. If you selected **Custom**, use the **Browse** button to select a custom template.

   • **Type of Projection**. Select **First Angle** or the default, **Third Angle**.

   • **Next View Label, Next Datum Label.** Specify the letter of the alphabet that will be used.

3   Click **OK**.
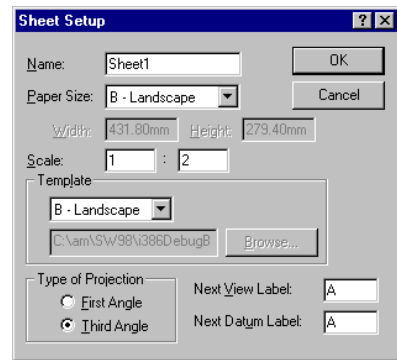
Multiple Sheets

You can add sheets to the drawing at any time.

To add a sheet:

1   Click **Insert**, **Sheet** or right-mouse click on the sheet tab and select **Add** from the menu. The **Sheet Setup** dialog box appears, with the default name of the next new sheet in sequence, for example, Sheet 2.

2   Set the parameters as described above, and click **OK**.
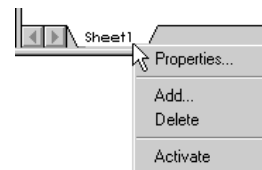
To delete a sheet:

1 Right-mouse click the sheet tab, and select **Delete**.

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

You can also click on the sheet in the FeatureManager design tree or the graphics window, and press the **Delete** key. Click **Yes** in the **Confirm Delete** dialog box.

To view a different sheet:

Click the sheet tab, or double-click the sheet icon in the FeatureManager design tree.

Customizing a Drawing Template

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

To edit the drawing template:

1 Right-mouse click the sheet or template icon in the FeatureManager design tree, or any blank area on the drawing sheet and select **Edit Template** from the menu.

2 To edit existing text in the title blocks, double-click on the text.
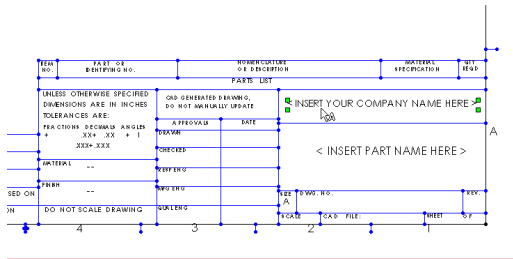
The **Properties** sheet for **Note** appears.

- Enter new text in the **Note Text** box.

- If you want to use a different font, deselect **Use Document's Font**, then click **Font** to select a new font style or size.

- Use **Insert Hyperlink** or **Link to Property** if desired. See **Hyperlinks** on page 9-19, and **Links to Custom Properties** on page 9-19 for more information.
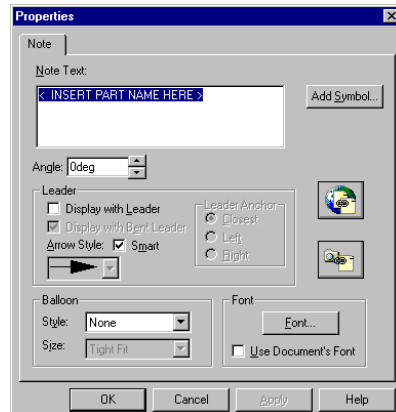
3 Click **OK** to accept the new font and/or text.

4   To add text, click on the sheet, then click **Insert**, **Annotations**, **Note** or A .
    Specify the text properties, then drag the text to the desired location.

5   To edit the template format by moving, deleting, and adding lines or text in the
    text blocks:

    • To delete, click the line or text and press the **Delete** key.

    • To move, click the line or text and drag to a new location.

    • To add lines, click the **Tools**, **Sketch Entity**, **Line** or ◺ .

6   To add objects from other applications, such as bitmaps of your company logo,
    standard note text from a file, etc., click **Insert, Object**.

    You may create a new object or insert one from a file. Objects may be *linked* or
    embedded in template files. See Chapter 14, *Object Linking and Embedding,*
    for more information.

To save the template:

Click **File**, **Save Template**.

• To overwrite a **Standard Template**, select the template from the list, and click
  OK.

• To save the template with a new name, click **Custom Template**. Then **Browse**
  to the desired directory, enter a new template name, and click **Save**.
  (Template files have the **.slddrt** extension.)

If you have added any custom file properties to the drawing, when you save the
template, the properties are saved with the template. Then, when you use the
template in another drawing or sheet, the custom properties saved with the
template are added to the new drawing.

See **Custom File Properties** on page 5-19 for more information.

To end editing the template and return to working with an individual sheet:

Click **Edit, Sheet**, or right-mouse click the sheet or template icon in the
FeatureManager design tree, or any blank area of the sheet, and select **Edit Sheet**.

The Drawing Window

The drawing window includes a FeatureManager design tree that is similar to the design tree in the part and assembly windows. The FeatureManager design tree for drawings consists of a hierarchical list of items pertaining to the drawing. There is an icon for each sheet; under each sheet, there are icons for the template and each view.

A ⊞ symbol beside an item's icon indicates that it contains associated items. Click the ⊞ to expand the item and display its contents.

Standard views contain the feature list for the part or assembly shown in the view. Generated views, such as detail or section views, contain different, view-specific items (detail circles, section lines, etc.).
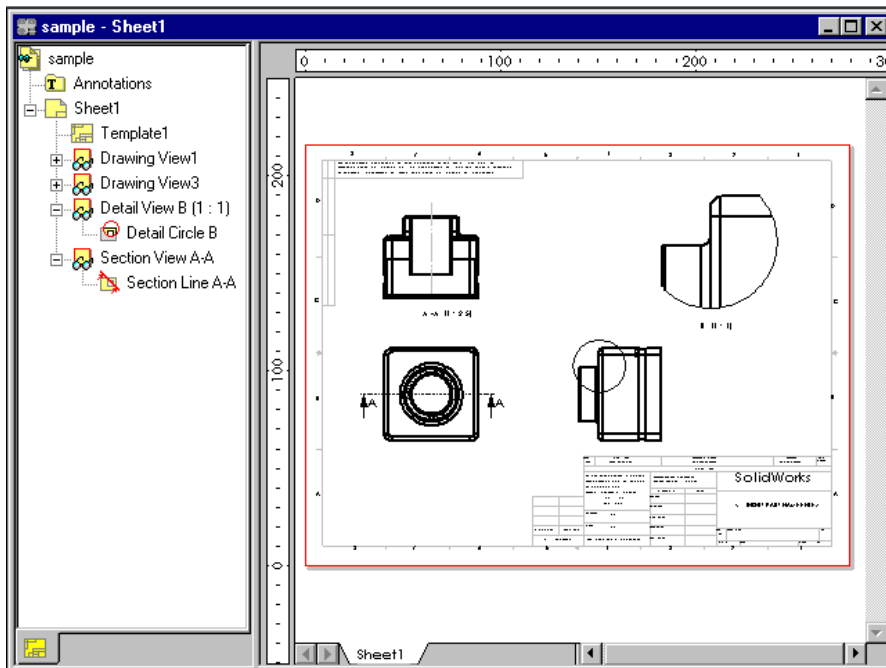
Views of Parts and Assemblies

You can generate drawings of a part or assembly in a variety of view types: **Standard 3 View**, **Projection**, **Named**, and **Relative to Model**. You can also create **Auxiliary**, **Detail**, **Section**, **Aligned Section**, or **Broken** views from existing views. The part or assembly document must be open and saved before you create a drawing of it.

You can also create an **Empty** view for sketching. See **2D Sketching** on page 8-36 for more information.

For some operations, you need to *activate* a view; for others, you *select* a view.

❑ *Activating* a view allows you to sketch entities on that view. This is necessary for sketching section lines, defining areas for detail views, etc. The border of an active view is a red shadowed box. To make a view active, double-click anywhere within the boundary, or right-click within the boundary and select **Activate View**.

To *de-activate* the view, either double-click again, or right-click and select **Activate Sheet**.

❑ *Selecting* a view allows you to create a projected view, to insert break lines for a broken view, to move the view on the sheet, to resize the view boundary, etc. A selected view has a green border with drag handles. To select a view, click anywhere within the boundary.

View Boundaries

As the cursor passes over the edge of a drawing view, the view boundary is highlighted. The size of the view boundary is calculated automatically, based on the size, shape, and orientation of the model shown in the view. You can adjust the size of the view boundary (enlarging the view boundary may make it easier to select or activate a view). You cannot make the view boundary smaller than the model displayed in the view. View boundaries, and the views they contain, may overlap.

To change the size of the view boundary:

1 Select the view.

2 Move the cursor over a green drag handle at the corner or side of the boundary until it changes to a resize cursor shape.

3 Drag the handle as needed to resize the boundary.
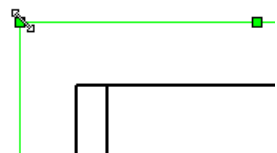
To turn the boundary display on/off:

On the **Drawings** tab of the **Options** dialog box, click **Display drawing view borders.** When checked, the boundary of each view is displayed as a gray box. When unchecked, boundaries are hidden. This option applies to *all* the views in the drawing.

Even if the boundaries are hidden, they still become highlighted when the cursor passes over them.

Standard 3 View

The **Standard 3 View** option under **Insert**, **Drawing View** creates three default orthographic views of a part or assembly displayed at the same time.

To create the Standard 3 View:

1 Open the saved part or assembly, and open a new drawing.

2 Click **Insert**, **Drawing View**, **Standard 3 View,** or on the Drawing toolbar. The cursor changes to include a small part symbol .

3 Switch to the part or assembly window (select it by name from the **Window** menu, or tile the windows, then select it), and click anywhere to select.

The drawing window returns to the front (if necessary) and the three views are placed in the drawing.

To create the Standard 3 View (another method):

The default view created when you drag and drop a part or assembly into a drawing is the **Standard 3 View**.

1 Open the part or assembly, and open a drawing window.

2 Drag the name from the top of the FeatureManager tree of a part or assembly document, and drop it into an open drawing window, *or*

Drag and drop a part or assembly document from the Windows Explorer window into an open drawing window.

The lower left view in a 3rd angle projection is the default **Front** view from the part or assembly. (For 1st angle, the front view is at the upper left.)
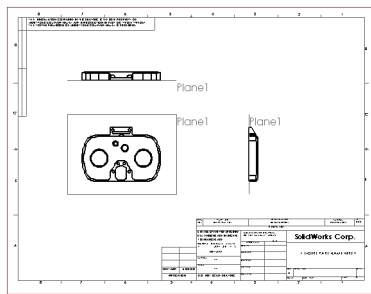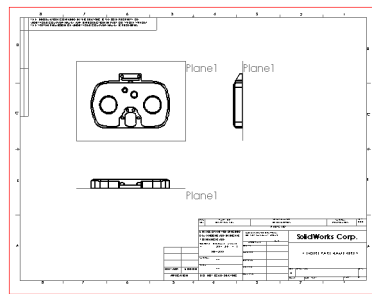
3rd Angle Projection   1st Angle Projection

The alignment of the top and side views is fixed in relation to the front view. The top view can be moved vertically, the side view can be moved horizontally.

For more information about arranging views on a sheet, see **Moving Views** on page 8-30 and **Aligning Views** on page 8-31.

Projection View   🖼

A **Projection** view is created by projecting an orthogonal view, using **First Angle** or **Third Angle** projection, as specified in the **Sheet Setup** dialog.

To create a projection view:

1   Select an existing view by clicking inside its boundary.

2   Click **Insert**, **Drawing View**, **Projection** or 🖼 on the Drawing toolbar. The cursor shape changes to ✛.

   As you move the cursor, a preview of the view boundary is displayed, snapped to the nearest projection. To override the snapping behavior as you place the projection view, press the **Ctrl** key as you move the preview around. To resume the snapping behavior while dragging, release the **Ctrl** key.

3   Move the cursor until the view is where you want it to be, then click to place the view.

4   Select a projection direction by clicking beside the selected view in the desired direction. You can project to the left, right, above, or below (in this example, the user clicked to the right of the selected view).

Selected view   Projected view
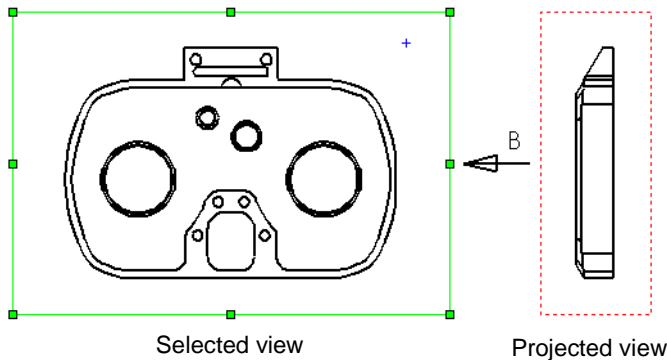
A projection view is placed on the sheet, aligned to the view from which it was created. By default, you can move a projection view only in the direction of the projection. See **View Alignment and Display** on page 8-30 for information about changing the alignment of views.

5   To display an arrow indicating the direction of projection, right-click the projection view on the sheet or in the FeatureManager design tree, and select **Properties**. Click **Display view arrow** and enter a label if desired (maximum of two characters).

Auxiliary View

An **Auxiliary** view is similar to a projection view, but unfolded normal to a reference edge in an existing view.

To create an auxiliary view:

1   Select the reference edge (*not* a horizontal or vertical edge, which would create a standard projection view).

2   Click **Insert**, **Drawing View**, **Auxiliary** or on the Drawing toolbar.

    As you move the cursor, a preview of the view boundary is displayed. To override the default alignment as you place the auxiliary view, press the **Ctrl** key as you move the preview around. To resume the snapping behavior while dragging, release the **Ctrl** key.

3   Move the cursor until the view is where you want it to be, then click to place the view.

    The auxiliary view is placed on the sheet, aligned to the edge from which it was created. By default, you can move an auxiliary view only in the direction of the projection. See **View Alignment and Display** on page 8-30 for information about changing the alignment of views

    If you use ISO/DIN/JIS/BSI dimensioning standards, a view arrow is added automatically.
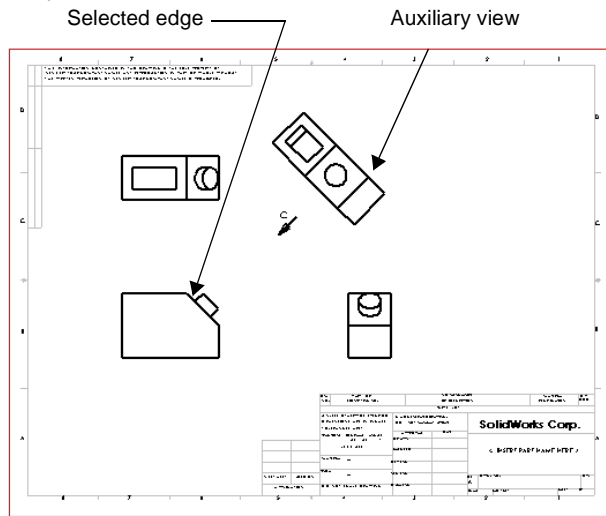
4   If you use ANSI dimensioning standards, you can display an arrow indicating the direction of projection. Right-click the auxiliary view on the sheet or in the FeatureManager design tree, and select **Properties**. Click **Display view arrow** and enter a label if desired (maximum of two characters).

Named View

You create a **Named** view by selecting from the view names in the model document. Named views include:

❑ Standard orthographic views (**Front**, **Top**, **Isometric**, and so on)

❑ **Current** model view

❑ Custom views created in the model by rotating the model, then saving the view by name. The entire part or assembly is displayed at the scale of the selected view, even if the selected view orientation in the part or assembly displays a zoomed-in view. For more information about creating a named view in the **View Orientation** dialog box, refer to **Using Named Views** on page 1-17.

To insert a named view:

1   In the drawing, click **Insert**, **Drawing View**, **Named View** or on the Drawing toolbar. The cursor changes to ⬡ .

2   Switch to the part or assembly window (select it by name from the **Window** menu), and click anywhere.

The **Drawing View - Named View** dialog box appears. The list contains the names of all the standard views, the **Current** view, and any custom views you created in the model.
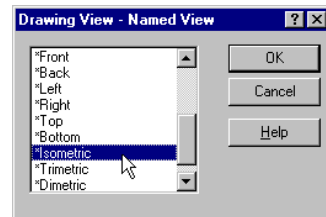
3   Double-click a named view from the list, or select a named view and click **OK**. The cursor changes to ✛ .

4   Switch to the drawing window, and click where you want to place the view. Named views are not aligned to other views by default, and may be moved freely to any location on the drawing sheet.

Default Front View

Isometric View

Saved Named View or Current View
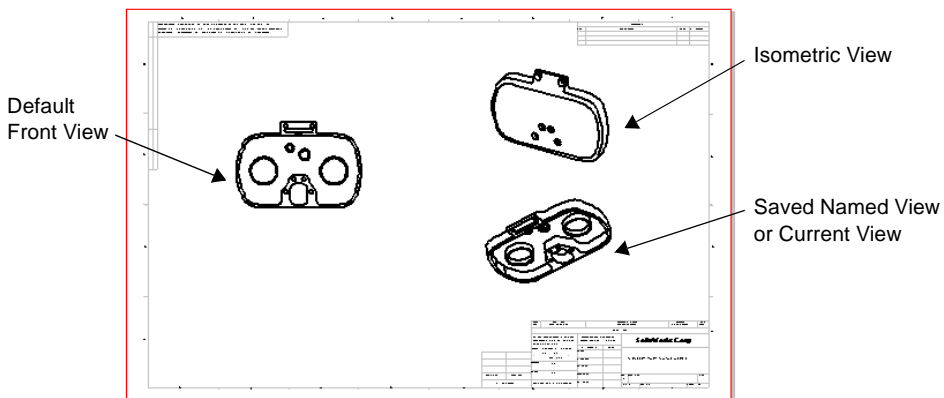
Relative to Model View $\boxed{}^{\mathbf{R}}$

A **Relative to Model** view is an orthographic view defined by two orthogonal faces or planes in the model and the specification of their respective orientations. This view type is also used to set the first orthographic view in a drawing if it is to be something other than a default view.

To insert a relative to model view:

1 Click **Insert**, **Drawing View**, **Relative To Model** or $\boxed{}^{\mathbf{R}}$ on the Drawing toolbar. The cursor changes to the model face selection shape $\mathbb{\&}\diamondsuit$.

2 Click a face or plane in the model window that you want to have a specific orientation.

The **Drawing View Orientation** dialog box appears.
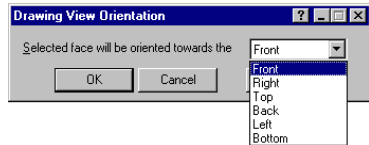
3 Select an orientation (**Front**, **Top**, **Left**, etc.), and click **OK**.

4 Select another face or plane, *perpendicular* to the first.

5 Select the orientation for this face from the **Drawing View Orientation** list.

6 Click with the $\Leftrightarrow$ cursor on the drawing sheet where you want to place the view. Relative to model views are not aligned to other views by default, and may be moved freely to any location on the drawing sheet.

Note: *Tiling* the drawing and model windows may be easier than switching windows when you need to select the model faces and view location.

Example:

A true view of an angled face perpendicular to the front face of a model is created in this manner. The angled face is specified as **Front** and the front face as **Left**. If the angle of the face in the model changes, the views update to maintain the orientation as originally specified.
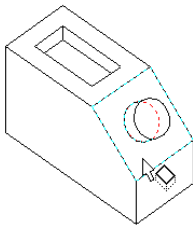
Click the angled face, and select **Front**
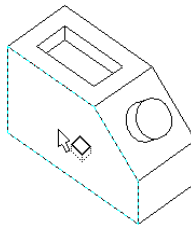
Click a perpendicular face, and select **Left**
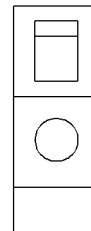
Resulting view, **Relative to Model**

Detail View

You create a **Detail** view to show a portion of a view, usually at an enlarged scale. This detail may be of an orthographic or 3D view or a section view.

To create a detail view:

1   Activate an existing view (double-click the view, or right-click and select **Activate View**).

2   Using a sketch entity tool, create a closed profile around the area to be detailed. (The profile is usually, but not necessarily a circle.)

3   Select one entity of the profile.

4   Click **Insert**, **Drawing View**, **Detail** or on the Drawing toolbar. As you move the mouse, a preview of the view boundary is displayed.

5   Click when the view is where you want to place the labelled detail view.

The note on the detail view includes the letter label and scale.

Sketched detail circle   Detail circle label   Detail view note

Active view   Detail view
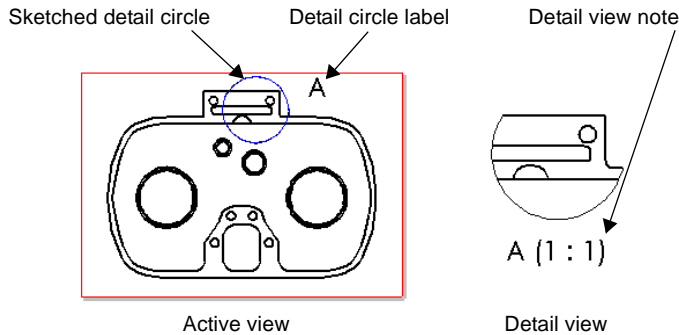
Detail views are not aligned to other views by default, and may be moved freely to any location on the drawing sheet.

To modify a detail view:

❑ To change the scale of the detail view, right-click the view on the sheet or in the FeatureManager design tree, and select **Properties**. Under **Scale**, *deselect* **Use sheet's scale**, enter a new scale ratio, and click **OK**.

❑ To change the detail view note text, double-click the text (or right-click the text and select **Properties**), and modify it as needed in the **Note** dialog box.

   • If you change the label in the note, the label on the detail circle updates.

   • If you change the scale in the note, the detail view scale updates.

❑ To change only the detail circle label, double-click it, enter the new label, and press **Enter**. The detail view note updates.
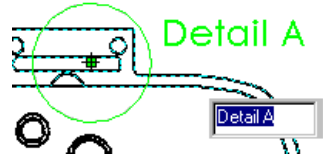
❑ To change the detail circle label font and letter, right-click the letter and select **Properties**. Enter a new letter if needed. Deselect **Use Document's Font**, click **Font**, then choose a new size or style.

❑ To move the note or the detail circle label, drag the item with the mouse.

To redefine the cutting edge of the detail circle:

1 Right-mouse click on the sketched detail circle and select **Edit Sketch**.

2 Click and drag the profile to change the shape or size. Note the cursor shape for the detail circle .

3 Click **Edit, Rebuild** or [8] on the Standard toolbar to update the detail view.

You can also reshape the detail circle by dragging it. The view automatically updates.

To specify the line font for the detail view border:

1 Click **Tools, Options**.

2 On the **Line Font** tab, under **Type of Edge**, click **Detail Border**, then choose a **Line Style** and **Line Weight**.

3 Click **OK**.

Broken View

You can use a broken view (or interrupted view) on the drawing of a long part that has a uniform cross-section. This makes it possible to display the part in a larger scale on a smaller size drawing sheet.

Reference dimensions and model dimensions associated with the broken area reflect the actual model values. See Chapter 9, *Detailing*, for information about dimensioning a drawing view.

To create a broken view:

1   Select the drawing view and click **Insert, Vertical Break** (or **Horizontal Break**). Two break lines appear in the view.

   You can add more than one set of break lines to a view; however, all breaks must be in the same direction. You can only add breaks to an *unbroken* view.

2   Drag the break lines to the places on the part where you want the breaks to occur.

3   Right-mouse click inside the highlighted boundary and select **Break View** from the menu.

   The part is displayed with a gap in the part geometry.
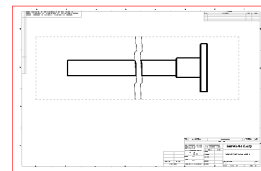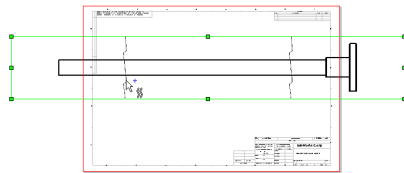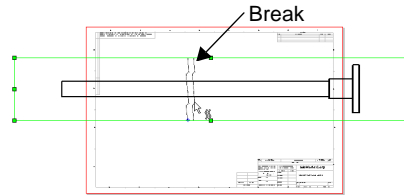
Break

To restore a broken view to its unbroken state:

Right-mouse click the broken view, and select **Un-Break View**.

To modify a broken view:

- To change the position of the break, drag the break lines.

- To change the width of the break gap, click **Tools**, **Options,** and select the **Detailing** tab. Enter a new value in the **Break Gap** box under **Break Lines**.

- To change the shape of the break lines, right-mouse click a break line and select a line style from the menu: **Straight Cut**, **Curve Cut, Zig Zag Cut**, or **Small Zig Zag Cut.**
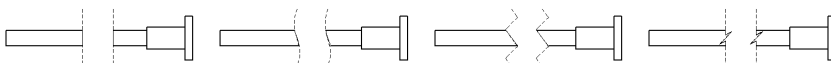
Section View

You create a **Section** view by partitioning the view with a section line. The view may be a straight cut section or an offset section defined by a stepped section line. The section line may also include arcs.

To create a section view:

1 Activate an existing drawing view (double-click the view, or right-click and select **Activate View**).

2 Click **Tools**, **Sketch Entity**, **Centerline** or **Line**, or click ◻ or ◻ on the Sketch Tools toolbar.

3 Sketch a single or stepped centerline or line through the model in the active view.

4 Select the sketched centerline or line (if it is not already selected). For a stepped line, you only need to select one segment.

5 Click **Insert**, **Make Section Line**.

Notice the arrows, indicating the direction of the cut. If necessary, double-click on the section line to reverse the direction of the arrows.

Now you can edit other section line properties that control the resulting view, if you wish. See **Modifying Section Views** on page 8-23 for more information.

6 With the section line still selected, click **Insert**, **Drawing View**, **Section** or ◻ on the Drawing toolbar.
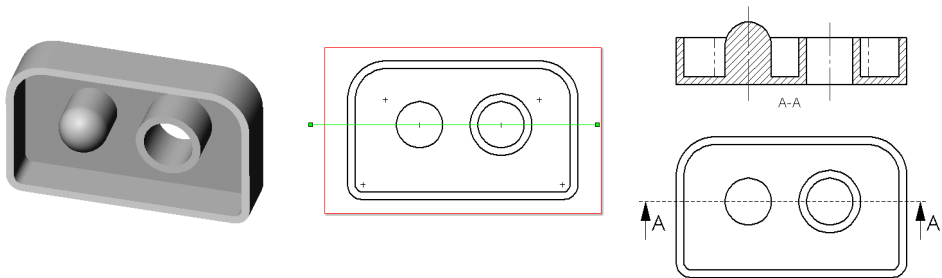
Note: If you prefer to create the section line and the default section view in a single step, skip Steps 5 and 6. Instead, after you select the sketched segment, click ◻ or **Insert**, **Drawing View**, **Section**.

As you move the cursor, a preview of the view boundary is displayed. By default, the view is aligned in the direction of the arrows on the section line. To override the default alignment as you place the view, press the **Ctrl** key. To return to the default alignment while dragging, release the **Ctrl** key.

In its default alignment, a section view can only move in the direction of arrows. See **View Alignment and Display** on page 8-30 for information about changing the alignment of views.

Aligned Section View

You can create an aligned section view through a model, or portion of a model, that is aligned with a selected section line segment.

To create an aligned section view:

1 Activate an existing drawing view (double-click the view, or right-click and select **Activate View**).

2 Click **Tools**, **Sketch Entity**, **Centerline** or **Line**, or click ◣ or ▮ from the **Sketch Tools** toolbar.

3 Sketch the section line. The section line should consist of two connected lines at an angle to each other.

4 Select the segment of the sketched section line to which you want to align the view.

5 Click **Insert**, **Make Section Line**.

   Notice the arrows, indicating the direction of the cut. If necessary, double-click on the section line to reverse the direction of the arrows.

   Now you can edit other section line properties that control the resulting view, if you wish. See **Modifying Section Views** on page 8-23 for more information.

6 With the section line still selected, click **Insert, Drawing View, Aligned Section** or on the Drawing toolbar.

Note: If you prefer to create the section line and the default aligned section view in a single step, skip Steps 5 and 6. Instead, after you select the sketched section line segment, click or **Insert**, **Drawing View**, **Aligned Section**.

You cannot change the angle of the line that you selected to align the view; you can change its position, however.
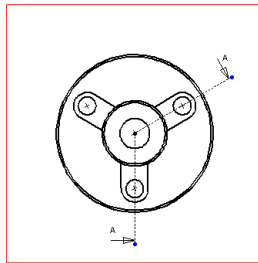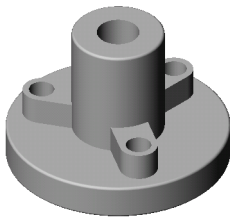
Active view with section line   Aligned section
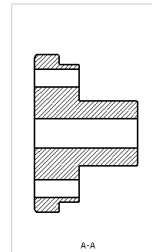
As you move the cursor, a preview of the view boundary is displayed. By default, the view is aligned in the direction of the arrows on the selected segment of the section line. To override the default alignment as you place the view, press the **Ctrl** key. To return to the default alignment while dragging, release the **Ctrl** key.

In its default alignment, an aligned section view can only move in the direction of arrows on the selected segment. See **View Alignment and Display** on page 8-30 for information about changing the alignment of views.

Modifying Section Views

❑ To edit the letter label on the section line, double-click the letter label, enter a new letter, and press **Enter**.

❑ To edit the note on the section view, double-click the text, make any necessary changes (edit the text, add symbols, change the font, etc.) in the **Note** dialog box, and click **OK**.

❑ To edit the location of section lines, drag them as needed. You can also right-mouse click the section line and select **Edit Sketch**, then make changes.

❑ To change the cut direction as indicated by the arrows, double-click anywhere along the section line.

❑ After changing the cut direction or editing the section lines, update the view using one of these methods:

  • Click 🔳 or **Edit**, **Rebuild**.

  • Right-mouse click the section view and select **Update View**.

  • Select the view and click 🔳 .

Modifying the crosshatch

Section views use the default crosshatch pattern, as defined in **Tools**, **Options, Crosshatch**. Changes to the default crosshatch pattern selection take effect in the *next* SolidWorks session. See **Crosshatch Options** on page 8-4 for information.

To modify the crosshatch pattern for a section view:

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

2 Specify the **Pattern**, **Scale** and **Angle**, and observe the **Preview**.

3   Under **Apply To**, choose whether to change only the selected **Region**, the section faces for the entire **Component** (in a section view of an assembly), or all the section faces in the **View**.

4   To remove the crosshatch for a layer (**Region**, **Component**, or **View**), select the layer in the **Apply To** list, and click **Remove Crosshatch**. (This option is only available for layers that can logically be removed.)

The crosshatch is removed a layer at a time. For example, if you remove the crosshatch from a region, any crosshatch applied earlier to the whole component is revealed. If you remove crosshatch from a component, the default crosshatch for the view is revealed.

5   Click **Apply** or **OK** to accept the changes, or click **Cancel**.

Modifying the centerline display

Centerlines are automatically added where needed (holes, circular extruded bosses or cuts, and revolved features) in a section view. To specify the distance to extend the centerline beyond the model edge, click **Tools, Options, Detailing,** and under **Dimensioning Standard**, enter a value for **Centerline extension**.

You can modify the length of the centerlines by dragging the endpoints. If you do not want the centerlines, you can either delete them from the section view, or you can turn off the display by clicking **View**, and *deselecting* **Axes**.

Modifying the section line properties

You can also make changes to a **Section View** or **Aligned Section View** by editing the section line properties. You can set these properties either before or after creating the section view.

If you want to set these properties *before* creating the view, sketch the section line, select it, then click **Insert, Make Section Line**. This makes the sketched line into a section line without automatically creating the associated view. Then right-mouse click the section line and select **Properties**.

If you want to modify the section view *after* it is created, right-mouse click the section line (not the view) and select **Properties**. You need to click **[8]** or **Edit**, **Rebuild**, or right-click the section view and select **Update view**, after modifying section line properties in order to update the view.

These two options affect only the labels:

- **Label.** Change the letter that is associated with this section line by selecting the letter and entering a new label.

- **Use Document's Font.** When checked, the section line text uses the same font as the rest of the document. When unchecked, you can click the **Font** button and specify the font style and size to use.
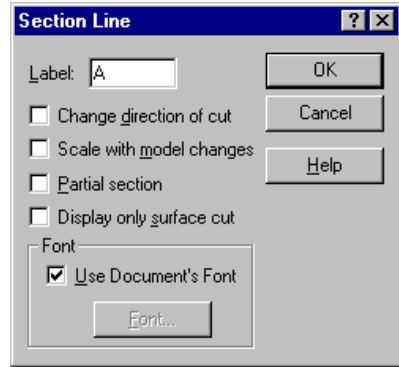
The remaining options in the **Section Line** dialog box control the representation of the model in the resulting view. They may be used individually or in combination, for either section views or aligned section views. Experiment until you get the results that you want. Remember to update the view after you make changes to the **Section Line Properties**.

The examples that follow are based on this model and the view containing the section line shown here.
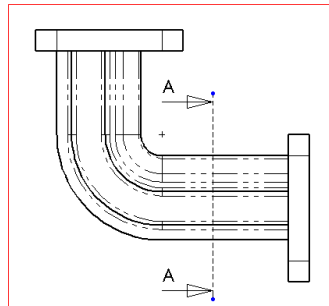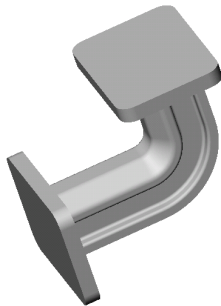
- **Change direction of cut.** Reverse the direction of the section line cut. This has the same effect as double-clicking the section line.

- **Scale with model changes.** The section line changes size parametrically with the model. This could result in a section cut that is not what you intended, however. The default for this option is off.

- **Partial section.** If the section line does not go all the way across the view, selecting this option causes the section view to include only the portion of the model crossed.

Change direction of cut checked for both views

Partial section
unchecked

Partial section
checked
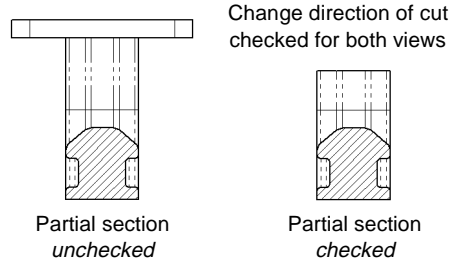
- **Display only surface cut.** A simplified view, with surfaces behind the section plane removed. Only the surface(s) actually cut by the section line are displayed in the section view.

Display only surface cut
unchecked

Display only surface cut
checked
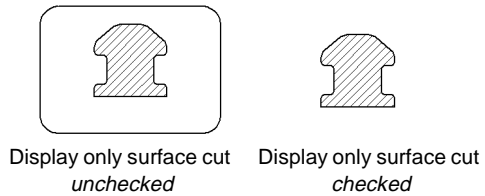
Section Views of Assemblies

When you create a section (or aligned section) view of an assembly, you can exclude (leave uncut) selected components by specifying the **Section Scope**.

To exclude components from a section view of an assembly:

1 Activate a view, and create a section line as described in **Section View** on page 8-21 or **Aligned Section View** on page 8-22.

2 Click **Insert, Drawing View, Section** [⬚], or **Aligned Section** [⬚]. The **Section Scope** dialog box appears.

3 On the active drawing view, or in the FeatureManager design tree, click the components that you want to exclude from the section view. The selected components are listed in the **Excluded Components** box.

   Note: When selecting components in the FeatureManager design tree, be sure to select them from the *active view*.

   To remove a component from the list, select the component again.

Both components included

Shaft excluded
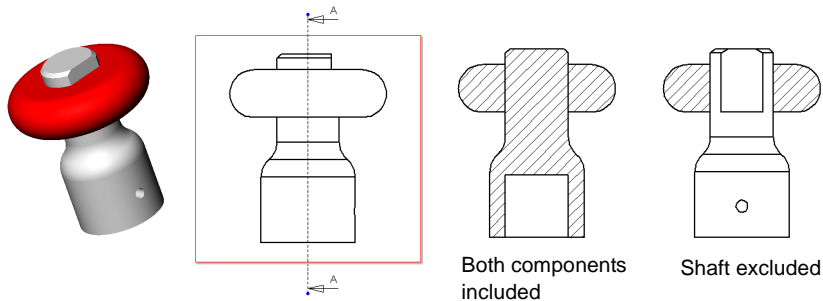
4 Set the **Exclusion Status** if the selected component is used more than once in the assembly (for example, if it is a member of a pattern, or if it is used as a component of more than one subassembly):

- If you want to exclude *all the instances* of the selected component in the assembly, click **Don't Cut All Instances**. In the resulting view, all instances of the selected component will be left uncut.

- If you want to exclude *only the selected instance*, leave this unchecked. In the resulting view, only the selected instance will be uncut; all others will be cut.

5 Click **OK**.

In the resulting section view, excluded components are not cut. They may be shown in the view, or you may remove them from the view by setting the section view property **Display only surface cut**. See **Modifying Section Views** on page 8-23 for more information.

To edit the section scope for the view:

Right-mouse click the *section* view, either on the sheet or in the FeatureManager design tree, select **Properties**, and click the **Section Scope** tab of the **Drawing View Properties** dialog box.

- To add components to the **Excluded Components** list, click the components in the section view (*not* the parent view).

- To remove a component from the list, select the component again.

- To change the **Exclusion Status** for components that are used more than once in the assembly, select the component in the **Excluded Components** list, then click **Don't Cut All Instances**.

View Properties

The **Drawing View Properties** sheet provides information about the drawing view and the model or assembly that it represents, the view's scale, dimension type, and configuration.

To examine or modify drawing view properties:

Right-click a drawing view on the sheet or in the FeatureManager design tree, and select **Properties**.

❑ **View Information**. Displays the name and type of the selected view.

❑ **Scale**. To change the scale of the view, deselect **Use sheet's scale**, then change the scale ratio value in the **Scale** boxes. The default scale ratio is 1:2.

❑ **Model Information**. Displays the model name and path to the document. If the selected view is a **Named View**, you can specify a different view orientation. Select a standard or user-defined view from the list.
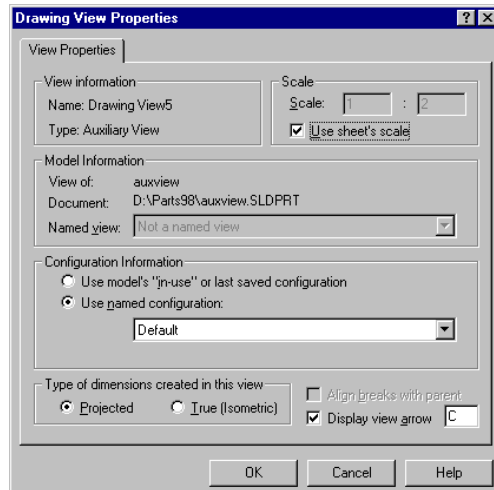
❑ **Configuration Information**. To specify the model configuration to use, click **Use model's "in-use" or last saved configuration**, or click **Use named configuration** then select an existing configuration from the list.

❑ **Show in exploded state**. To use the exploded view of an assembly, select this option. (You must create an exploded view in the assembly document first.)

❑ **Type of Dimensions Created in This View**. To specify the dimension type, select either **Projected** or **True (Isometric).**

❑ **Align breaks with parent**. To align a broken view with the broken view that it was derived from, select this option. (Click **Edit, Rebuild**, if necessary.)

❑ **Display view arrow**. To display an arrow indicating the direction of projection, select this option and specify the label (two characters maximum).

Updating Views

The update mode for all the views of a new drawing is controlled by the option **Automatic drawing view update when model changes** on the **Drawing** tab of the **Tools, Options** dialog box. When checked (the default), all views are updated automatically when the model changes. When unchecked, you update the views manually. You can change this option for the current drawing at any time.

In manual view regeneration mode, you can update all the views at once when you are ready, or update them one by one. When the model changes, any drawing views that need to be updated are marked with a light gray crosshatch pattern.

Note: You cannot activate or edit a drawing view that needs to be updated.

To change the update mode in the current drawing:

Right-mouse click the Drawing icon at the top of the FeatureManager design tree, and click **Auto view regeneration**. If there is no check mark, manual view regeneration is enabled.

To manually update drawing views:

1 Change to manual view regeneration mode as described above.

2 To update all the views at once, either:

- Click **Edit, Update All Views.**

- Click **Edit, Rebuild** or to update any views that need to be updated.

- Press **Ctrl-Q** to force a regeneration of the entire drawing.

- Right-mouse click the Drawing icon at the top of the FeatureManager design tree, and click **Update All Views**.

3 To update a single drawing view, either:

- Select the view, then click **Edit, Update View,** or click on the Drawing toolbar. (Use **Tools, Customize** to add the **Update View** icon to the toolbar, if necessary.)

- Right-mouse click the view, and select **Update View**.

You may select the view either in the FeatureManager design tree or on the drawing sheet for either method. You also can **Ctrl**-click several views at once for update in this way.

View Alignment and Display

Moving Views

To move a view:

Move the mouse over the boundary of a view, and drag the view to a new location when the boundary is highlighted. You can also select a view, then move the cursor over the boundary until the move cursor ⌖ appears, then drag. Note the following exceptions:

❑ **Standard 3 View - 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.

❑ **Standard 3 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, Projection, Section,** and **Aligned Section** views are aligned with the parent views from which they are created, and only move in the direction of the projection.

❑ **Broken** views respect the alignment of the view before it was broken.

❑ **Named**, **Detail, Relative to Model,** and **Empty** views may be moved freely on the sheet; they are not aligned to any other views by default.

To break a view's default alignment:

For views that have a default alignment, you can break the alignment and move the views independently.

Right-click inside the view boundary (not on the model), and select **Alignment, Break Alignment** from the menu, or click **Tools, Align Drawing View, Break Alignment**.

To return a view to its default alignment:

You can return a view whose alignment you have broken to its original alignment.

Right-click inside the view boundary (not on the model), and select **Alignment, Default Alignment**, or click **Tools, Align Drawing View, Default Alignment**.

Aligning Views

You can change the alignment of views that are not aligned by default, or views whose alignment you have broken as described in the previous section. You can rotate a view based on a linear model edge, or you can align one view to another.

To rotate a drawing view:

1 Select a linear model edge in a drawing view.

2 Click **Tools**, **Align Drawing View**, **Horizontal Edge** or **Vertical Edge**.

The view rotates until the selected edge is oriented horizontally or vertically.

If there are any views projected from a view you change this way, they update to maintain their projection.

Selected edge
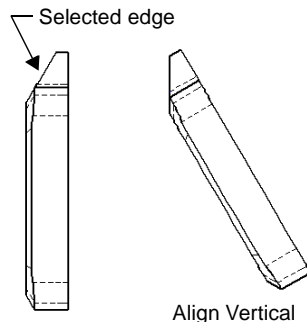
Align Vertical

To align one drawing view to another:

1 Select a drawing view, and click **Tools, Align Drawing View, Horizontal to another view** or **Vertical to another view**.

- or -

Right-click a drawing view, and select **Alignment, Align Horizontal** or **Align Vertical**.

The cursor shape changes to ⬚.

2 Select the reference view to which you want to align.

The centers of the views are aligned along the chosen direction. The alignment is maintained if you move the reference view.

To break an alignment created this way, right-click the view and select **Alignment, Break Alignment** from the menu, or click **Tools, Align Drawing View, Break Alignment**.

Tangent Edge Display

To control the display of tangent edge lines in a drawing view:

Right-mouse click within the view border (not on the model) and select **Tangent Edge**, then **Tangent Edges Visible, Tangent Edges with Font** (as specified on the **Line Font** tab of the **Options** dialog box), or **Tangent Edges Removed**.
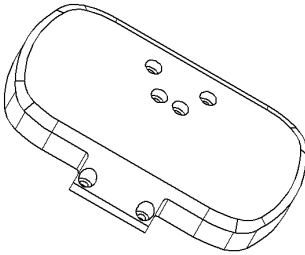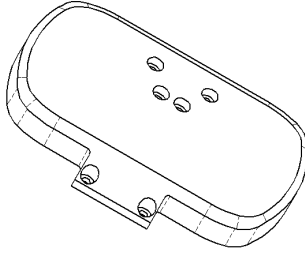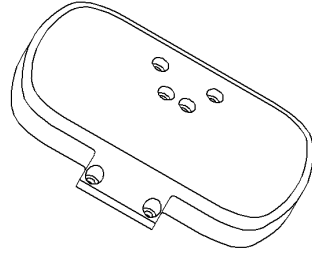
Tangent Edges Visible   Tangent Edges with Font   Tangent Edges Removed

Line Format

The tools on the Line Format toolbar let you modify the appearance of individual lines, edges, and sketch entities in a drawing.

To change the format of a line, edge, or sketch entity:

1   Select the line, edge, or sketch entity you want to change. To select more than one line at a time, press the **Ctrl** key as you select.

2   Click a tool on the Line Format toolbar and change the format as necessary:

   • **Line Color.** Click ⬛, choose a color from the palette, and click **OK**.

   • **Line Thickness.** Click ☰, and choose a line thickness. As you move the mouse over the menu, the thickness name (**Thin**, **Normal**, **Thick**, etc.) is displayed in the Status bar. Corresponding line weights for printing are defined in **File, Page Setup**, on the **Printer** tab, under **Line Weights**.

   • **Line Style.** Click ▦, and choose a style. As you move the mouse over the menu, the corresponding line style name (**Solid**, **Dashed**, **Phantom**, etc.) is displayed in the Status bar.

To return a line to its original format, right-mouse click the line, and select **Reset line font**.

View Visibility

You can use **Hide View** to hide an entire view while working on a drawing. Once hidden, you can use **Show View** to see that view again. If you hide a view that has a dependent detail or section view, you are prompted with the option of hiding the dependent views as well.

You also can selectively hide and show edges in a drawing view.

To hide/show views:

1   Right-mouse click the view on the sheet or in the FeatureManager design tree, and select **Hide View**.

2   To re-display a view, right-mouse click the view, and select **Show View**.

Note: The views that you hide are invisible unless the view is active (the standard red outline is visible), or the menu selection **View**, **Show Hidden Views** is checked (the view boundary is displayed with a gray **X**). The cursor shape changes to and the view boundary highlights when the cursor passes over a hidden view.

To hide/show edges:

Right-mouse click an edge in a drawing view, and select **Hide Edge**. The edge is removed from the display, but it still is highlighted when the cursor passes over it.

To display the edge again, right-mouse click the edge, and select **Show Edge**.

Printing a Drawing

You can print or plot an entire drawing sheet, or just a selected area of the sheet. See Chapter 1, SolidWorks Fundamentals, for basic information about printing and plotting any SolidWorks document.

To print the whole drawing sheet:

1   Click **File, Print**. In the **Print** dialog box, under **Print range**, click **All** or specify the **Pages** you want to print.

2   Under **Scale**, check **Scale To Fit** to print the entire sheet on the page, or deselect the option and specify a **Scale** value (by percent).

3   Click **OK**.

To print a selected area of a drawing sheet:

1   Click **File, Print**. In the **Print** dialog box, under **Print range**, click **Selection**, and click **OK**.

The **Print Selection** dialog box appears, and a selection frame is displayed on the drawing sheet. The frame reflects the current printer settings (page size, orientation, etc.) as defined under **File, Page Setup, Print Setup**.

2   Select a scale factor to apply to the selected area.

- **Model scale (1:1).** This is the default. This means that the selected area is printed at *actual* size; a model dimension of 100mm is printed at 100mm. The default sheet scale is used to calculate the correct print size; therefore, for views that use a scale different than the default sheet scale, you may need to use a custom scale to achieve the desired result.

- **Sheet scale (*n:n*).** The selected area is printed as it appears on the full sheet. If the drawing size and the paper size are the same, the entire sheet is printed. Otherwise, only the selected area is printed, as it would appear on the full sheet.

- **Custom scale.** The selected area is printed at a scale factor you define. Enter the values you want in the boxes, then click **Apply scale**.

When you change the scale factor, the selection frame size changes accordingly.

3   Drag the selection frame to position it over the area you want to print. You can pan or zoom the view, or change the sheet while the selection frame is displayed.

4   Click **OK** to print the selected area.

Sending Drawing Documents

You can send a drawing document to another system using electronic mail. You can choose to send only the drawing itself, or you can send the drawing and all referenced documents. This function uses the electronic mail application installed on your system.

To mail a drawing document to another computer:

1   With a drawing document active, click **File, Send To**.

2   In the **Send Mail** dialog box, click a radio button to select one of the following:

  • Send the current document only.

  • Send the current document and the other document(s) that it references.

    A list of the referenced documents, the number of documents, and the combined size of the documents is displayed.

    Note: The size of the combined documents may be important to you because mailing large size documents may cause transmission problems.

    If you choose to send the drawing and referenced documents, you do not have to send all the documents. You can:

  • Click a filename you do not want to send, and click **Remove**.

  • Click a removed filename and click **Attach** to replace the file.

  • To return all the removed files, click **Attach All**.

3   Click **OK** to send the document(s).

4   Enter your mail password, the mailing address, and any other information requested by your mail application.

2D Sketching

You can create drawing geometry using 2D sketched geometry only, without reference to existing models or assemblies. This sketched geometry may be controlled by relations (collinear, parallel, tangent, and so on), as well as parametric dimensions.

Before starting to sketch geometry, it is often desirable to display a grid. Right-mouse click the active drawing sheet and select **Display Grid** from the menu.

To start sketching, click one of the icons on the **Sketch Tools** toolbar or click **Tools, Sketch Entity**, and select a tool. Then you can:

- Create geometry by selecting **Line, Centerpoint Arc, Tangent Arc, 3 Pt Arc, Circle, Ellipse, Centerpoint Ellipse, Spline, Rectangle, Point**, or **Centerline.**

- Reference existing geometry by selecting **Convert Entities,** which places coincident copies of selected edges to reference from.

- Copy geometry about a centerline using the **Pattern/Mirror** tool.

- Modify the sketch using the **Fillet** and **Trim/Extend** tools.

When you dimension sketched geometry, the geometry automatically updates when you modify dimensions and then click **Edit**, **Rebuild**.

You can move, rotate, or scale sketched geometry on a drawing by using the **Modify** function. Click **Tools, Sketch Tools, Modify** or 🔄 on the Sketch toolbar. The behavior of **Modify** in a drawing is the same as its behavior in a sketch. For more information about using **Modify**, see **Modify Sketch** on page 2-18.

To define relations on 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. Available choices are based on the entities types you picked.

3 Click **OK**.

For more information about adding and deleting geometric relations, see **Geometric Relations** on page 2-28 and **Display/Delete Relations** on page 2-32.

Empty Drawing View

You can create an empty drawing view to contain your sketch geometry. When this view is activated, all sketch geometry added belongs to this view. The sketch geometry can then be scaled, moved, and deleted as a group while still retaining the editability of the individual sketch entity.

To create an empty drawing view:

Click **Insert, Drawing View, Empty**.

Converting Sketched Entities

To convert sketched curves to construction geometry:

1 Select a sketched line, arc, or circle in a drawing.

2 Click **Tools, Sketch Entity, Construction Geometry**.

– or –

1 Right-click a sketched line, arc, or circle on a drawing.

2 Select **Construction Geometry** from the right-mouse menu.

9

Detailing

You can add much of the necessary detailing of your models in the part and assembly documents. This includes dimensions, notes, symbols, etc. Then you can import the dimensions and annotations from the model into a drawing.

Once in the drawing, you can add other annotations, reference dimensions, and a bill of materials if required.

This chapter discusses the following detailing topics:

❑ Selecting detailing options

❑ Dimensioning in drawings

❑ Annotating parts, assemblies, and drawings

❑ Adding a bill of materials to an assembly drawing

Detailing Options

You have a variety of options to customize your documents to meet your company's detailing standards. See Appendix A for more information about the SolidWorks 98 options.

To specify the default detailing options:

1   Click **Tools, Options,** then click the **Detailing** tab.

2   Specify the defaults as needed:

- **Dimensioning Standard.** Select a standard, and choose options for display of broken leaders, dual dimensions, datums, trailing zeroes, alternate section arrow style (ANSI only), and centerline extension (in section views).

- **Notes.** Click **Note Font** to set the font style and size. Click **Balloon** to specify balloon types for notes and BOM callouts. Specify the use of bent leaders and leader anchor.

- **Break Lines.** Specify the gap between break lines in a broken view.

- **Datum Feature Symbols.** Specify the letter label for the next symbol.

- **Center Marks.** Specify the size and line display.

- **Witness Lines.** Specify the gap and extension.

- **Dimensions**. Click **Dim Font, Tolerance** or **Precision** to specify values for those properties. Specify options for use of parentheses on reference dimensions, system separator, snap and centering behavior.

- **Virtual Sharp.** Specify how virtual intersections appear, for example, on a corner where a chamfer or fillet has been added. (This applies to both drawings and sketches.)

- **Arrows.** Specify the **Size** for dimension and annotation arrows. Specify a different **Section/View Size** (for arrows), if needed.
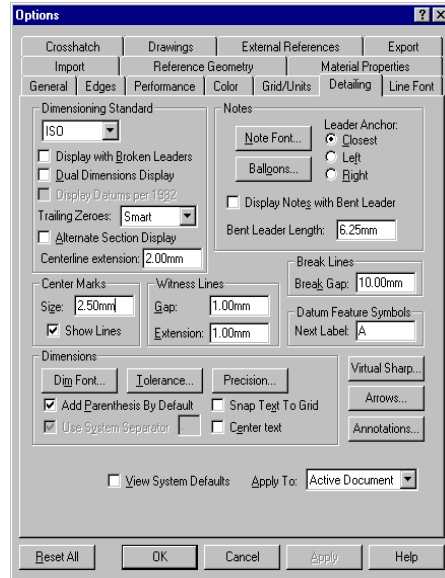
Select the style to use for each **Attachment** condition. Select a style for **Dimension** arrows, and set options as needed for dimension arrow placement.

Specify **Dimension offset distances** for use with baseline dimensions (see **Baseline Dimensions** on page 9-6) and the **Align Parallel** ⊞ tool.

- **Annotations.** Set the **Display Filter** to display the types of annotations you want. Set other options as needed for annotation display.

  If you use JIS dimensioning standards, specify the size (1, 2, 3 characters) or scale for surface finish symbols.

Annotating Drawing Documents

You can add many types of annotations to SolidWorks *part* and *assembly* documents. Additionally, for *drawing* documents, you can:

❑ **Import the annotations, dimensions, and reference geometry from the assembly and part documents into the drawing.**

Changes that you make to imported annotations and dimensions in the part or assembly document are updated in the drawing automatically. Changes you make to imported annotations and driving dimensions in the drawing are updated in the part or assembly document automatically.

Note: One-way associativity can be set between drawings and models during the software installation. This permits changes to the drawing that do not change the model. This option can only be reset with a new installation of the software.

The imported items are displayed in the **Imported Annotations** color defined in **Tools, Options, Color** tab. You can drag the attachment point along the edge or face where it is attached, but you cannot re-attach an annotation to a different edge/face/vertex or free space. The attachment of the annotation in the drawing does not affect the attachment in the part or assembly.

For more information about importing reference geometry, see **Reference Geometry in a Drawing** on page 9-15.

❑ **Add annotations and reference dimensions in the drawing.**

Annotations and reference dimensions added in a drawing do not affect the part or assembly document. Reference dimensions and annotations are displayed in the **Reference Annotations** color defined in the **Tools, Options, Color** tab.

For more information about creating reference dimensions in a drawing, see **Adding Reference Dimensions** on page 9-6. For more information about creating annotations in any type of SolidWorks document, see **Creating Annotations** on page 9-17.

Dimensions in a Drawing

Dimensions in a SolidWorks drawing are associated with the model, and changes in the model are reflected in the drawing.

Typically, you create dimensions as you create each part feature, then insert those dimensions into the various drawing views. Changing a dimension in the model updates the drawing, and changing an imported dimension in a drawing changes the model.

You may also add dimensions in the drawing document, but these are *reference* dimensions, and are *driven*; you cannot edit the value of reference dimensions to change the model. However, the values of reference dimensions change when the model dimensions change.

By default, imported dimensions are black. This includes dimensions that are blue in the part or assembly document (such as the extrusion depth). Reference dimensions are gray and appear with parentheses. To display reference dimensions without parentheses, *deselect* **Add Parentheses by Default** on the **Detailing** tab of the **Options** dialog box, or right-click an individual reference dimension and *deselect* **Show Parentheses**.
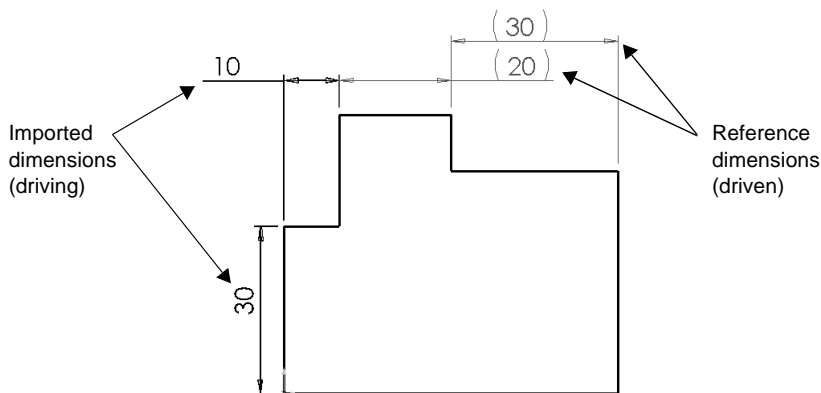
Importing Model Dimensions

You can import the dimensions that belong in a view all at once, or you can selectively import the dimensions for a component or feature.

Dimensions are placed only in the views where they are appropriate. Duplicate dimensions are not imported automatically. Once a dimension has been imported to one view, it is not imported again to another view.

You may delete a dimension from one view, then import it to a different view, or you may move or copy it to another view (see **Moving and Copying Dimensions** on page 9-8).

To insert the model dimensions for a drawing view:

1 Select a view you wish to dimension.

2 Click **Insert, Model Items.**

   The **Insert Model Items** dialog box appears.

3 Select **Dimensions**.

   To display the instance count in a pattern or the revolution count in a helix, select **Instance/ Revolution Counts.**

4 Click **OK**.
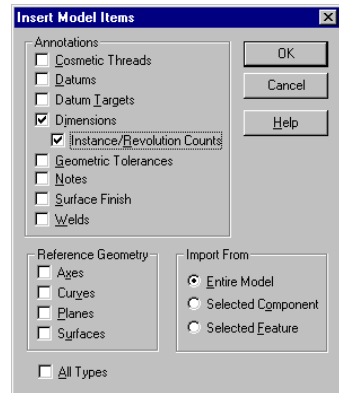
To insert model dimensions for a selected component or feature in a drawing:

1 In a part drawing, click on a face of the feature; in an assembly drawing, click on a component or the face of a feature, *or,*

   Expand the **Drawing View** in the FeatureManager design tree where you want to add the dimensions, and select the component or feature.

2 Click **Insert, Model Items,** and select **Dimensions** (and **Instance/Revolution Counts** if needed).

3 Under **Import From:**

   • Click **Selected Feature** to add the dimensions for only the selected feature.

   • Click **Selected Component** to add the dimensions for the selected component (assembly component or feature of an assembly component selected).

   • Click **Entire Model** to add the dimensions for the entire part or all the assembly components.

4 Click **OK**.

Adding Reference Dimensions

You can use the same methods to add parallel, horizontal, and vertical reference dimensions to a drawing as you use to dimension sketches. See **Dimensioning Sketches** on page 2-22 for more information. Additionally, you can create *ordinate* or *baseline* dimensions in a drawing.

Note: Reference dimensions are not driving dimensions. You cannot edit reference dimensions to change the model. However, when you change the model, the reference dimensions update accordingly.

To add parallel dimensions to a drawing:

1 Click 🖊 on the Sketch Relations toolbar, or click **Tools, Dimensions, Parallel**.

2 Click the geometry to dimension. As you move the cursor around the model, a preview of the dimension type is displayed. The cursor position relative to the attachment points causes the dimension to snap to the appropriate type (horizontal, vertical, linear, radial, etc.).
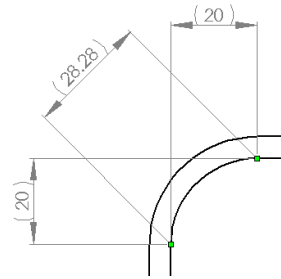
To override the snap behavior, before selecting the points to dimension, right-click and select either **Horizontal Dimension** or **Vertical Dimension** (or click **Tools, Dimensions, Horizontal** or **Vertical**).

3 When the preview indicates the desired dimension type and location, click to place the dimension.

Baseline Dimensions

To add baseline dimensions to a drawing view:

1 Click **Tools**, **Dimension**, **Baseline** (or click 🖊 on the Sketch Relations toolbar, then right-click and select **Baseline Dimension**.)

2 Click the edge or vertex you want to use as a baseline.

3 Click the edges or vertices you want to dimension using the same baseline.

As you click each item, the dimension is placed in the view.
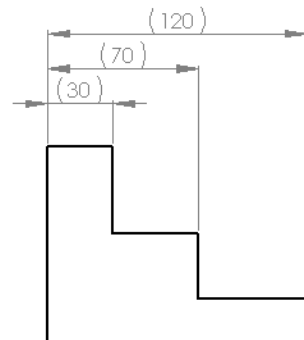
The dimensions are aligned using the spacing values specified for **Dimension offset distances** (click **Tools, Options, Detailing, Arrows** to modify the values).

Baseline dimensions are automatically grouped, so that when you drag any member of the group, all the members move together, maintaining the spacing.

Ordinate Dimensions

Before you add ordinate dimensions, *deselect* the **Single command per pick** option on the **General** tab of the **Options** dialog box, if necessary.

To insert ordinate dimensions:

1 Click **Tools**, **Dimension**, **Ordinate** or **Horizontal Ordinate** or **Vertical Ordinate** (or click [icon] on the Sketch Relations toolbar, then right-mouse click, and select **Ordinate Dimension, Horizontal Ordinate**, or **Vertical Ordinate**).

2 Click the entity at the zero position, and click to place the dimension outside the model.

3 Click the edges or vertices you want to dimension using the same ordinate. As you click each item, the dimension is placed in the view, aligned to the zero ordinate.
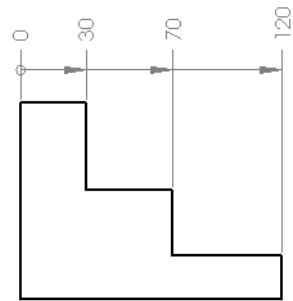
Ordinate dimensions are automatically grouped, so that when you drag any member of the group, all the members move together, maintaining the alignment.

To disconnect a dimension from the alignment group, right-click the dimension, and select **Break Alignment**.

To remove the arrows:

Right-mouse click any of the ordinates, and select **Properties**. Deselect **Display as chain dimension**.

To add additional ordinate dimensions:

1 Right-mouse click any of the ordinate dimensions, and select **Add to Ordinate**.

2 Select an entity to dimension, and click to place the dimension. The added dimension is aligned and grouped to the others in the same ordinate group.

To line up ordinate dimensions:

Right-mouse click any of the ordinate dimensions, and select **Align Ordinate** from the menu. Any ordinates that have been disconnected from the group, and moved, are re-aligned to the zero dimension. However, they are *not* returned to the alignment group. See **Aligning Dimensions** on page 9-9 for more information.

To jog ordinate dimensions to reduce crowding:

1 Right-mouse click an ordinate dimension to move and select **Jog**. A check mark appears next to the menu item.

2 Pull the dimension in any direction. The leader bends so that you can place the dimension freely.

3 To return the dimension to its original place, right-click the dimension, and deselect **Jog**.
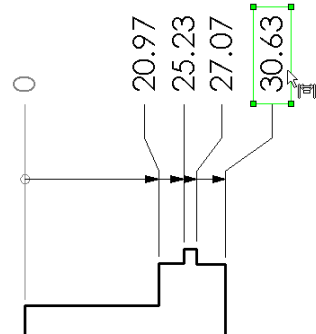
Moving and Copying Dimensions

Once dimensions are displayed, you can move them within a view or to another view. When you drag a dimension from one place to another, the dimension reattaches to the model, as appropriate. You can only move or copy dimensions to a view where the orientation is appropriate for that dimension.

- To *move* a dimension *within the view*, drag the dimension to the new position and release the mouse button.
- To *move* a dimension *from one view to another view*, press the **Shift** key as you drag the dimension to another view, then release the mouse button.
- To *copy* a dimension from one view to another view, press the **Ctrl** key as you drag the dimension to another view, then release the mouse button
- To move or copy several dimensions at once, hold the **Ctrl** key as you select the dimensions.
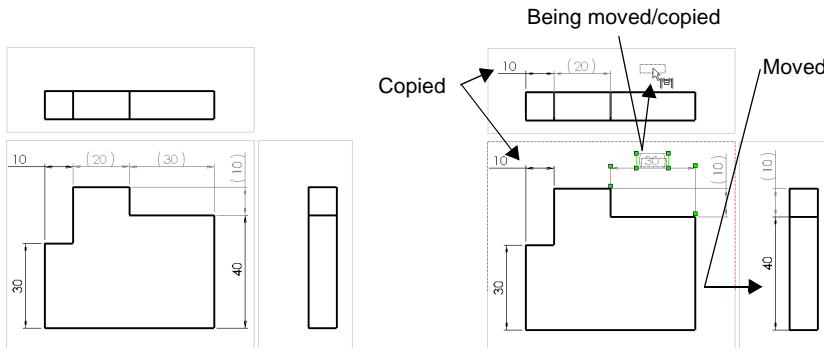
Aligning Dimensions

To distribute parallel dimensions:

1 Select a group of dimensions, either by dragging a rectangle around them, or by holding the **Ctrl** key while selecting them.

2 Click **Tools, Dimensions, Align Parallel,** or ⬚ on the Drawing toolbar.

The dimensions are distributed with the parallel arrows spaced uniformly. To specify the spacing values, click **Tools, Options**, select the **Detailing** tab, and click the **Arrows** button. Under **Dimension offset distances**, enter values for **From model** and **From last dimension**.

The dimensions are grouped so that they move together when you move any one of them, and the spacing is maintained.
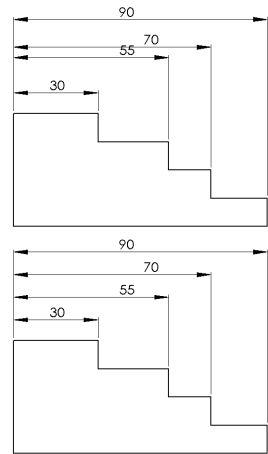
To align dimensions collinear:

1 Select a group of dimensions, either by dragging a rectangle around them, or by holding the **Ctrl** key while selecting them.

2 Click **Tools, Dimensions, Align Collinear,** or ⬚ on the Drawing toolbar.

The dimensions are aligned with the arrows collinear, in line with the last dimension chosen.

The dimensions are grouped so that they move together when you move any one of them.
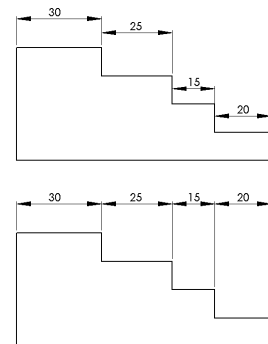
To ungroup dimensions that are aligned (baseline, ordinate, parallel or collinear):

Right-click the dimension to ungroup, and select **Break Alignment**. The selected dimension is disconnected from the group and can move freely; any others that were grouped before remain connected.

To break the alignment of the entire group, select all the dimensions in the group (drag a rectangle around them, or hold the **Ctrl** key while selecting them). Right-click one of them, and select **Break Alignment**.

To see if a dimension is aligned to others:

Right-click the dimension you want to know about. If it is a member of an aligned group, you can select **Show Alignment**. A blue dot appears on each of the other dimensions in the aligned group.

If the selected dimension is not a member of an aligned group, the **Show Alignment** item does not appear on the right-mouse menu.
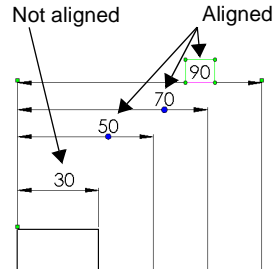
To center dimension text between witness lines:

Right-click a dimension (linear, radial, or angular), and select **Center text**.

If the text is inside the witness lines (or inside the circle, in the case of a radial or diameter dimension), it snaps to the center, and is locked in position between the witness lines. If the text is outside the witness lines, it remains outside. To unlock the text position, right-click the dimension, and *deselect* **Center text**.

You can also drag the text toward the center of the witness lines to snap it to the center. When you center dimension text this way, its position is not locked between the witness lines, so you can move it freely.

Modifying Witness Lines

You can change the default attachment point of dimension witness lines, and you can slant the witness lines if needed.

To change the model attachment point of dimension witness lines:

1   Click a dimension. Handles are displayed on the witness and extension lines.

2   Drag the handle on the *witness* line (at the attachment point) to the desired position or to a vertex (watch for the inferencing cursor). If you choose a vertex, the default witness line gap is used.
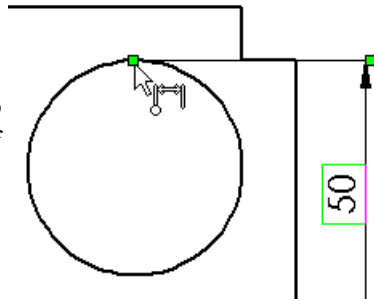
To return the witness line to its original position, right-click the dimension and select **Default Witness Snap**.

To slant witness lines:

1 Click a dimension. Handles are displayed on the witness and extension lines.

2 Drag the handle on the *extension* line to slant the witness lines as needed.

 To return the dimension to its original position, right-click the dimension and select **Remove slant**.
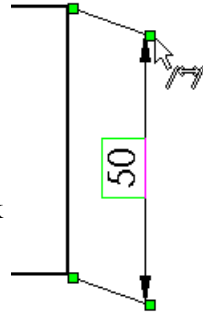
Modifying Dimension Values

To modify a dimension value in a drawing:

1 Double click the text of the dimension. The dimension value is displayed in the **Modify** box.

2 Use the spinbox arrows to change the value, or enter a new value and press **Enter**.

3 Click ✓ to accept the current value and exit,

 - or -

 Click ✗ to restore the original value and exit.

 Note: Click ±? to reset the spin box increment.
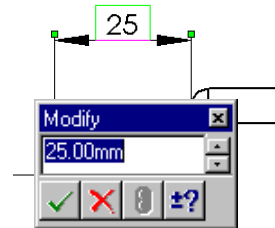
Dimension Properties

You set the default properties for dimensions on the **Detailing** tab of the **Options** dialog box. See **Detailing Options** on page 9-2, or Appendix A, for more information.

You can modify properties of selected 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 as described below.

3 Click **OK**.

General Properties

The following properties are common to all types of dimensions.

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

- **Name.** The name given to a dimension and used in relations associated with the dimension.

- **Full Name.** The full name of the dimension which consists of the **Name**, described above, and the name of the sketch or feature in which it is found (for example, *D2@Sketch1*). The **Full Name** is used in equations and design tables. It is updated when you edit the **Name**.

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

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

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

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

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

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

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

Selection-specific Properties

Depending on the type of dimension (linear, radius, reference, driving, etc.) you choose, some of these additional properties may apply.

- **Driven.** Specifies that the dimension is driven by other dimensions and conditions.

- **Read Only.** Determines whether feature dimensions can be changed or not.

- **Display as Dual Dimension**. Enables dimension display in two different units.

- **Diameter dimension**. Specifies that arc dimensions are diameter, *not radius* dimensions. To convert a diameter dimension to a radius dimension, deselect this option.

- **First / Second Arc Condition**. Choose **Center, Min,** or **Max** to specify the way to dimension distances between an arc or circle and another entity. See **Dimensioning Arcs and Circles** on page 2-24 for more information.

- **Foreshortened radius**. Specifies that the radius dimension line is foreshortened when the centerpoint of a radius is outside of the drawing or interferes with another drawing view.

- **Display with parenthesis**. Specifies whether to display reference dimensions with parenthesis.

- **Display second arrow**. Specifies that a second arrow is used for radial dimensions when **Outside arrows** are selected.

- **Dimension to inside of arc**. Specifies that dimensions to arcs are always to the inside of the arc.

- **Display as chain dimension**. Displays ordinate dimensions with an arrow from the base point to the dimension position.

- **Display with solid leader.** Specifies the display of dimensions with solid leaders.

- **Display as linear dimension.** Allows the display of a radius or diameter dimension as a linear dimension.

Modify Text

You can add a variety of symbols to the text of a dimension.

To change the text of a dimension:

1 In the **Dimension Properties** dialog box, click **Modify Text.**

2 Place the cursor in the **Dimension Text** box where you want to add a symbol or text. The dimension value is represented by **<DIM>** in the middle box.

3 Modify the text as needed. As you make modifications, the text and symbols are displayed in the **Preview** box.

- Add text or symbols before or after the dimension **<DIM>** (middle box).

- Add text or symbols above or below the dimension (upper or lower box).

4 To add a symbol:

    a) Place the cursor in the text box where you want the symbol to appear and click **Add Symbol**.

    b) Select a library from the **Symbol Libraries** list.

    c) Select a symbol name from the list, and click **OK**.

5 To replace the dimension value **<DIM>** if it was deleted accidentally, click **Add Value.**

6 Click **OK**.

Tolerance

To modify the tolerance of a dimension:

1 In the **Dimension Properties** dialog box, click **Tolerance.**

2 Under **Tolerance Display**, select the type of tolerance to display: **None, Basic, Bilateral, Limit, Symmetric, MIN, MAX** or **Fit.**

3 Specify **Maximum / Minimum Variation** values appropriate for the type of tolerance that you selected.

    Linear / Angular Tolerance indicates the type of dimensions to which the current changes apply. This is provided for information only.

4 Specify the **Font**. Either **Use dimension's font**, or deselect the checkbox, then specify the **Font Scale** (a number from 0 to 10.0) or **Font Height**.

Dimension Visibility

You can change the visibility of an individual dimension. Model or reference dimensions may be *hidden* or *shown* to control the appearance of the drawing, temporarily or permanently.

To change the visibility of a dimension:

1 Click **View**, **Hide/Show Dimensions.** The **Show/Hide** cursor appears.

2 Select the dimensions that you wish to hide or show. Select again to toggle between hidden and shown states.

Note: Any previously hidden dimensions re-appear in a dim gray color when you choose **Hide/Show Dimensions** again.

Reference Geometry in a Drawing

The reference geometry contained in a model is hidden by default in drawing views. When you import the reference geometry, all entities of the selected types are shown.

To import reference geometry:

1 Click a view, then click **Insert, Model Items**.

2 In the **Insert Model Items** dialog box, under **Reference Geometry**, select the types to display (**Axes, Curves, Planes, Surfaces**), then click **OK**.

Planes, axes and curves are shown in gray. Surfaces are shown as visible lines (black).

To toggle the display status of an individual entity, right-click the entity in the FeatureManager design tree, and select **Hide** or **Show**.
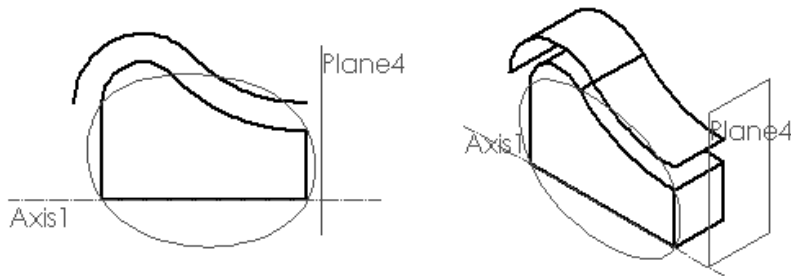
Annotations

Annotations behave like dimensions in each type of SolidWorks document. You can add them in the part or assembly document, then import them to the drawing views, or you can create them in the drawing. See **Annotating Drawing Documents** on page 9-3, and **Moving and Copying Dimensions** on page 9-8 for more information about how dimensions and annotations behave.

Importing Model Annotations

You can import the annotations that belong in a view all at once, or you can selectively import the annotations for a component or feature.

To insert model annotations in a drawing:

1   Select the view you wish to annotate.

2   Click **Insert, Model Items.**

3   In the **Insert Model Items** dialog box, select the annotation types you want to add, or click **All Types** to import all annotations, dimensions, and reference geometry.

4   Click **OK**. The annotations that are appropriate to the selected view are displayed.
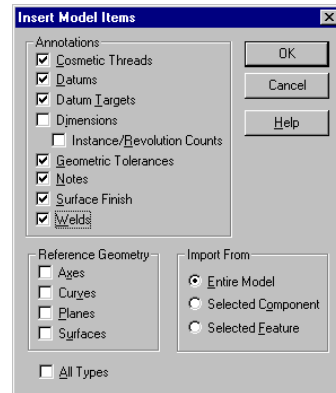
To insert model annotations for a selected component or feature in a drawing:

1   In a part drawing, click on a face of the feature, or in an assembly drawing, click on a component or the face of a feature.

   Alternatively, you can expand the **Drawing View** in the FeatureManager design tree where you want to add the annotations, and select the feature(s).

2   Click **Insert, Model Items,** and select the annotation types to add.

3   Under **Import From:**

   • Click **Selected Feature** to add the annotations for only the selected feature.

   • Click **Selected Component** to add the annotations for the selected component (assembly component or feature of an assembly component selected).

   • Click **Entire Model** to add the annotations for the entire part or all the assembly components.

4   Click **OK**.

Creating Annotations

You can add *all* types of annotations to a drawing document. You can add *most* types in a part or assembly document, then import them into a drawing document. However, there are some types that you can add *only in a drawing document.* Annotations that are limited to drawing documents are indicated as such in the sections that follow.

To add annotations to any SolidWorks document, click **Insert, Annotations** and select from the available annotation types in the pull down menu.

You can also click one of the icons in the Annotations toolbar. Only those annotation types that are appropriate for the selected entity are available for selection; the others are displayed in gray.
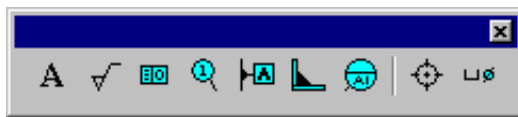
Aligning Annotations

You can control the snap behavior of detail items that you move. Click **Tools, Options**, and select the **Drawing** tab. Under **Detail Item Snapping**, you have these options:

- **Infer when dragging corner**. When checked, you drag the selected item by the *corner handles* to infer to the *corner* of another detail item.

- **Infer when dragging center**. When checked, you drag the *center* of the item to infer to the *center* of another detail item.

- To ignore all inferences, and to prevent the display of inferencing lines, press the **Alt** key while dragging detail items.

To align annotations with neighboring annotations:

1 Click a detail item (note, balloon callout, etc.).

2 Drag the item until inferencing lines appear and the item snaps to align with its neighbor.

3 Release the mouse button.

Inferencing line

Note   A
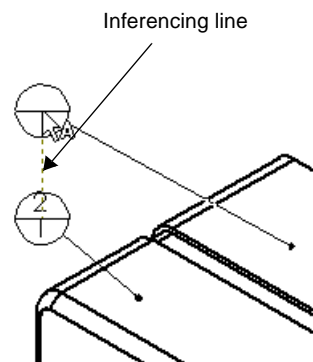
You can create a variety of notes. A note may be free floating or placed with a leader pointing to an item (face, edge, etc.) in the document. It may contain simple text, symbols, parametric text and/or hyperlinks. A note can have a maximum of 1023 characters.

You can set the default note properties (font, arrow and balloon styles, leader display, etc.) by clicking **Tools, Options**, and the **Detailing** tab.

To create a note:

Dimension name as parametric text.
Resulting note:
2.00 is part height.

1 Select a note location on the drawing sheet, or the attachment point on the model, if the note is to include an attached leader.

2 Click **Insert**, **Annotations**, **Note** or A on the Annotations toolbar.

3 Type the text into the **Note Text** dialog box.

   To add a *parametric* value to a note, select the corresponding dimension. Parametric text changes when the dimension changes.

4 Select the necessary options for **Leader**, **Balloon**, and **Font**.

5 If a modifying symbol is needed, click **Add Symbol**. In the **Symbols** dialog box, click **Use Symbol**, select a library from the **Symbol Libraries** list, then select the symbol, and click **OK**.

6 Click **OK**.

7 If you selected **Display with Leader**, select the note location on the drawing.
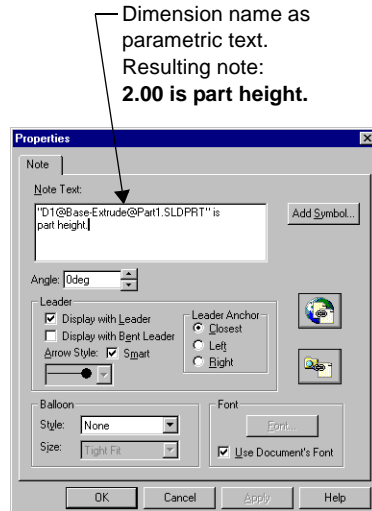
You can cut and paste a note along with its leader by selecting the note and using **Ctrl-X** and **Ctrl-V**. You can copy and paste a note using **Ctrl-C** and **Ctrl-V**.

To edit a note:

1 Either double-click the note, or right-mouse click the note and select **Properties** from the menu.

2 Edit the properties as needed, and click **OK**.

Links to Custom Properties

If you define *custom file properties* in the document, you can link the text of a note to the value of the property. Then if the value of the property changes, the note text changes automatically.

To link a note to a property:

1   In the **Note** dialog box, click **Link to Property** 🔗.

2   Click the dropdown list under **Property Name**. Any properties that have values assigned are shown in the list.

    If you want to add or change a property, click **Edit Properties**. See **Custom File Properties** on page 5-19 for more information.

3   Select the property from the list, and click **OK**. The **Note Text** shown in the dialog box is **$PRP:"*propertyname*"**. However, in the part document, the note text is the *value* assigned to the property.

Hyperlinks

You can add an embedded hyperlink in a note, or you may add a floating hyperlink in any SolidWorks document. The hyperlink may be to a document on the Internet, your local network, or on your own hard drive.

If the link is to a website, selecting the hyperlink launches a web browser if one is not already running, and opens the associated URL (website). If the link is to a SolidWorks document, clicking the link opens the document. If the link is to another type of document, such as a Word or Excel file, clicking the hyperlink launches the appropriate application, and opens the document.

The actual path of a hyperlink embedded in a note does not appear in the note text, A hyperlinked note is displayed in blue; when the mouse moves over a note that contains a hyperlink, the cursor changes to a hand, and the tooltip indicates the path of the link.

A floating hyperlink (one that is not attached to note text) displays the full path of the link (for example, **HTTP://www.solidworks.com**) in blue.

To create an embedded hyperlink:

1   Create a **Note** as described above, and click
    Insert Hyperlink 🌐 in the **Note** dialog box.
    You can also add a hyperlink while editing a
    note's properties.

2   Enter the path (or browse to the document)
    to which you want to link the note text. For a
    World Wide Web location, enter the website
    name in the format **www.*website.com.***
    (You do not need to enter **HTTP://**)

3   Click **Use relative path for hyperlink** if you want to the specify a directory path
    relative to the current location instead of the full pathname.

4   Click **OK** to accept the path.

To create a floating hyperlink:

1   Click **Insert, Hyperlink**.

2   Enter the path (or browse to the
    document) and click **OK**.

    If you edit the properties of a floating hyperlink, you will see that it is a note,
    the text of which is **$URL** (a variable representing the path of the link).

    Because clicking a hyperlinked note activates the link, moving or editing a
    hyperlinked object can be difficult. Move the cursor over the object until you
    see the annotation cursor ⟋ₐ, then drag the object to move it, or right-click the
    object to access its properties.

Notes with multiple leaders

You can select multiple attachment locations for a note. You can add, move, and
remove leaders of existing notes.

To create a note with multiple leaders:

1   Click the attachment points while holding the **Ctrl** key.

2   Click **Insert**, **Annotations**, **Note**, and proceed as described above.

To add more leaders to an existing note:

1   With *nothing* selected, move the cursor over the attachment point of a note's
    leader until you see the ⟋ₐ cursor.

2   Press the **Ctrl** key, drag the leader to a new attachment point, and release.

To delete extra leaders:

1 Select the note. The text and the attachment points are highlighted.

2 Click the attachment point on the leader to delete, watching for the ⬚ cursor.

3 Press the **Delete** key.

Center Mark ⊕

(Drawings only)

You can place axis lines for showing center marks on circles or arcs that can be used as reference points for dimensioning. You set the default size of center marks and choose whether or not to display extended axis lines on the **Detailing** tab of the **Options** dialog box.

To create center marks:.

1 Click ⊕ on the Annotations toolbar, or **Insert, Annotations, Center Mark**.

2 Click the circles or arcs.
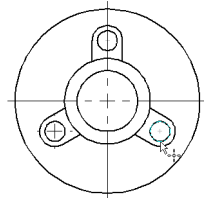
To change the display attributes of individual center marks:

1 Right-mouse click the center mark and select **Properties**.

2 Click to uncheck **Use document's defaults**.

3 Make the changes that you want and click **OK** to accept the changes; click **Cancel** to close the dialog box without saving any changes.

Hole Callout ⌐∅

(Drawings only)

You can add hole callout symbols to a drawing view.

1 Click **Insert, Annotations, Hole Callout** or ⌐∅ on the Annotations toolbar.

2 Click the edge of a hole.

The **Modify Text of Dimension** box appears, with symbols appropriate to the selected hole. See **Modify Text** on page 9-13 for more information.

3 Modify as necessary, and click **OK**. The **Hole Callout** tool remains active, and you can continue to add callouts to other holes until you select another tool.
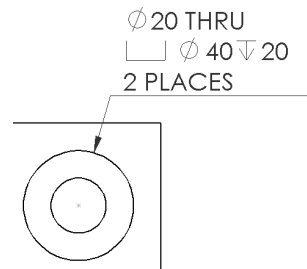
Cosmetic Thread

You can represent threads on a part, assembly, or drawing, and you can attach a thread callout note.

A cosmetic thread differs from other annotations, in that it is a *feature* contained in the **Annotations** folder in the FeatureManager design tree.

Cosmetic threads added in a part or assembly can be imported to a drawing view. If you add a cosmetic thread while working in a drawing view, the part or assembly is updated to include the cosmetic thread feature.

To insert cosmetic threads:

1 On a cylindrical feature, click a circular edge where the thread begins.

2 Click **Insert**, **Annotations, Cosmetic Thread**.

3 In the **Cosmetic Thread** dialog box, select the thread to apply:

- Select **Blind** to run the thread for a specified distance, and enter the distance in the value box.

- Select **Up to Next** to run the thread to the next face.

4 Enter a value in the **Minor Diameter/Major Diameter** box.

5 In the **Thread Callout** box, enter thread callout text, if desired.

6 Click **OK**, then click **Rebuild** or ![icon] on the Standard toolbar to update the part.
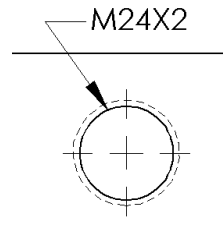
To edit a cosmetic thread:

1 In a part or assembly document, expand **Annotations** folder of the FeatureManager design tree.

2 Right-click the **Cosmetic Thread** feature, and select **Edit Definition.** The **Cosmetic Thread** dialog box appears.

3 Edit as needed, click **OK**, then click **Rebuild** ![icon].

To edit the thread callout on a cosmetic thread:

If you only need to edit the text of the thread callout note, you can double-click the note (or right-click the note and select **Properties**) in the drawing view, and edit it as needed in the **Properties** sheet for **Note**. See **Note** on page 9-18 for more information.

Surface Finish Symbol √

You can specify the surface texture of a part face by using a **Surface Finish Symbol**. You select a model surface, then define a symbol that specifies the values and options for the finish that you want.

To insert a surface finish symbol:

1 Click a face or edge in a part, an assembly, or a drawing view.

2 Click √ on the Annotations toolbar or **Insert, Annotations, Surface Finish Symbol.**

3 Construct the symbol for the selected surface using the options in the **Surface Finish Symbol** dialog box. As you build the symbol, a preview is displayed.

- **Symbol.** Select a symbol type from **Basic**, **Machining Required**, **Machining Prohibited**, or one of the **JIS** types.

- **Direction of Lay.** Select the direction of the surface pattern from the list.

- **Roughness.** Specify the **Maximum**, and **Minimum** height deviation from the mean plane of the surface, and the **Spacing** between peaks and valleys that form the texture of the surface.

- **Special Requirements.** Define any necessary requirements, such as **Production Method/Treatment**, **Sampling Length**, or **Other Roughness Values**.

- **Material Removal Allowance.** Specify the amount of stock to be removed by machining.

- **Leader.** Check **Show** to display the symbol with a leader. Check **Bent Leader** to allow the leader to have a bend. For **Arrow Style,** you can use the **Smart** option, or deselect the option, and choose a different type of leader end from the list.

- **Rotated.** Check this box to rotate the symbol 90°.
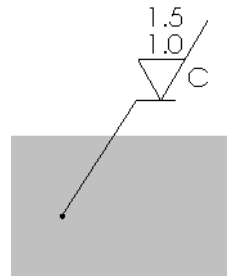
4 Click **OK** when you are satisfied with the surface finish symbol.

You can drag a surface finish symbol with a leader to any location. If you attach a leaderless symbol to an edge, then drag it off the model edge, an extension line is created.

To edit a surface finish symbol, double-click the symbol (or right-click the symbol and select **Properties**) and make the necessary changes in the **Surface Finish Symbol** dialog box.
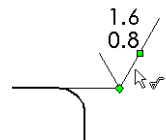
Datum Feature Symbol ⊩▣

You can attach a datum feature symbol to the following items:

- In a part or assembly, on a planar model surface or a reference plane

- In a drawing view, on a surface that appears as an edge (not a silhouette) or a section view surface

- A geometric control frame

To insert a datum feature symbol:

1 Click ⊩▣ on the Annotations toolbar, or **Insert, Annotations, Datum Feature Symbol.** The cursor changes to ⥤▣.

2 Click the surface on which you want to place a datum feature symbol.

   The datum letters are assigned alphabetically (the letters **I, O, Q, S, X,** and **Z** are omitted in ANSI only).
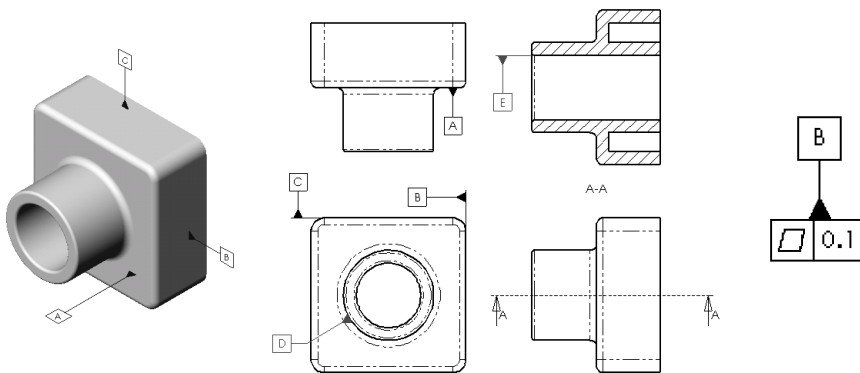
If you drag a datum feature symbol off the model edge, an extension line is added automatically.

To edit the datum feature symbol:

1 Double-click the symbol (or right-click the symbol and select **Properties**).

2 Edit as needed:

   - Enter a new letter in the **Label** box to change the label.

   - Click **Display with filled triangle** to use a filled triangle at the base; otherwise, an open triangle is used.

   - Click **Display with shoulder** to allow a bent leader.

3 Click **OK** to save your changes; click **Cancel** to exit without saving the changes.

Datum Target

You can attach a datum target and symbol to a model face or edge in any document. Both the target (point or area) and the associated symbol are created in one step.

To create a datum target and symbol:

1 Click a model edge or face.

2 Click 🔵 on the Annotations toolbar, or **Insert, Annotations, Datum Target**.

   The **Datum Target Symbol** dialog appears.

3 Select a target shape: **Point** ☒ , **Circle** ⌀ , or **Rectangle** ▨ .

   If you choose circle or rectangle, enter the target area size in the boxes.

4 Select options as necessary:

   • **Display target area size outside.** Places the size values outside the target symbol balloon.

   • **Display target area.** Displays a crosshatch area of the specified size at the selected location.

   • **Display datum target symbol.** Displays a balloon containing the symbol information.

5 Enter the **Datum Reference** labels in the boxes.

6 Specify the **Leader Information**:

   • Check **Bent Leader**, if appropriate.

   • Check **Smart Arrow** to use the default attachment arrows, or deselect the option, then choose an **Arrowhead Style** from the list.

   • Select a **Line Style** (**Solid** or **Dashed**).

7 Click **OK** to accept the datum target symbol you constructed; or click **Cancel**.
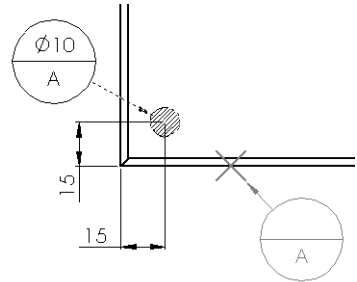
To change the location of the datum target symbol (either the balloon or the crosshatch area or point), click and drag.

To edit the datum target symbol, double-click the symbol, or right-mouse click the symbol and select **Properties** from the menu. The **Datum Target Symbol** dialog appears. Make changes to the options as described above.

In a *drawing*, you can dimension to the center of a datum target area. Click either the symbol or the area to add the dimension.

Note: The drawing view must be *active* in order to add dimensions to the symbol.

Geometric Tolerancing Symbol   ⊞⊠

SolidWorks supports the ANSI Y14.5 Geometric and True Position Tolerancing guidelines. You define the control symbols that are placed in the drawing in the **Geometric Tolerance** dialog box. The **Geometric Tolerance** dialog box displays the feature control frame as you develop it. You can also create a composite geometric tolerance frame.

To create a geometric control frame:

1   Select a model entity or a dimension. If you choose an edge, the leader of the frame will be normal to the selected edge.

2   Click **Insert**, **Annotations**, **Geometric Tolerance** or ⊞⊠ on the Annotations toolbar.

3   In the first **Feature Control Frame** area, click **GCS (Geometric Characteristic Symbol)**. In the **Symbols** dialog box, under **Symbol Libraries**, make sure **Geometric Tolerancing** is selected. Choose a symbol, and click **OK**.

     As you make each selection, a preview of the resulting frame is displayed in the box.

4   Enter a tolerance value in the **Tolerance 1** box. Click ⌀ if you want to include a **Diameter** symbol.

5   Click **MC**, choose the material condition symbol for **Tolerance 1**, and click **OK**.

6   Repeat for **Tolerance 2.**

7   Enter tolerance values and material condition symbols for the **Primary**, **Secondary**, and **Tertiary** datums.

8   If you are creating a composite frame, enter the values for the second **Feature Control Frame**, then click the **Composite Frame** check box.

9   Click the **Show PTZ** check box if you want to enter a **Projected Tolerance Zone** height. Enter a value in the **Height** box.

10 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.

11 When you are finished, click **OK** and drag the geometric tolerance frame to the desired position.
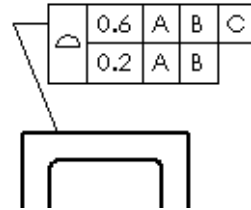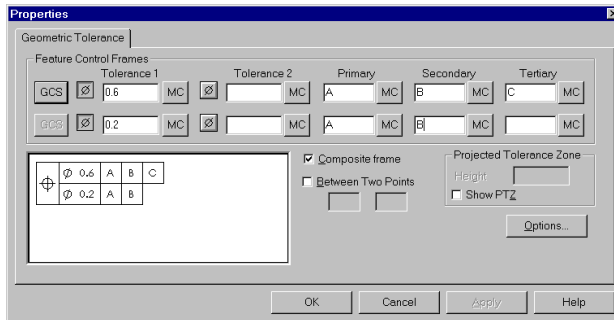
To edit a tolerance control frame:

1 Double-click on the control frame (or right-click the frame, and select **Properties**).

The **Geometric Tolerance** dialog box appears with the current values filled in.

2 Edit as necessary.

3 Click **OK** to update the geometric control frame.

To attach a geometric tolerance frame to a dimension:

You can attach a geometric tolerance frame to a dimension in a drawing view.

1 Create a geometric tolerance frame as described above. You can attach the symbol to a model entity, or create a floating frame, not attached to anything.

2 Select the frame. Note the handles at the corners.

3 Drag the frame by a corner handle until the frame is over the dimension value (watch for the dimension cursor), and release the mouse button.

4 Drag inside the frame to reposition it, if needed. You can place the frame either grouped with the dimension text, or outside the witness lines with a leader.

5 To detach the frame, select it and drag a corner handle again. A dimension may have only one attached geometric tolerance frame.
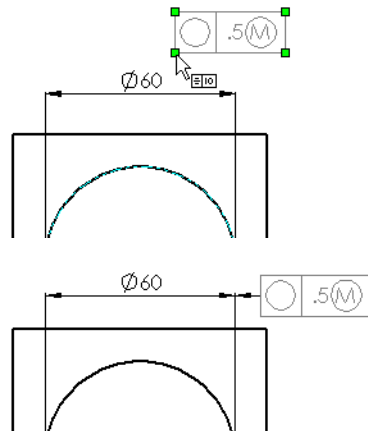
Bill of Materials

(Drawings only)

The **Bill of Materials** menu option lets you insert a bill of materials into the drawing of an assembly.

Note: You must have the Microsoft Excel 97 spreadsheet program installed on your computer in order to insert a bill of materials (BOM) into a drawing.

If you select **Automatic update of BOM** in **Tools, Options**, on the **Drawings** tab, assembly changes (adding, deleting, or replacing components), are updated automatically. Otherwise, you may need to click **Rebuild** 🔲 to update the BOM.

To insert a bill of materials into a drawing:

1   With a drawing view selected, click **Insert, Bill of Materials**.

2   Choose the way to list subassemblies and their components in the bill of materials (for assigning item numbers):

- **Show parts only.** Subassemblies are not listed as items; subassembly components are listed as individual items.

ITEM NO.	QTY.	PART NO.
1	1	box
2	3	handle
3	2	divider1
4	1	divider2

- **Show top level subassemblies and parts only.** Subassemblies are listed as items; subassembly components are not shown.

ITEM NO.	QTY.	PART NO.
1	1	box
2	3	handle
3	1	divider_assembly

- **Show assemblies and parts in an indented list.** Subassemblies are listed as items; subassembly components are listed below the subassembly item to which they belong, but are not given an item number.

ITEM NO.	QTY.	PART NO.
1	1	box
2	3	handle
3	1	divider_assembly
	2	divider1
	1	divider2

3   Choose from these options:

- **Use Summary Info title as part number**. If you assigned a part identifier number in the title box of the **Summary Info** for the part (under **File, Properties**), use that identifier in the **Part No.** column of the bill of materials. Otherwise, the part name is used.

- **Add new item by extending top border of table.** Choose this option if you want new components to be added at the *top* of the BOM instead of at the *bottom*.

4   Click **OK**. A bill of materials is displayed that lists the parts in your assembly. The default note font for the drawing is used for the text.

To edit the bill of materials:

1 Double-click the bill of materials. While the bill of materials is active, it is displayed with shaded borders, and row and column headers. Excel toolbars replace the SolidWorks toolbars.

2 Edit the worksheet as needed, using Excel functions. You can:
- Edit the text
- Delete columns or rows (but you may not add new ones)
- Adjust the column width
- Change the text alignment (left, right, center) by column
- Change the text font (one per worksheet)

3 Click outside the worksheet to close it and return to editing the drawing sheet.

To move the bill of materials:

1 Click on the BOM. The cursor changes to the move shape.

2 Drag the spreadsheet to where you want to place the bill of materials, then release the mouse button.

To show part or subassembly configurations in the bill of materials:

If you want the bill of materials to use a name other than the document name in the part number column (for example, if you want to use the configuration name for a part or subassembly that has more than one configuration), you need to edit the **Advanced** properties of the configuration.

1 In the part or subassembly document, click the **Configuration** icon at the lower left of the window to change to the Configuration Manager view.

2 Right-click the configuration you want to use, and select **Properties**.

3 Click **Advanced**. Under **Part No. displayed when used in a Bill of Materials**, select one of the following:

- **Use the document name.** Specifies that the part or assembly name is used.

- **Use name below.** When selected, a text box opens, containing a name to be used in the bill of materials. The configuration name is filled in by default. Alternatively, you may enter a name (such as a part number) of your own choosing.
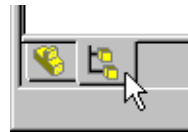
4   If you *always* want a sub-assembly to be listed as a single component in the bill of materials, click **Don't show child components in BOM when used as sub-assembly.**

When unchecked, the child components are listed individually in the BOM, when **Show parts only** or **Show assemblies and parts in an indented list** is selected (see page 9-28).

5   Click **OK**.

To save a bill of materials as an Excel file:

1   Click the bill of materials on the drawing sheet.

2   Click **File, Save As**. The **Save as type** is set to **Excel Files (*.xls)** by default.

3   Enter a file name, and click **Save**.

You can use the saved bill of materials in Excel, or as an OLE object in other applications. The saved file is *not linked* to the bill of materials in the drawing; edits to one do not affect the other.

Custom properties in a bill of materials

You can use *custom file properties* to add more information to a part or subassembly document. Then you can add columns for these properties to the bill of materials template found in **\install_directory\lang\english\Bomtemp.xls**.

To add custom columns to the bill of materials:

1   Add custom properties to the part or subassembly files. See **Custom File Properties** on page 5-19 for information about how to add custom properties.

2   Open the file **\install_directory\lang\english\Bomtemp.xls** (in Excel).

3   In the first empty cell in the first row (**E1**, if this is first time you have edited this file), enter the name of the custom property you want displayed in that column, and press **Enter**.

4   Click the cell to select it, then click **Insert, Name, Define**. The name in the selected cell is listed in the **Names in workbook** list. This name must match the name of the custom property from the part or subassembly document.

5   Click **Add**, then click **OK**. A bill of materials will now include the added column, with the value specified in the custom file properties of the part or subassembly document.

This example uses the custom property **Description**, part numbers were entered in the **Title** box of the file's **Summary Info** (**File, Properties**), and **Use Summary Info title as part number** was checked.

ITEM NO.	QTY.	PART NO.	DESCRIPTION
1	1	240-5892	bracket
2	1	N/A	crank assembly
3	1	001-5432	yoke, male
4	1	001-5431	yoke, female
5	1	003-7790	spider
6	1	120-1121	pin, long
7	2	120-1104	pin, short

Balloon Callout 🔍

You can create balloon callouts in a drawing document. If you previously inserted a bill of materials in a drawing, the balloons label the parts in the assembly and relate them to item numbers on the bill of materials.

You set the default BOM balloon properties (size, style, text) on the **Detailing** tab of the **Options** dialog box.

Balloons are automatically suppressed when the components they reference are suppressed.

Balloons may also be added in part and assembly documents.

To insert balloon callouts:

1 Click 🔍 on the Annotations toolbar, or **Insert, Annotations**, **Balloon**.

2 Click on a part in a drawing view.

A balloon containing the item number on the BOM attaches to the part.

If you choose the **Type** of **Circular Split Line**, you can choose what to display in the **Upper** and **Lower** portions of the balloon: **Item Number**, **Quantity**, or **Custom.**

3 To move the balloon or leader arrow, select and drag the balloon, or drag the leader by the handle.
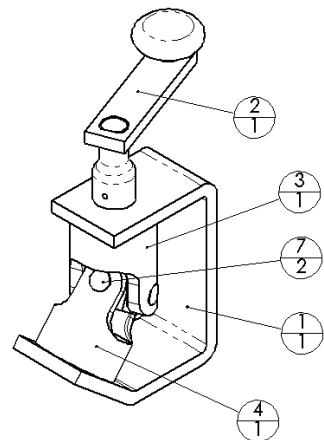
To change the balloon properties:

1 Double-click on the balloon, or right-click the balloon and select **Properties**.

2 Make the changes that you want in the **Note** dialog and click **OK**.

Weld Symbol

When you create a weld bead in an assembly, a weld symbol is created automatically on the **WeldBead** component, representing the parameters you specified. You can edit this symbol to add more information to it. You can also construct weld symbols independently in any SolidWorks document.

See Chapter 12, *Welding*, for more information about creating weld beads.

To insert a weld symbol:

1 Click a face or edge where you want to indicate a welded joint.

2 Click **Insert**, **Annotations**, **Weld Symbol** or click ⬓ on the Annotations toolbar. The **Properties** sheet for **Weld Symbol** appears.

3 In the **Dimension** box, enter a value for the weld dimension.

4 Click the **Weld Symbol** button and select a weld type from the list in the **Symbols** dialog box.

5 Enter additional dimensions or values as needed for this symbol in the box to the right of the **Weld Symbol** button.
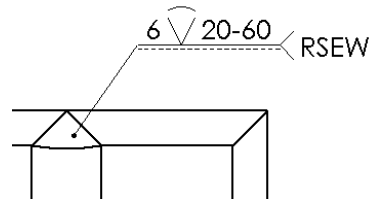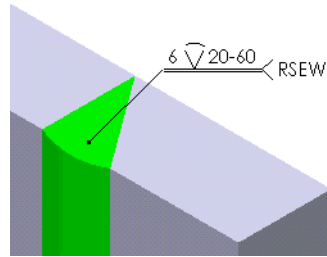
(In this example, the length and the pitch of the weld is entered in the box.

6 Continue constructing the weld symbol by choosing options from the dialog box. A preview of the symbol is displayed as you add more information.

The options available depend on the dimensioning standard you use (ISO, ANSI, etc.).

To edit a weld symbol:

Right-click the weld symbol, and select **Properties**. Edit as needed, and click **OK**.

Area Hatch

(Drawings only)

You can apply a crosshatch pattern to a selected face, or to a closed loop in a sketch.

To add crosshatch to a model face:

1 Activate (double-click) a drawing view.

2 Select a model face.

3 Click **Insert, Area Hatch**.

The selected face is filled, using the default crosshatch pattern specified under **Tools, Options, Crosshatch.**

To delete the crosshatch from a model face:

1 Right-click the crosshatch area, and choose **Select Other** [icon].

2 Right-click the area until the crosshatch is highlighted, then left-click to select.

3 Press the **Delete** key.
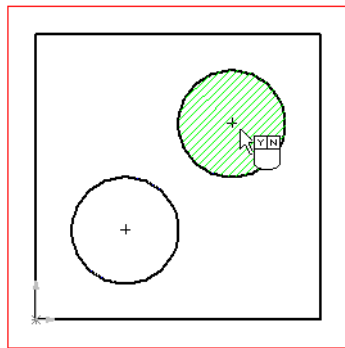
To add a crosshatch area to a sketched area:

1 Sketch a closed profile.

2 Select one segment of the profile.

3 Click **Insert, Area Hatch**.

You can change the shape and size of the sketched profile as needed. The crosshatch updates to fill the modified shape.

To delete the crosshatch pattern, select the pattern, and press the **Delete** key. The sketched profile remains.

To edit the crosshatch pattern:

1 Select the crosshatch area. (If the area is on a model face, activate the view and use **Select Other**.)

2 Click **Edit, Properties**.

3 Choose a different **Pattern** and adjust the **Scale** and **Angle** if needed.

4 Click **OK**.

10

Library Features and the
Feature Palette™ Window

A *library feature* is a feature, or combination of features, that you create once and then save in a library for frequent use.

You can create commonly used features, such as holes or slots, in common sizes, and save them as library features to use whenever you need them. You can use several library features as building blocks to construct a single part. This can save time, and it can help insure consistency in your models.

The Feature Palette™ window helps you organize and use both library features and commonly used parts. The Feature Palette window gives you quick access to the features and parts you use most, by simply dragging and dropping palette items into the SolidWorks window.

This chapter describes:

❑ Creating and editing library features

❑ Adding library features to parts

❑ Organizing features and parts in the Feature Palette window

❑ Inserting palette items in parts and assemblies

Library Features

A library feature can contain one or more extruded bosses or cuts, ribs, fillets, drafts, holes, domes, cosmetic threads, reference planes or axes, or sketches.

You can edit a library feature once it has been placed in a part. After a library feature is added to a part, there is no link between the target part and the original library feature part; if you edit one, the other does not change.

You cannot insert a library feature that includes a base feature into a part that already has a base feature (you cannot have two base features in a single part). Library features usually consist of features built on a base feature, but not the base feature itself.

You can only add a library feature to a part, or while editing a part in the context of an assembly. You cannot add a library feature to an assembly itself.

Creating a Library Feature

To create a library feature, you first create a base feature, then create the features that you want as part of the library feature on the base. You save library features with the **.sldlfp** extension.

To create a library feature:

1   Open a new part, sketch a profile, and create a base.

2   Create the feature(s) you want to have in the library feature.

Dimension the features to the base if you want to use the dimensions to locate the library feature when you place it on the target part.

Library feature with three features: a boss, a hole, and a fillet.

Base

3   In the FeatureManager design tree, select the feature(s) you want to save as the library feature.

To select more than one feature, hold the **Ctrl** key while you select. To select several features in a row, click the first feature, hold the **Shift** key, then click the last feature.

Do not select the **Base** feature unless you plan to insert the library feature into new, empty parts only.

4   Click **File, Save As**. The **Save As** dialog box appears.

5 From the **Save As Type** list, select **Lib Feat Part Files** (***.sldlfp**). Enter a name in the **File name** box and click **Save**.

The part icon beside the part name in the FeatureManager design tree changes to the library feature icon, and each feature that is included is marked with a letter **L**.

Note: To create a library feature from an existing part, open the part, select those features that you want to use in your library feature, and save them as a library feature part (***.sldlfp**).

To edit a library feature:

You can change which features are included as part of a library feature in the library feature part document (**.sldlfp**).

To add another feature, right-click the feature to add, and select **Add To Library**.

To remove a feature, right-click the feature, and select **Remove From Library**.

Adding a Library Feature to a Part

To add a library feature to a part:

1 With the target part open, click **Insert, Library Feature**. The **Insert Library Feature** dialog box appears.

2 Navigate to the directory where the library feature is located, and select the library feature file (**.sldlfp**). The library feature's image appears in the **Preview** area of the dialog box if the **Preview** option is checked.

3 Click **Open.** Two windows and a dialog box appear, and tile automatically:

- the library feature window
- the target part window
- the **Insert Library Feature** dialog box

In the **Insert Library Feature** dialog, there is at least one mandatory **Reference,** and there may be optional references also. A mandatory reference is preceded by an exclamation point; an optional reference is preceded by a question mark. Dimensional references are optional references.

4 To locate the library feature on the target part, click the **Reference** entity (**Plane**, **Edge**, etc.) that is listed as **Mandatory.** When you click the entity on the target part, the exclamation point in the **Reference** area changes to a check mark.
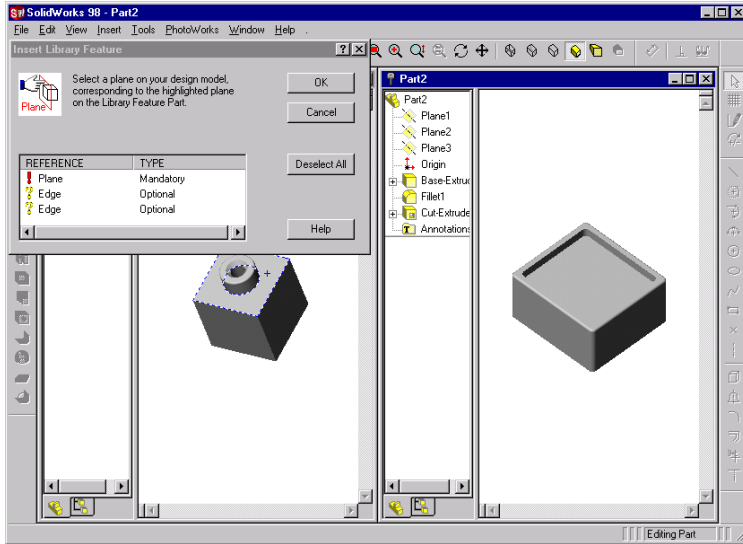
5 Select any optional references. As you click each item in the **Reference** list,
notice that the corresponding entity is highlighted *in the library feature
window*. Select the corresponding entity *in the target part window*.

To deselect an entity, either double-click the
checkmark or click **Deselect All**.

6 Click **OK**.

The library feature is added to the target part.
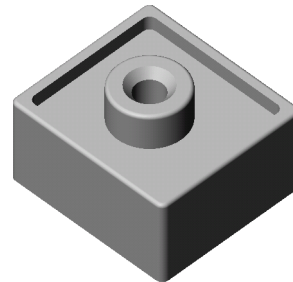
Editing a Library Feature

After the library feature is inserted into a part, you can edit a library feature using
the same techniques that you would use to edit any SolidWorks feature, such as
Edit Sketch or **Edit Definition**.

You can move the library feature to a different location on the target part by
modifying the locating dimensions. Double-click the library feature icon in the
FeatureManager design tree to expose the dimensions.

You can edit an existing library feature and save it with a new name (and **.sldlfp**
extension) to create additional, similar library features.

To delete a library feature from a part, click the library feature icon in the FeatureManager design tree. Press the **Delete** key and click **Yes** to confirm the deletion.

Adding Color

You can apply color 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 Click **Tools, Options**, and select the **Color** tab.

2 In the **Features** box, select **Library Feature**.

3 Click **Shading** in the **System** box, click the **Edit** button and select a color from the color palette (or create a custom color), and click **OK**.

4 To change the **Material Properties** such as **Transparency**, **Shininess**, etc., click the **Advanced** button and adjust the sliders.

5 Click **Apply** and **OK**.

Dissolving a Library Feature

After the library feature is inserted into a part, you can dissolve (break down) a library feature into the individual features it contains.

To break down a library feature into the features it contains:

1 In the FeatureManager design tree, right-mouse click the library feature icon.

2 Select **Dissolve Library Feature** from the menu.

The library feature icon is removed, and each of the features it contains is listed individually in the FeatureManager design tree.

The Feature Palette Window

The Feature Palette window gives you quick access to the features and parts you use most often. It helps you organize features and parts in a way that suits your specific needs.

Palette features and library features are similar in many ways. You create a palette feature in the same way as a library feature. The palette feature is displayed as a library feature in the FeatureManager design tree of the target part. You can dissolve a palette feature the same way as a library feature.

Palette features differ from library features in these ways:

❑ When you add a palette feature to a part, you can simply drag and drop the feature roughly where you want it to be, then place it precisely by re-attaching and modifying any locating dimensions saved in the feature.

❑ You can edit the dimensions of the palette feature as you insert it into the part.

❑ Not all library features are suitable for use in the Feature Palette window. A palette feature is limited to *one* mandatory reference on the target part. Library features that use multiple mandatory references (such as draft features that use several faces) may not be added through the Feature Palette window.

You can also use the Feature Palette window to add parts to an assembly. You can organize parts in logical groups, and you can see the a miniature graphical view of each palette part. This makes it easy to locate, select, and insert the parts you want.

Displaying the Feature Palette Window

The Feature Palette window has several default tabs. Each tab represents a directory; each palette item contained in that directory is displayed as a "thumbnail" graphic. Palette items can include parts (**.sldprt** files) and library features (**.sldlfp** files).

To display the Feature Palette window:

Click **Tools, Feature Palette.** The Feature Palette window stays open, and on top of the SolidWorks window, while you work. If the Feature Palette window is displayed when you exit SolidWorks, it will be displayed when you start SolidWorks the next time.

To dismiss the Feature Palette window, either click ☒ in the upper right corner of the window, or click **Tools, Feature Palette** again to *uncheck* the item in the menu.

Organizing Palette Items

The Feature Palette window has several default tabs. You can create and name new tabs to suit your specific needs, and you can place items on each tab according to the way you want to organize the contents.

Palette items are stored in these directories:

*install_directory***\data\Palette Features***tab_name*

*install_directory***\data\Palette Parts***tab_name*

You add items to a tab by placing **.sldprt** or **.sldlfp** files in the appropriate directory.
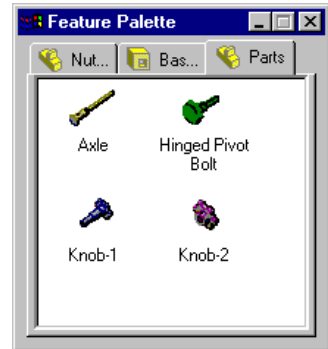
To create a new tab in the Feature Palette window:

1 Using Windows Explorer, navigate to one of these directories:

*install_directory***\data\Palette Features**

*install_directory***\data\Palette Parts**

2 Create and name a new folder. The name you give the new folder will be the name of the new tab.

3 Move or copy the part (**.sldprt**) or library feature (**.sldlfp**) files to the new directory.

4 To view the new tab and its contents, close and re-open the Feature Palette window. The thumbnail view of each item is based on the last saved view.

Note: Empty tabs are not displayed in the Feature Palette window. There must be at least one part or feature in the directory for the tab to appear.

To change the contents of a Feature Palette tab:

1 Navigate to the directory you want to modify.

2 Add, move, or delete the files as needed, using Windows file management commands.

3 Close and re-open the feature palette window to update the tabs.

To delete a tab:

1 Exit SolidWorks.

2 Navigate to the directory for the tab to delete, move the contents to a new location, if necessary, then delete the directory.

Adding a Palette Feature to a Part

Try using either **Hidden Lines Removed** or **Hidden in Gray** view mode, in order to see the preview and dimensions easily.

To add a palette feature to a part:

1   With a part open, click **Tools, Feature Palette**. The Feature Palette window appears.

2   In the Feature Palette window, click the tab that contains the desired feature.

3   Drag the feature from the Feature Palette window and drop it onto the part. Observe the preview as you drag. Drop the feature in the general area of the face where you want the feature to be placed.

The **Edit this Sketch** dialog box appears, and remains open while you position the sketch.

4   Click **Zoom to Selection** 🔍 to get a closer view of the feature.

Modify Sketch cursor

5   Click **Modify Sketch** 📐, and move or rotate the sketch as needed. See **Modify Sketch** on page 2-18 for more information.

If the feature has locating dimensions, they are left dangling, and are displayed in brown.

6   Re-attach the dangling dimensions. Select a dimension, then drag the red handle on the dimension line.

Dangling dimensions

The cursor has the ⃠ shape until it is over a suitable entity for re-attachment. When you release the handle over an edge or vertex, the dimension re-attaches and the handle turns green.

7   Adjust the values of the locating dimensions. (Double-click the dimension, change the value, and press **Enter**.) You can also adjust the locating dimensions in the next step, or later, by editing the appropriate sketch.

8   Click **Next** in the **Edit this Sketch** dialog box. The **Change Dimensions** dialog box appears.

The **Name** and **Value** of each dimension is displayed. If the dimensions are named in the part where you created the library feature, those names are used. Otherwise, the default dimension names *D1* through *Dn* are used. See **Dimension Properties** on page 2-26.
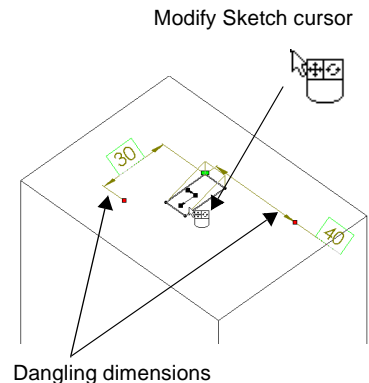
9 To display a feature dimension on the model, click either the **Name** or **Value**.

To modify a dimension, double-click the **Value,** and enter a new value. Click **Apply** to see the changes.

10 When you are satisfied with the dimension values, click **Finish**.

The palette feature **LibFeat** appears in the FeatureManager design tree.
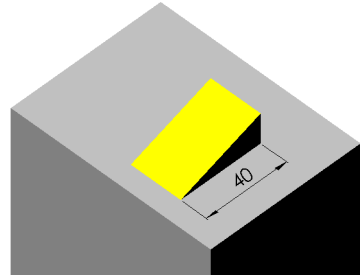
Adding a Palette Part to an Assembly

To add a palette part to an assembly:

1 With the assembly open, click **Tools, Feature Palette**. The Feature Palette window appears.

2 In the Feature Palette window, click the tab that contains the desired part.

3 Drag the part from the Feature Palette window and drop it into the assembly window.

See Chapter 6, *Working with Assemblies*, for information about positioning and mating assembly components.

Palette Parts as Derived Parts

You can use the Feature Palette window to create a derived part, based on the selected palette part.

1 With an *empty* part document open, click **Tools, Feature Palette**. In the Feature Palette window, click the tab that contains the desired part.

2 Drag the part from the Feature Palette window and drop it into the part window.

3 When you are asked if you want to create a derived part:

- Click **Yes** if you want to create a derived part (a part which has another part as its first feature). See **Derived Parts** on page 5-13 for more information.

- Click **No** if you want to edit the palette part itself, stored in *install_directory***\data\Palette Parts***tab_name*. The palette part is opened in the current window. This is the same dragging the part from Windows Explorer into the SolidWorks window.

- Click **Cancel** to abandon the operation.

11

Sheet Metal

This chapter introduces the SolidWorks sheet metal functionality and describes:

❑ Designing sheet metal parts

❑ Bend types

❑ Sheet metal features in the FeatureManager design tree

❑ Rolling back and rebuilding the design

❑ Bend allowances

❑ Auto Relief

❑ Creating a sheet metal parts

❑ Adding additional features

❑ Editing bends

❑ Creating a flat pattern configuration

❑ Sheet metal drawings

Designing Sheet Metal Parts

Sheet metal parts are generally used as enclosures for components or to provide support to other components.

You can design a sheet metal part on its own without any references to the parts it will enclose, or you can design the part in the context of the assembly containing the enclosed components. In the latter case, you can reference the enclosed components to create the geometry for the sheet metal part.

Creating Bends

Before...   ...and after inserting bends

You can create a solid model and then create bends in the model by specifying appropriate bend parameters such as bend radius and allowance.

Inserting bends in the model yields the results shown here. The sharp corner is replaced by a filleted bend.
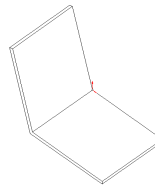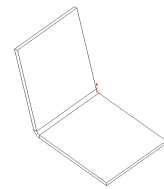
Creating Bend Reliefs, Walls, and Tabs

Walls or tabs can be added to a sheet metal part as designs dictate.

The **Auto Relief** option automatically adds bend reliefs where necessary.
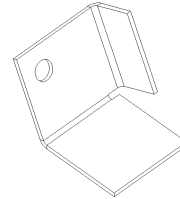
Creating Flat Patterns

When the design is complete (or at any time during the design process) you can create a flat pattern of your sheet metal part.
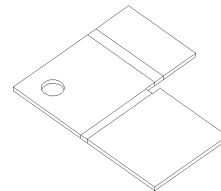
Bend Types

There are three types of bends in SolidWorks: Sharp Bends, Round Bends, and Flat Bends.

Sharp Bends

A *sharp bend* is created by adding bends to a model that was extruded as a thin feature. The original bends were defined in the sketch. Click ![icon] or **Insert, Features, Bends** to create the model on the right. See **Creating a Sheet Metal Part Using Sharp Bends** on page 11-10.
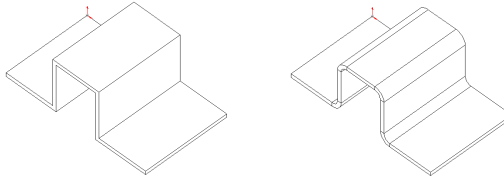
Round Bends

A *round bend* is created from filleted corners created in the sketch or during the thin feature extrusion, using the **Auto Fillet** option. When you click ![icon] or **Insert, Features, Bends**, the model does not change in appearance. However, bend lines and allowances are added to the part, and are visible when the model is flattened. See **Creating a Sheet Metal Part Using Round Bends** on page 11-13.
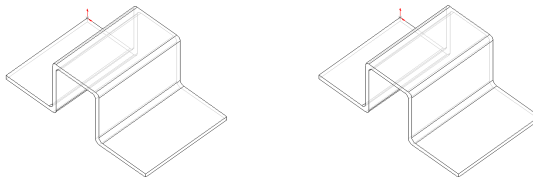
Flat Bends

Flat bends are created from lines sketched in the flat state of a sheet metal part. Bend lines are sketched as shown in the flat model on the left. Exiting the sketch creates the model on the right. See **Creating a Sheet Metal Part from a Flat Model** on page 11-14.
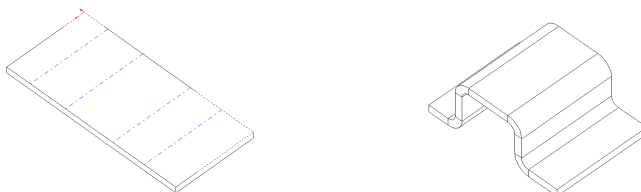
Sheet Metal Features in the FeatureManager Design Tree

When you click 🪝 or **Insert, Features, Bends**, two distinct stages are applied to the sheet metal part. First, the part is flattened and a bend allowance is added. The developed length is calculated, based on the bend radius and bend allowance. Then, the flattened part is refolded to create the formed, folded version of the part. Three new features and icons appear on the FeatureManager design tree that are specific to sheet metal operations. These features are:

❑ Sheet-Metal

❑ Flatten-Bends

❑ Process-Bends

These three features represent what can be thought of as a process plan for the sheet metal part.
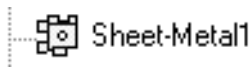
Sheet-Metal1   **Sheet-Metal** contains the definition of the sheet metal part itself and represents the result of the **Insert, Features, Bends** operation.

This feature stores the default bend parameter information (thickness, bend radius, bend angle, and auto relief offset ratio) for the entire part.

Flatten-Bends1   **Flatten-Bends** represents the flattened part. This feature contains information related to the conversion of sharp and filleted corners into bends.

Each bend generated by SolidWorks from the model is listed as a separate feature under **Flatten-Bends**. Bends generated from filleted corner edges are listed as **Round-Bends**; bends generated from sharp corner edges are listed as **Sharp-Bends**.

The **Sharp-Sketch** listed under **Flatten-Bends** is the sketch that contains the bend lines of all sharp and round bends generated by the system. This sketch cannot be edited but can be hidden or shown.
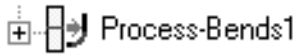
 Process-Bends represents the transformation of the flattened part into the finished, formed part.

Bends created from bend lines specified in the flat are listed under this feature. **FlatSketch,** listed under **Process-Bends**, is a placeholder for these bend lines.

Features that are listed after **Process-Bends** icon in the FeatureManager design tree do not appear in the flattened view of the part. (You flatten the view of the part using Rollback. See **Rolling Back and Rebuilding the Design** on page 11-6.)

For example, in the sheet metal part shown here, the cylindrical boss was added after the **Process-Bends** feature.
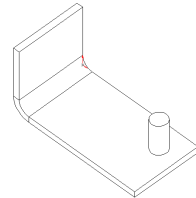
If you rollback to **Process-Bends**, the flattened part does not include the boss.
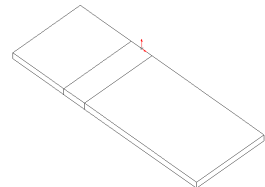
To see the cylindrical boss in the flattened state, reorder the boss to occur before the sheet metal feature.
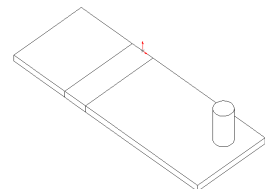
Rolling Back and Rebuilding the Design

A *rollback* reverts the model to the state it was in before the selected feature was created. When you design sheet metal parts, you may use rollback often to revert to previous stages in the design process. This section discusses the ways you can perform a rollback.

To rollback (Method 1):

1   In the FeatureManager design tree, select the feature you want to rollback to.

2   Click **Edit, Rollback**.

To rollback by dragging the rollback bar (Method 2):

1   Place your cursor over the rollback bar in the FeatureManager design tree. The cursor changes to a hand.

2   Click to select the rollback bar. The bar changes color from yellow to blue.

3   Drag the rollback bar up the FeatureManager design tree until it is above the feature you want rolled back,

   - or -
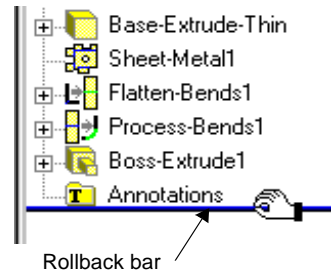
   Use the up and down arrow keys on the keyboard to move the rollback bar up or down. (Check **Arrow key navigation** in **Tools, Options, General** to enable this functionality.)

Note: You do not have to select a feature before you move the rollback bar. Note also that the rolled back icons are grey.
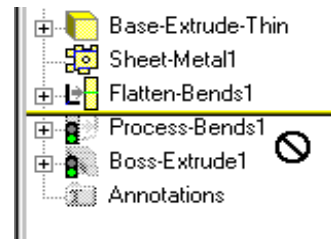
To rollback to before Process-Bends (Method 3):

Click ![icon] the **Flattened** tool on the Features toolbar. Click **Flattened** again to restore the bends.

To rollback to before Sheet-Metal, before any bends were inserted (Method 4):

Click ![icon] the **No Bends** tool on the Features toolbar. Click **No Bends** again to restore the bends.

Throughout the following sections, whenever you need to perform a rollback, you can use one of these methods.

To rebuild (roll forward):

To rebuild (roll forward) your part after performing a rollback, drag the rollback bar down the FeatureManager design tree. You can drag the rollback bar to the bottom of the FeatureManager design tree to rebuild the entire part, or drag it one or more features at a time to step through the regeneration of each rebuilt feature. Throughout this section, whenever you need to perform a rebuild, drag the rollback bar down to the bottom of the FeatureManager design tree.

Bend Allowances

There are three ways to specify a sheet metal bend allowance. You can

- Specify a K-factor
- Specify an explicit value
- Use a bend table

K-Factor

K-Factor is a ratio that represents the location of the neutral sheet with respect to the thickness of the sheet metal part. Bend allowance using a K-Factor is calculated as follows:

$$BA = \Pi/2(R + KT)\ A/90$$

where:

    BA = bend allowance
    T = material thickness
    R = inside bend radius
    A = bend angle in degrees
    t = distance from inside face to neutral sheet
    K = K factor which is t / T
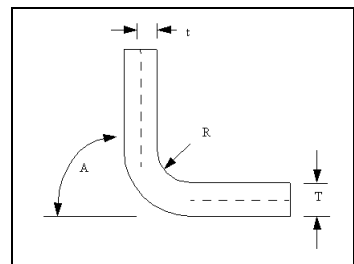
Note: The angle A is the angle through which the material is bent, hence it is not always the angle shown on the drawing.

Explicit Bend Allowance Value

You can specify an explicit bend allowance for any sheet metal bend by entering the value when creating the bend.

Note: For a given bend radius and angle, the specified bend allowance value should be between the length of the inner edge and the outer edge of the bend.

Bend Table

A sample bend table is provided in *lang\english\sample.btl*. To use your own bend table when performing sheet metal operations, copy and rename this table and edit it to specify required bend allowances.

You can edit the bend table using any text editor. A portion of the sample bend table is shown below:

```
 sample.btl - Notepad                                              _ □ X
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.

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.
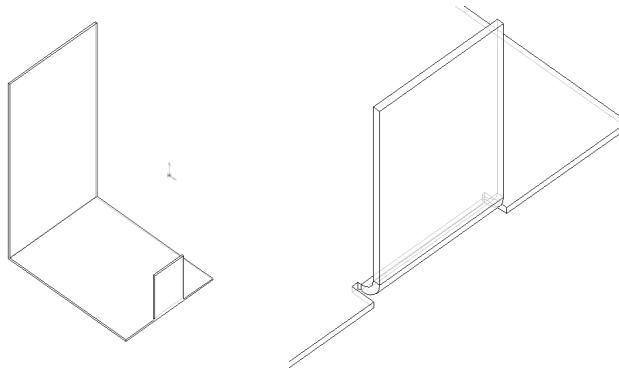
Relief cuts are added automatically at both sides of the wall in this example. (If only one relief cut is needed, only one is added.) The cuts added are of the minimum size required to insert the bend and flatten the part.

When you choose the **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.
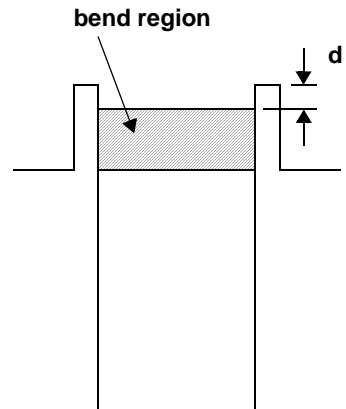
Creating a Sheet Metal Part Using Sharp Bends

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 or **Insert, Base, Extrude**.

3   In the **Extrude Feature** dialog box, specify the desired **Depth** and **Type**.

4   On the **Thin Feature** tab, specify the desired **Type** and **Wall Thickness**.

5   Click **OK**.

   For more information about thin feature extrusions, see **Thin Features** on page 4-6.

To add a wall:

Drag this end away from bend
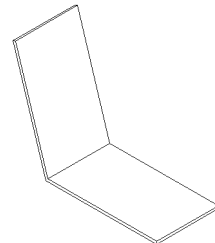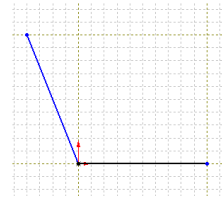
1   Open a sketch on the face of the part where the new wall will be attached.

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 or **Insert, Boss, Extrude**.

5   On the **End Condition** tab, set the **Type** to **Blind** and specify the **Depth**.

6   On the **Thin Feature** tab, specify the same **Thickness** as the base feature.

7   Click **OK**.
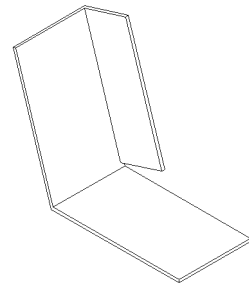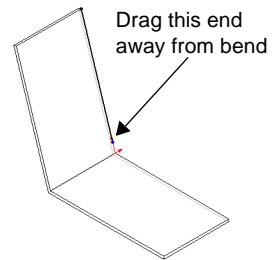
   The new wall is added. It is generally a good practice to add all the walls in this manner before inserting the bends.

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 Click ![icon] or **Insert, Features, Bends**.

The **Flatten-Bends** dialog box appears.

3 Enter a **Default Bend Radius**.

4 Select a **Bend Allowance** type and value:

- If you select **Use Bend Table**, select the name of the table file. See **Bend Table** on page 11-8 for more information.

- If you select **Use K-Factor**, specify the value based on the calculation. See **K-Factor** on page 11-7 for more information.

- If you select **Use Bend Allowance**, specify the value.

5 Make sure **Auto Relief** is checked, and specify an **Offset Ratio**. See **Offset Ratio** on page 11-9 for more information.

6 Click **OK**.

The sharp corner edges of the part are replaced by bends, and the necessary relief cut has been added at the side wall.

7 Double-click the **Flatten-Bends** feature (or click the ⊞ beside its name) to expand the feature.

Notice the **Sharp-Bend** features and a **Sharp-Sketch** feature are listed under it.
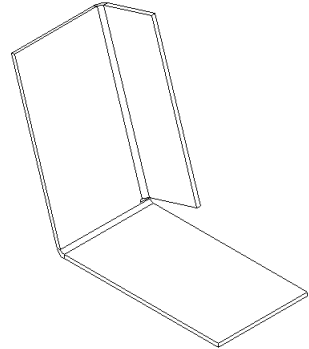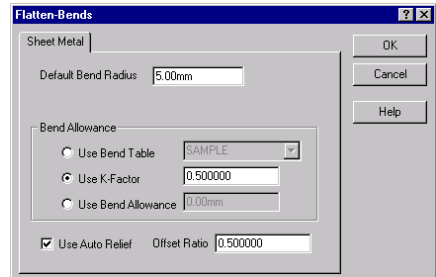
Adding Additional Features

When designing sheet metal parts, it may be necessary to work on the flattened state to add features such as 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 by dragging the rollback bar in the FeatureManager design tree.

To create a cut across a bendline:

1   Click [icon] to rollback above the **Process-Bends** feature.

The part is shown in the flattened state.

2   Select the top face of one of the flat regions and open a sketch.

3   Sketch the profile of the cut.

4   Click [icon] or **Insert, Cut, Extrude**.

5   Set the **Type** to **Through All**, and click **OK**.
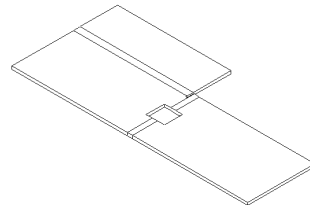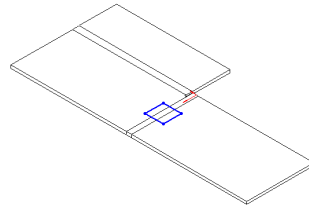
To create a tab:

1   While the part is still rolled back, open a sketch on one of the faces.

2   Sketch a tab with one side coincident with a model edge.

3   Click [icon] or **Insert, Boss, Extrude**.

4   In the **Extrude Feature** dialog box:

   • Click **Link to Thickness** to set the boss thickness equal to the base thickness.

   • Click **Reverse Direction** if the extrusion is extending the wrong way.

5   Click **OK**.

6   Click [icon] to rebuild the part.

The part is folded back with the added cut and tab.
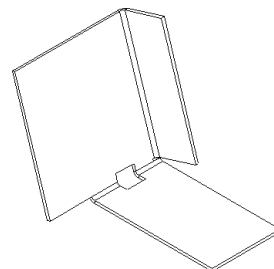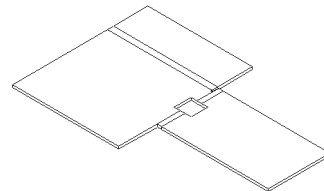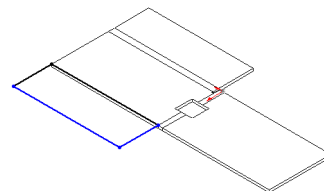
Creating a Sheet Metal Part Using Round Bends

To create a sheet metal part using round bends:

1 Create a thin feature part:

- Check the **Auto Fillet** option in the **Thin Feature** tab of the **Extrude Thin Feature** dialog box.

- Specify the **Fillet Radius** (inner bend radius).

2 Select the fixed face.

3 Click ![icon] or **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**.
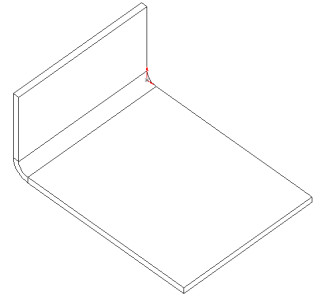
Note that a **RoundBend** appears in the FeatureManager design tree under **Flatten-Bends**.

Creating Hems

Because you can create bends of up to 180 degrees, you can make hems of various types in a sheet metal part. SolidWorks supports open, closed, double, and teardrop hems.

To create a hem:

1 In an existing sheet metal part, click the ⊞ beside a **Process-Bends** feature.

2 Right-mouse click the sketch and select **Edit Sketch**.

3 Sketch a line to define the fold of the hem, and exit the sketch.

The part folds on the sketched line, using the default radius and angle. You need to edit the bend to give it radius and angle values suitable for a hem.

4 Right-click the **FlatBend** feature and select **Edit Definition**.
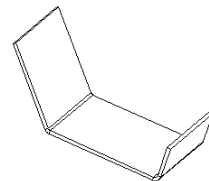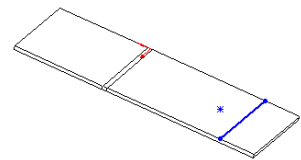
5   Enter new values for **Radius** and **Angle**. The **Angle** may be between 0 and 180 degrees.

> **Note:** The bend radius must allow for a small gap between overlapping faces so that they do not touch.

6   Click **OK**. The hem re-folds using the new values.
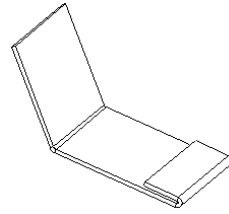
Creating a Sheet Metal Part from a Flat Model

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 ⬚ or **Insert, Features, Bends**.

4   In the **Flatten-Bends** dialog box, enter the **Bend Radius**, and specify the **Bend Allowance** method and value.

The sheet metal features appear in the FeatureManager design tree even though the part appears unchanged.

5   Expand the **Process-Bends** feature in the FeatureManager design tree.

6   Right-click the **Flat-Sketch** under **Process-Bends** and select **Edit Sketch**.

7   Sketch lines or centerlines where you want bend lines.

8   Place a sketch point on the face that you want to remain fixed.

9   Exit the sketch to rebuild the part.

Bends are created using the sketched bendline and the specified bend parameter information.

> **Note:** You cannot specify a bendline for a Flat Bend that lies between existing Sharp or Round bends.
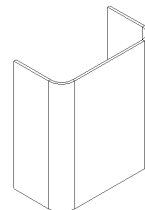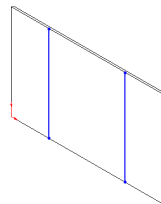
Editing Bends

You can edit bend information for sheet metal features. The scope of the changes depends upon which feature is edited.

To edit bend parameters for the entire part:

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

2 In the **Sheet-Metal** dialog box, change the **Default Bend Radius** or **Bend Allowance** method as desired.

3 Click **OK**.

To edit bend parameters for all bends listed under Flatten-Bends:

1 In the FeatureManager design tree, right-mouse click the **Flatten-Bends** feature and select **Edit Definition**.

2 In the **Flatten-Bends** dialog box, change the **Default Bend Radius** or **Bend Allowance** method as desired.

3 Click **OK**.

To edit bend parameters for all bends listed under Process-Bends:

1 In the FeatureManager design tree, right-mouse click the **Process-Bends** feature and select **Edit Definition**.

2 In the **Process-Bends** dialog box, change **Default Bend Radius** or **Bend Allowance** method as desired.

3 Click **OK**.

To edit bend parameters for an individual bend:

1 In the FeatureManager design tree, right-mouse click the bend you want to edit and select **Edit Definition**.

2 Change **Radius**, **Angle**, **Bend Direction,** and **Bend Allowance** as desired.

3 Click **OK**.

Notes on editing bends:

❑ You cannot change the bend angle for sharp bends. This angle is calculated from the model.

❑ You cannot change the bend angle or radius for round bends. The bend angle is calculated from the model and the bend radius is specified in the sketch or in the definition of the fillet feature.

Creating a Flat Pattern Configuration

Creating a flat pattern is equivalent to suppressing the last **Process-Bends** feature and all features after it. You can save this as a configuration to use in a drawing.

To create a flat pattern configuration:

1 Create and name a new configuration. (See Chapter 5, *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 **Edit, Suppress** or drag up the rollback bar.

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 8, *Drawing,* for more information on creating drawings.

To create a drawing of a flat pattern:

1 Create a part configuration showing a flat pattern.

If none of the standard views shows the flat part in such a way that the bend lines are clearly displayed, create a named view in a suitable orientation.

2 Create a new drawing.

3 Click **Insert, Drawing View, Named View**.

4 Change to the part window, click anywhere, then select the desired view from the **View Orientation** list.

5 Return to the drawing window and click to place the view on the sheet.

To remove display of the bend region lines:

1 In the drawing window, right-click the drawing view in the FeatureManager design tree.

2 Select **Tangent Edge, Tangent Edges Removed**.

To show the bend lines in the view of the flat pattern:

1 Either in the part document or the drawing view, expand the **Flatten-Bends** and **Process-Bends** features.

2 Right-click **Sharp-Sketch** and **Flat-Sketch** in the FeatureManager design tree, and click **Show**.

Tips for Designing Sheet Metal Parts

The order in which you add features is key to getting the desired results. The recommended approach for designing sheet metal parts is as follows:
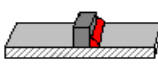
❑ Design the part in the bent-up state, if possible.

❑ Add **Insert, Features, Bends** only after the functional design is complete; that is, add all the features before inserting bends.

❑ If some features such as bend reliefs and additional walls need to be added after inserting bends, add these features after doing a **Rollback** to the **Process-Bends** feature. This step will rollback the model to the flattened state.

❑ If you do not want certain features to be visible in the flattened state (features such as holes that are drilled after the sheet metal part is bent), add these features *after* the **Process-Bends** feature.

❑ To ensure that the part has uniform thickness, use the **Link to Thickness** features in the **Extrude** dialog box.

Welding

This chapter introduces the SolidWorks welding feature and describes:

❑ Weld types

❑ Top surface delta and radius calculations

❑ General procedures for creating a weld

❑ Examples of Creating V-Butt, and Square Butt welds

❑ Editing a weld

Weld Types

You can add a variety of weld types to an assembly. The weld types 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 root		
Single-bevel butt with root		
Single-U butt		
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.
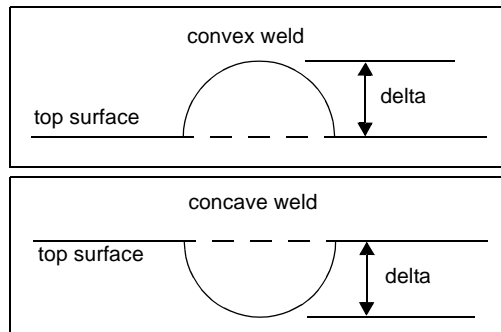
Radius (Fillet Welds Only)

For a fillet weld, you need to specify the radius of the weld. The radius, *r*, is measured from the point where the parts join.
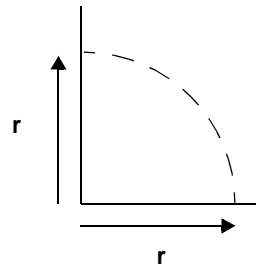
General Procedure for Welding Components

To weld two components:

1 Create an assembly and mate the components, using mates that are suitable to the specific weld type you plan to use. Refer to the section **Examples of Creating Weld Beads** on page 12-6 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**.
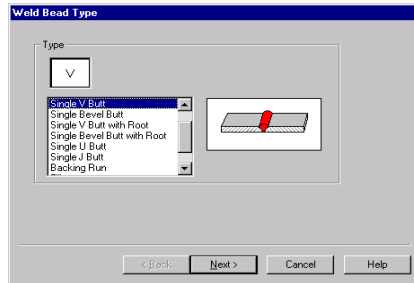
5 In the **Weld Bead Surface** dialog box:

- 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.
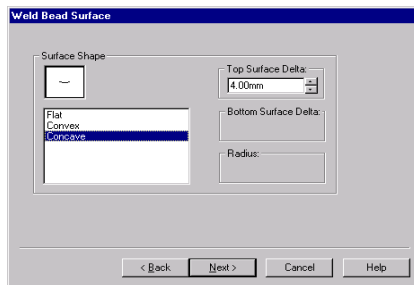
6 Click **Next**.

7 In the **Weld Bead Mate Surfaces** dialog:

- 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.
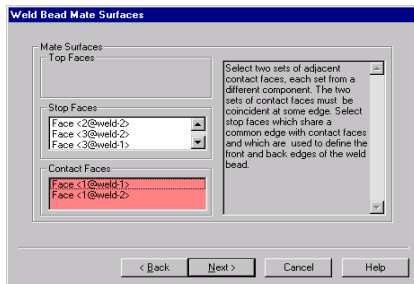
Note: Some **Mate Surfaces** edit boxes may be grayed out because they are not needed for the selected weld bead.

8 Click **Next**.

9 In the **Weld Bead Part** dialog box, accept the default name for the **Weld Bead Part** or enter a new part name, and click **Finish**.

Weld Face and Edge Selection Tips

To make face and edge selection easier, you can,

- View the assembly with hidden lines in gray

- Place the assembly in isometric view orientation

- Edit the distance in a distance mate to enlarge the space between the contact faces temporarily. After you select the faces and complete the creation of the weld bead, you can edit the distance mate again. The weld bead between the two components is updated to reflect the final mating distance.
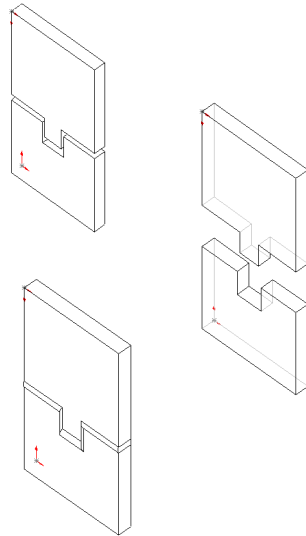
Annotations for Weld Beads

Weld symbols are automatically generated by the system when you create a weld bead.

To move the symbol into position:

1 Click where the arrow meets the weld edge.

   Green drag handles appear on the symbol and the arrow.

2 Drag the symbol by the handle and release the mouse button when it is in the position you want.
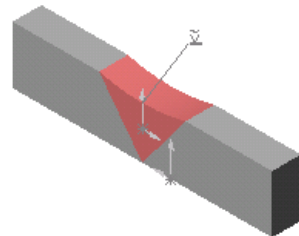
To add the weld symbol to the assembly drawing:

1 With a drawing view active, click **Insert, Model Items**, and select **Welds** from the menu.

2 Click **OK**.

Refer to the **Weld Symbol** on page 9-32 for more information.

Examples of Creating Weld Beads

Example 1: V-Butt Weld

The assembly shown here calls for a V-Butt type weld. The components are mated using three coincident mates:

- Between the edges where the contact faces meet

- Between the top faces

- Between the stop faces on the front of the assembly
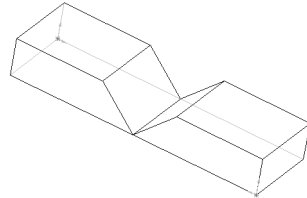
To add a V-Butt weld:

1   Click **Insert, Assembly Feature, Weld Bead**.

2   In the **Weld Bead Type** dialog box, select **Single-V Butt**, and click **Next**.

3   In the **Weld Bead Surface** dialog box, select a **Surface Shape** of either **Flat**, **Concave** or **Convex**, and specify the **Top Surface Delta** value.

4   Click **Next**.

5   In the **Weld Bead Mate Surfaces** dialog box:

- Click the **Contact Faces** box and select the slanted faces of the components.

- Click the **Stop Faces** box and select the two sides of each component (four faces total).

contact faces

stop faces

6   Click **Next**.

7   In the **Weld Bead Part** dialog box, accept the default name for the **Weld Bead Part** or enter a new part name.

8   Click **Finish**.
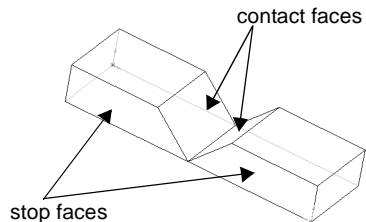
The weld bead is added between the two components.
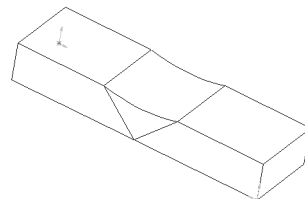
Example 2: Square Butt Weld

The assembly shown here calls for a Square Butt type weld. The components are mated as follows:

- A distance mate between the contact faces

- A coincident mate between the top faces

- A coincident mate between the stop faces

coincident

distance

coincident
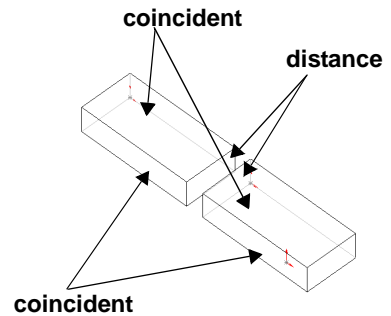
To add a square butt weld:

1 Click **Insert, Assembly Feature, Weld Bead**.

2 In the **Weld Bead** dialog box, select **Square Butt**, then click **Next**.

3 In the **Weld Bead Surface** dialog box, select either **Flat**, **Convex**, or **Concave,** and specify the **Top Surface Delta**. 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 opposite face).

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.
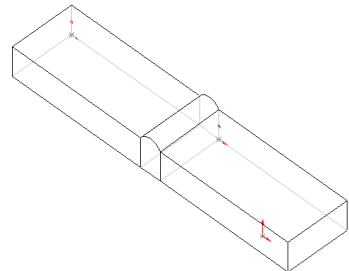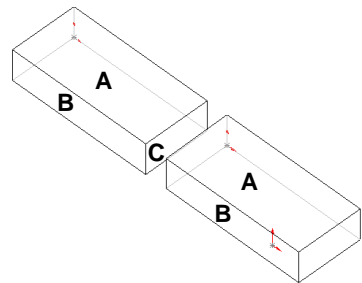
Editing a Weld

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.

Importing and Exporting Files

This chapter describes these SolidWorks file import and export topics:

❑ Setting import/export options

❑ Exporting and importing the following file types:

- IGES

- Parasolid Text and Binary

- STL (Export only)

- ACIS™

- DXF/DWG

- VRML

- TIFF

- STEP

- VDAFS

Exporting SolidWorks Documents

You can export SolidWorks files to a number of formats for use with other applications. Use the **File, Save As** function from the main menu to save the active document to disk with a different format.

❑ **You can save part documents as:**

IGES (.igs)	Parasolid text files (.x_t)	Parasolid Binary files (.x_b)
STL (.stl)	ACIS™ (.sat)	STEP AP203 (.step)
TIFF (.tif)	VRML (.wrl)	VDAFS (.vda)

❑ **You can save assembly documents as:**

IGES (.igs)	Parasolid Text files (.x_t)	Parasolid Binary files(.x_b)
STL (.stl)	ACIS (.sat)	STEP AP203 (.step)
TIFF (.tif)	VRML (.wrl)	

❑ **You can save drawing documents as:**

DXF (.dxf)	DWG (.dwg)	TIFF (.tif)

To export a SolidWorks document with a new format:

Before you export a SolidWorks document, make sure that you have the Export options set as necessary for your purposes. Refer to **Setting Export Options** on page 13-3.

1   Click **File, Save As**.

    The **Save As** dialog appears.

2   Select a format from the **Save as File Type** box.

3   Click **Save**, or click **Cancel** to exit without saving.

Setting Export Options

Before exporting a file make sure the **Options** settings are appropriate for the target application. Options are available for the following file types:

IGES	**Parasolid**	**DXF/DWG**	**VRML**
STL	**ACIS**	**TIFF**	

To set export options:

1 Click **Tools, Options**.

2 Click the **Export** tab.

3 Choose options as desired, and click **OK**.
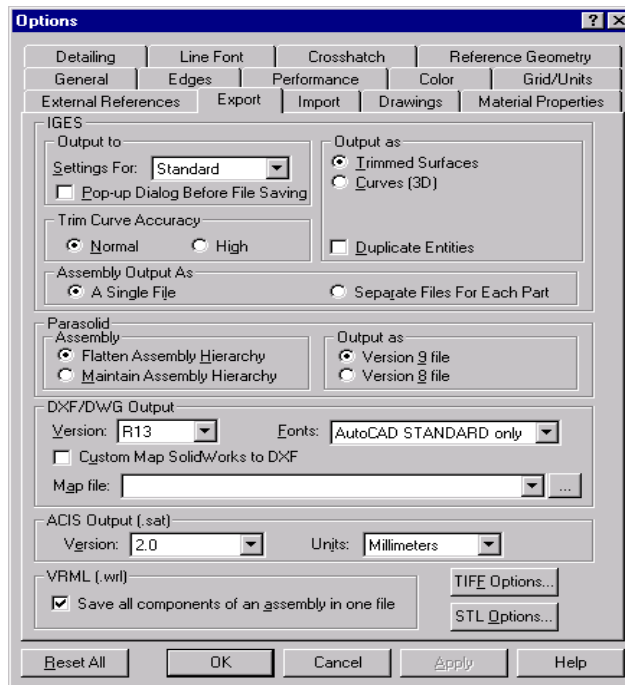
IGES Files

The following IGES entity types are supported for import and export:

IGES Entity Type	Entity Name
408	Singular subfigure instance entity
406 Form 12	External reference file
314	Color definition entity
308	Subfigure definition entity
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
114 Import only	Parametric spline surface
112	Parametric spline curve
110	Line
106 Form 12 Import only	Copious Data - 3-D piecewise linear curve
106 Form 11 Import only	Copious Data - 2-D piecewise linear curve
104 Import only	Conic arc
102	Composite curve
100	Circular arc

If you want the SolidWorks software to try to form a solid from the IGES data for use in SolidWorks, export the solid or surfaces from the source system as trimmed surfaces (Entity type 144) or as untrimmed surfaces (Entity types 128, 122, 120, 118, 114). For best results, export a single solid or set of surfaces which form a single closed volume as trimmed surfaces.

Also, set the number of significant digits (precision) as high as possible to ensure high-fidelity translation.

Exporting IGES Files

You can export SolidWorks part and assembly documents to IGES format.

Note: Parts and surfaces exported to IGES format retain their part/surface color and display in color when in Shaded mode.

To specify default export options for IGES:

Click **Tools, Options**, and select the **Export** tab.

IGES settings

❑ **Output to.** If you choose to output as **Trimmed Surfaces**, the following options are available:

- **Settings For:** Select the mechanical design application program to which you want to export your SolidWorks files.

- **Pop-up Dialog Box Before File Saving.** Check this option if you want the program to display the **Settings For:** list every time you save an IGES file so you can select a different format type. (You do not have to change the **Export** options page each time you export an IGES file to a new format.)

❑ **Output as.**

- **Trimmed Surfaces.** The faces of the 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

Note: If the system you are exporting to is not listed, refer to the manuals of that system to see which entities are supported and choose an appropriate setting.

- **Curves (3D).** The solid body is converted to 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. Because each edge is included once in each of two composite curves forming the boundaries of the faces that meet at each edge, the curves are duplicated. For each pair of duplicate curves, one curve has the opposite sense (direction) from the other.

❑ **Trim Curve Accuracy**. Select **Normal** or **High**. Adjust this setting if you experience problems importing into another system.

- **Normal.** Select this option when smaller file size is preferred and high curve accuracy is not needed.

- **High.** Select this option when high curve accuracy is essential; the file size is larger than when using the **Normal** setting.

❑ **Assembly Output As.** Click either **A single file** or **Separate files for each part**.

  If you select **Separate files** for each part, a new directory is created that has the same name as the assembly. The assembly components are saved as individual IGES files in that directory.

  If there are sub-assemblies in the assembly, a new subdirectory is created for each one, and all components in the sub-assembly are saved as IGES files under the corresponding subdirectory.

To export a SolidWorks part file to IGES:

1 Select the faces or surfaces you want to export. (If no faces or surfaces are selected, the entire solid is exported.)

2 Click **File**, **Save As**.

3 Select **IGES Files (*.igs)** in the **Save 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 or surfaces, indicate whether you want to export the **Selected Face(s)**, **Selected Surface(s)**, or the **Solid Body**, then click **OK**.

To export a SolidWorks assembly file to IGES:

1 With an assembly open and active, click **File, Save As**.

2 In the **Save As** dialog box, scroll the **Save as type** box to select I**GES Files (*.igs)**.

3 Click the **Save** button.

Importing IGES Files

You can import surfaces from IGES files into SolidWorks and use them to:

- Create a base feature from a group of IGES surfaces that form a closed volume

- Trim a part with an imported surface

- Create a base, boss, or cut feature by thickening an imported surface

You can import 3-D curves from IGES files and use them to:

- Create planes used in pierce point relations in sketches

- Create path and guide curves for sweeps and lofts

You can specify the default response that you want SolidWorks to make if there are errors while importing IGES files.

To select default responses to various IGES import errors:

1   Click **Tools, Options**, and select the **Import** tab.

The dialog displays a list of four error types.

2   From the pull-down lists next to the error types, select the default method that you want SolidWorks to use to resolve problems.

- The solutions for **Surface Errors**, **Surface Flaws**, and **Significant Digits** are:
  Ask me what to do (ask for a solution each time a problem occurs)
  Try to knit into solid
  Try to form reference surfaces
  Cancel operation

- The solutions for **Blanked Surfaces** are:
  Ask me what to do
  Ignore blanked surfaces
  Do NOT ignore blanked surfaces
  Cancel operation

3   When you have selected the default responses, click **OK**.

Using Imported IGES Files

To create a base feature or import 3-D wireframe curves from an IGES file:

1   Click **File**, **Open**.

2   Select **IGES Files (*.igs)** in the **Files of Type** box.

3   Select an IGES file.

4   Click **Open**.

If there is wireframe geometry in the file, it will read the IGES data and form curves for the IGES wireframe entities.

If there are surfaces in the file, the surfaces 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 SolidWorks software finds that the surfaces in the IGES file represent multiple closed volumes, then one part will be made for each closed volume. An assembly file will also be made which includes the imported parts positioned relative to the assembly origin according to how the surfaces are defined in the IGES file.

- If the attempt fails, the IGES surfaces are grouped into one or more reference surfaces (named **Surface1, 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 **Surface** 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 3-16.

Exporting/Importing Parasolid Files

You can export SolidWorks part or assembly documents to Parasolid Text Files or Parasolid Binary Files.

To specify default export options:

1 Click **Tools, Options**, and select the **Export** tab.

2 In the **Parasolid** area of the dialog box, select the default method for exporting assemblies:

- **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 of assemblies, sub-assemblies, and parts, is maintained. The structure mirrors the original assembly, its nested subassemblies, and parts.

3 Under **Output as**, 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**.

To import a Parasolid file:

1 Click **File, Open.**

2 In the **Open** dialog box, set **Files of Type** to **Parasolid Files (*.x_t)** or **Parasolid Binary Files (*.x_b)**.

3 Browse to the desired file, and click **Open**.

The selected file is opened in a part window.

Exporting STL Files

You can export SolidWorks part and assembly documents to STL format.

To specify default export options:

1 Click **Tools, Options**, and select the **Export** tab.

2 Click the **STL Options** button.
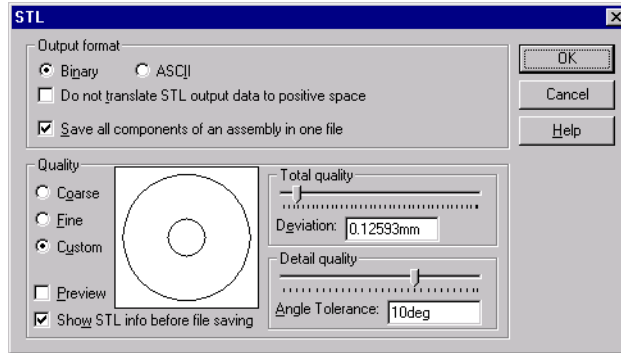
The STL dialog box provides the following options:

- **STL Output Format.** Choose **Binary** or **Ascii**.

- **Do not translate STL output data to positive space.** Check this box if you want exported parts to maintain their original position in global space, relative to the origin.

- **Save all components of an assembly in one file.** If you do <u>not</u> check this option, each component is written out relative to the assembly origin and saved in an individual file.

❑ **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. (See the **Preview** option below.)

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**. If you have selected **Custom** resolution (above), check this option to preview a faceted version of the model while you are adjusting the sliders. (Move the dialog box out of your way to view the faceted model.)

❑ **Show STL Info before saving.** Choose this option to see a faceted display of the model and an information box each time you save an STL file. The information box reports the file size, number of triangles, and the path and name of the file. Click **Yes** to save the file in the indicated location; click **No** to cancel the operation.

To export 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**.

Exporting/Importing ACIS Files

To specify default options for ACIS files:

    1 Click **Tools, Options**, and select the **Export** tab.

    2 In the **ACIS Output (.sat)** area of the dialog box, select the defaults for exporting ACIS files:

        • **Version.** Select the appropriate version for the target system.

        • From the **Units** pull-down list, select the default units of measure to use for ACIS files: **Millimeters**, **Centimeters**, **Meters**, **Inches**, or **Feet**.

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**.

        If the units of measure were not explicitly specified in the file, a pop-up dialog gives you the opportunity to specify the units now.

        The selected file is opened in a part window.

Exporting/Importing DXF/DWG Files

Only drawing documents can be saved to .DXF or .DWG files.

To specify default options for DXF/DWG files:

    1 Click **Tools, Options**, and select the **Export** tab.

    2 In the **DXF/DWG Output** area of the dialog box, select the defaults for exporting **DXF** or **DWG** files:

        • **Version.** Select the appropriate version for the target system.

        • **Fonts.** Select either **Truetype Fonts** or **AutoCAD STANDARD only**.

- **Custom map SolidWorks to DXF.** Click the checkbox to implement DXF mapping.

To export a 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 list **DXF files (*.dxf)** or **DWG files (*.dwg)**.

3 Browse to the desired file, and click **Open**.

   The **Open DXF/DWG File** dialog box appears.

   You have the option of importing to either a drawing or a part.

4 If you choose **Import to Drawing**, adjust the drawing sheet size and units of measure if necessary.

   - or -

   If you choose **Import to Part**, adjust the units of measure if necessary.

5 Click **OK**.

   The selected file is opened on a drawing sheet or in a part, as you specified.

   If the file is opened to a part, it forms 3-D curves.

Exporting/Importing VRML Files

VRML files can be used to display 3D graphics over the Internet.

To specify the default for VRML assembly files:

1   Click **Tools, Options**, and select the **Export** tab.

2   In the **VRML (*.wrl)** area of the dialog box, select the default for exporting VRML assembly files:

   • **Save all components of an assembly in one file.** If you do <u>not</u> check this option, a new directory is created that has the same name as the assembly, and each component is saved in an individual file in the new directory.

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**.

Note: When you use **Save As** to a VRML file (.wrl), the software takes the **View Section** option into account. If **View Section** is on, only the visible geometry is output to the .wrl file. Appropriately, this is different from the way the software behaves when you save to an IGES, STEP, Parasolid, or ACIS file.

To import a VRML file:

1   Click **File, Open.**

2   In the **Open** dialog box, set **Files of Type** to **VRML Files (*.wrl)**.

3   Browse to the desired file, and click **Open**.

   The selected file is opened in a part window.

Note: The import of VRML files attempts to knit the VRML polygons into a solid. There are no contoured surfaces in the imported file; all surfaces are planar. This import option is intended as a last resort for getting data into SolidWorks. In many cases (especially complex models) the data will not knit into a usable solid, and is useful for viewing purposes only.

Exporting/Importing a TIFF File

You can save any SolidWorks document as a TIFF image.

You can import a TIFF image and use it as a background of a part or assembly. You can capture the image as it appears on a normal view (screen) or you can capture the image as it appears on a print preview. For drawing documents you can select the sheet size and scale factor.

To specify the default TIFF image format for export:

1   Click **Tools, Options**, and select the **Export** tab.

2   Click the **TIFF options** button.

3   Select from the following options and then click **OK** to set the default format; click Cancel to close the window without saving the format.

4   Click **OK** to exit the **Export** dialog.

TIFF Options:

Image Type

- From the pull-down menu, select either **RGB Full Color**, or **Black & White** image.

Compression Scheme

- From the pull-down menu, select one of the following: **Uncompressed, Packbit**, or **Group 4 FAX**. (For Group 4 FAX compression, you must use the **Black & White** image type.)

Note: Uncompressed, Packbit and Fax Group 4 (for Black & White only) formats are supported for both import and export.
RGB, Palette, Black & White images are supported for import.
RGB and Black & White images are supported for export.

Capture Type

- Select either **Print capture** or **Screen capture**.

If you select **Print** capture, you can specify **DPI** (dots per inch), the **Paper size** and the **Scale** factor.

Object Linking and Embedding

You can use OLE to take advantage of features of other applications while in a SolidWorks document. You can also link or embed a SolidWorks part, assembly, or drawing document to another OLE-compliant application.

For example, you may need to perform advanced mathematical calculations to determine some dimensions on a part. Or you may want to include a SolidWorks part in a document such as a product data sheet.

OLE allows you to both bring data generated by other applications into the SolidWorks application and to place SolidWorks data into other applications.

This chapter introduces Object Linking and Embedding (OLE) and describes:

❑ Linking vs. embedding

❑ Using data from other applications in a SolidWorks document

❑ Bringing SolidWorks data into other applications

Linking vs. Embedding

When using OLE, you can *link* or *embed* files. Linking files allows you to continuously change the contents of a file in all places where it appears without having to edit each individual occurrence. Embedding a file allows you to keep the edits to the file specific to the place in which you embedded it.

Linking Files

When you link a file, the file remains in its original location. Anything you change in the original file affects all the files to which it is linked.

For example, if you edit a SolidWorks assembly document that is linked to several Microsoft Word documents, the changes you make are reflected in both the original SolidWorks file and all the Word documents. Also, when you double-click on a SolidWorks image in a Word document, the SolidWorks application launches allowing you to edit the original file (if you have SolidWorks installed on your PC).

The illustration below shows an example of a linked file.

Original SolidWorks file

File containing linked SolidWorks file

Editing the original file changes the linked file and editing the linked file changes the original file.
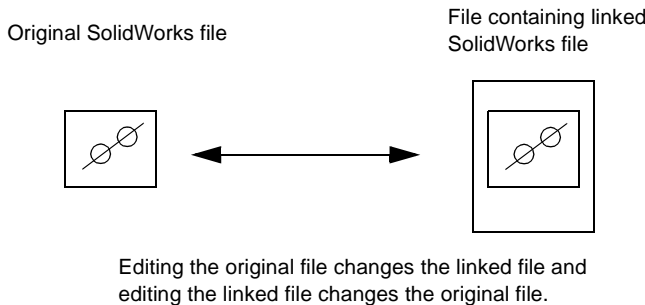
Linking is useful if you have data that may change and that you use in more than one place. By changing the original file, you automatically update the information in all the places to which it is linked.

Note: Keep track of the files to which you link data to avoid changing information in places you do not expect.

Embedding Files

If you embed one file in another, the original file becomes part of the file in which you embedded it. If you embed a SolidWorks document in a Word document, and if the SolidWorks application is loaded on the PC, double-clicking the image on the Word document page opens the SolidWorks application with the document active. You can edit the SolidWorks document and any changes you make affect only that document. Likewise, any changes you make to the original SolidWorks file do not affect the part embedded in the Word file.

Embedding is useful if you want discrete control over the data.

The illustration below shows an example of an embedded file.

Original SolidWorks file

File containing an embedded SolidWorks file. Changes were made to the embedded file only.

No link
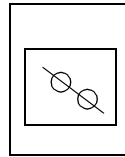
Editing the embedded file does not change the original file. Editing the original file does not change the embedded file.

Using Data from Other Applications in SolidWorks

With OLE, you can use other applications to generate data that you can then use in the SolidWorks application to control your parts, assemblies, and drawings. For example, you can calculate a dimension of a part using a math application. Or you can control the dimensions of a part using an Excel 97 spreadsheet. You can then link or embed the math application or Excel 97 spreadsheet to the desired dimension of the part in SolidWorks. Refer to **Design Table** on page 5-16 for details.

To use data from another application in a SolidWorks file:

1 With a SolidWorks file open, click **Insert**, **Object**.

2 To dynamically create and
   insert an object:

   • Click **Create New** in the
     Insert Object dialog box.
     The object appears in the
     SolidWorks file. Toolbars
     and menu options related to
     the object type, or
     application, that you
     selected are added to the
     SolidWorks window, allowing you to use the application's tools to edit the
     object you inserted.
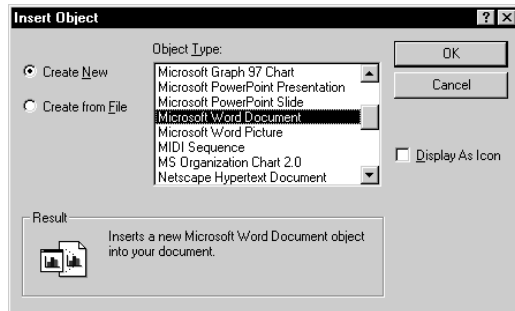
   To insert an existing object:

   • Click **Create from File** in
     the **Insert Object** dialog
     box. A file name appears
     in the **File** field.

   • Click **Browse** to select the
     desired object, and click
     OK.

3 To link the object to the
   SolidWorks file, click the
   Link check box in the **Insert
   Object** dialog. Otherwise the object is embedded.

4 To insert the object as an icon in the SolidWorks file, click **Display As Icon**.

5 Click **OK** to close the **Insert Object** dialog box.
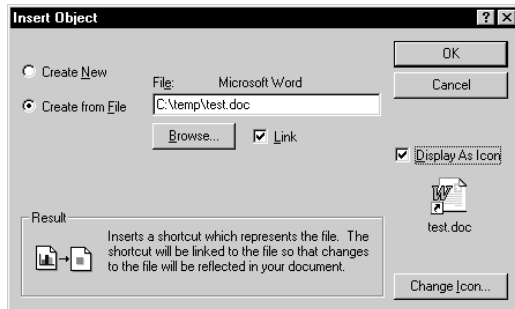
A Microsoft Word File in an Assembly

In this example, a Microsoft Word file, linked to the SolidWorks application, contains information about one of the components. Double-click on the Microsoft Word icon to view the contents of the file.
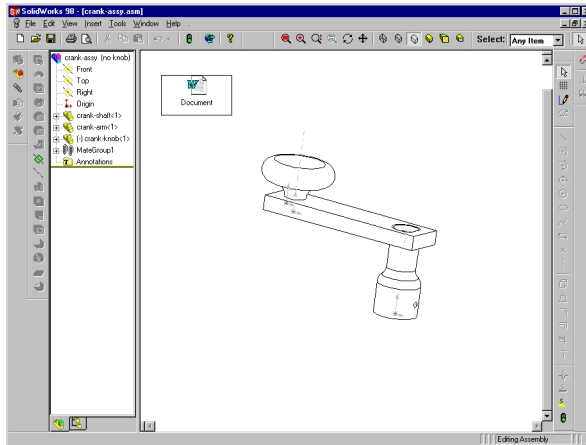
An OLE object in a SolidWorks Drawing

When you link an OLE object to a drawing document the default behavior is that the OLE object appears on every sheet of a multi-sheet document. If you want the OLE object to appear only on a single sheet, right-mouse click the icon and select **Show on this sheet only** from the menu.
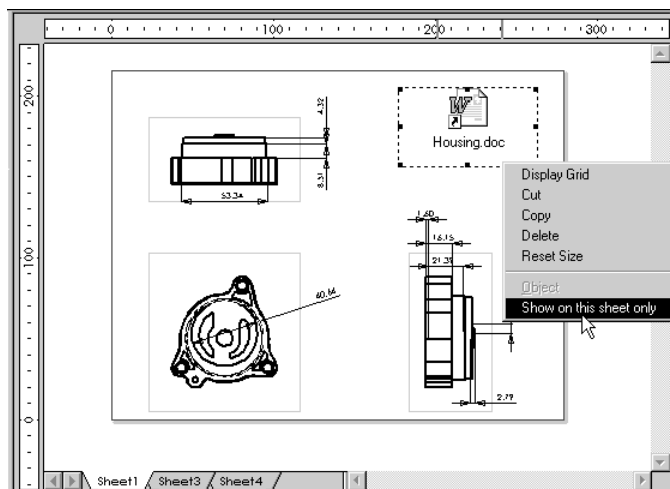
You can drag and drop selected OLE objects into SolidWorks documents. For example, if you drag highlighted text from a Microsoft Word file into an open SolidWorks document, the text becomes an embedded object.

Bringing SolidWorks Data into Other Applications

You may want to use a SolidWorks part, assembly, or drawing document in another application. Using OLE, you can link or embed the SolidWorks document with any other OLE-compliant application. For example, you can place a part in a product data sheet you created in Microsoft Word.

To use a SolidWorks file in another application:

1   In the desired application, click **Insert**, **Object**.

A dialog box appears allowing you to insert an existing object or to dynamically create and insert a new object. The dialog box that appears depends on the application you are using.

2   Select the desired options, including whether you want to link the SolidWorks file to this file and whether you want the SolidWorks file to appear as an icon in this file.

The SolidWorks file appears.

Example: An Assembly in a Word Document

This example shows a Word document containing a linked SolidWorks assembly document with the contents displayed.
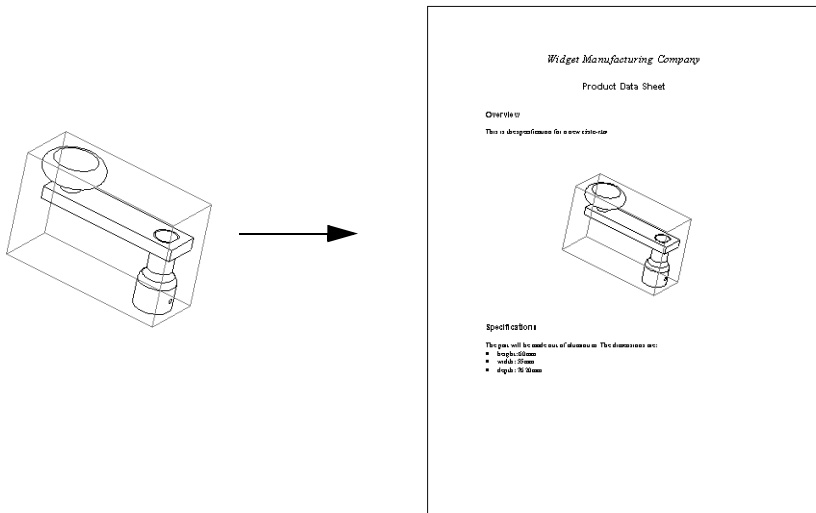
You can embed SolidWorks documents in other OLE containers by selecting the component's icon in the FeatureManager design tree and using the **Copy** and **Paste** functionality.

A

SolidWorks 98 Options

This appendix provides a list of all the options available for customizing part, assembly, and drawing documents in SolidWorks 98. To access these options, click **Tools**, **Options** and select one of the following tabbed pages:

- ❏ Color
- ❏ Crosshatch
- ❏ Detailing
- ❏ Drawings
- ❏ Edges
- ❏ Export
- ❏ External References

- ❏ General
- ❏ Grid/Units
- ❏ Import
- ❏ Line Font
- ❏ Material Properties (for parts only)
- ❏ Performance
- ❏ Reference Geometry

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.
- **Display with Broken Leaders** specifies how dimensions are displayed.
- **Dual Dimensions Display** specifies that two dimension types are used.
  Specifies the placement of the dual dimension: either **On Top** or **On Right** (on the same line).
- **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.

- **Alternate Section Display.** If you are using ANSI standard, you can use an alternate style of section display. With this style the section line does not display across the drawing view; the arrow ends stop at the ends of the section cut.

Alternate   Normal
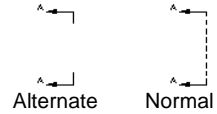
- **Centerline extension.** This value controls the centerline's extension length beyond the section geometry when it is in a drawing view. The default value is set according to standards, but you can select the value in the edit box and enter a new value. The new centerline length is used the next time the section is rebuilt.

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.

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.

Break Lines

- **Break Gap.** Lets you specify the size of the gap between break lines in a broken view in a drawing.

Datum Feature Symbols

- Specify the letter of the alphabet that will be used for the next Datum Feature Symbol. The default letter is "A." Successive labels are in alphabetic order.

Dimensions

- **Dim Font.** Lets you specify the font type 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.
- **Precision**. Sets the dimension precision and tolerance values for **Primary Units**, **Angular Units**, and **Alternate Units**.
- **Add Parentheses By Default.** Specifies that reference dimensions in drawings are displayed within parentheses.
- **Snap Text to Grid.** Specifies that the placement of dimension text snaps to the grid in a drawing or a sketch.
- **Use System Separator.** Specifies that the default system decimal separator is used in the display of decimal numbers. (To set the system default use **Control Panel, International (**or **Regional Settings), Number Format, Decimal Separator.**) To set a decimal separator different from the system default, click to deselect and enter the symbol that you want to use (usually the period or the comma).
- **Center Text.** Specifies that the dimension text is centered between its witness lines.

Virtual Sharp

- Represents a sketch point at the virtual intersection point of two entities. Dimensions and relations to the virtual intersection point are retained even if the actual intersection no longer exists, such as when a corner is removed by a fillet. Select one style of virtual sharp from the list.

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.

Annotations

- Specifies a filter to specify which annotations are displayed by default.
- Sets annotations properties.

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.

Automatic drawing view update when model changes. When checked, all views are updated automatically when the model changes. When unchecked, you update the views manually.

Show contents while dragging drawing view. When checked, the model is displayed as you drag a view. When unchecked, only the view boundary is shown while dragging. (Similar to **Show window contents while dragging** in Microsoft Windows.)

Detail View Scaling. Specifies the scaling for detail views. The scale is relative to the scale of the original drawing.

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.

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 as Phantom**–Tangent edges display using the Phantom style line font (a broken line).

- **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.

Export Options

Specifies the settings for exporting 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.

Assembly Output As

- Specify exporting assembly files to IGES as **A single file** or as multiple files, **Separate files for each part**.

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.

DXF/DWG Output

Select either **R12**, **R13**, or **R14**. This allows you to specify if a file saved with a **.dxf** or **.dwg** extension will be in R12, R13, or R14 format.

- **Fonts.** Select either **Truetype Fonts** or **AutoCAD STANDARD only**.

- **Custom map SolidWorks to DXF.** Click the checkbox to implement custom DXF mapping.

    When the **Custom map SolidWorks to DXF** option is checked, you can either create a new map file or select a file from the **Map File:** list. Use the **Browse** button to search for a map file.

ACIS (.sat)

Version

From the pull-down list, select the default version in which to save files with the **.sat** extension: either **Version 1.7**, **Version 2.0**, **Version 2.1, or Version 3.0**.

Units

From the pull-down list, select the default units of measure to use for ACIS files: **Millimeters**, **Centimeters**, **Meters**, **Inches**, or **Feet**.

VRML (.wrl)

Save all components of an assembly in one file. Remove the check to specify that all assembly components will be saved in individual files in a directory that has the same name as the assembly.

TIFF Options

Click the **TIFF Options** button to specify the following options:

Image Type

• From the pull-down menu, select either **RGB Full Color**, or **Black & White** image.

Compression Scheme

• From the pull-down menu, select one of the following: **Uncompressed**, **Packbit**, or **Group 4 FAX**. (For Group 4 FAX compression, you must use the **Black & White** image type.)

Select either Print capture or Screen capture.

If you select **Print capture**, you can specify **DPI** (dots per inch), **Paper size**, and **Scale** factor.

STL

Click the **STL Options** button to specify the following options:

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.

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.

Don't prompt to save read-only referenced documents (discard changes). Specifies that when this part is saved or closed, no attempt will be made to save its referenced documents that are opened for read-only access.

Allow multiple contexts for parts when editing in an assembly. You can create external references to a single part from more than one assembly context. Any individual feature or sketch within the assembly may only have one external reference, however.

Search document folder list for external references. Specifies that the document folder list is searched to 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

Automatically generate names for referenced geometry. (This option is off by default.)

When this option is off, you can mate to parts for which you have read/only access because you are using the internal face IDs of the parts. Unless you will use component replacement, leave this option off, especially in a multi-user environment.

When this option is on, you automatically create surface identifiers (for example: Face1, Face2) at the time you mate the part, therefore you need write access to the part, in most cases. Turn this option on if you intend to do component replacement using the same surface identifiers, remembering that you need write access to the parts you are using. (Rename the corresponding edges and/or faces on the replacement component to match the edge/face names on the original part.)

Update component names when documents are replaced. 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 = Under defined, Red = Over defined, Green = Selected. (Over defined and dangling points are always displayed, regardless of the **Options** setting.)

- **Infer from model.** When sketching on the face of an extruded part, inferencing lines and the inferencing cursor relate to the lines of the part.

- **Prompt to close sketch.** With this option selected, if you make a sketch with an open profile that can be closed with the model edges to extrude a boss, the system displays a dialog, **Close Sketch with Model Edges?** You can choose the model edges to close the sketch profile and the direction. The **Extrude Feature** dialog then appears. Otherwise, only the **Extrude Feature** dialog is available.

- **Create sketch on new part.** When you select **New, Part**, the part window opens with the sketching area and sketch tools immediately available.

- **Enable silhouettes.** With this option, silhouettes are selectable and you can use the following sketch tools on these edges: **Convert Entities**, **Offset Entities**, **Add Relation**, and **Dimension**.

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**.)

- **Display plane when shaded.** When you are editing a sketch in **Shaded** mode, this option displays the translucent plane. (By default, old models will have this checked.)

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

Over defining Dimensions

- **Prompt to set driven state.** When checked, specifies that when you add an over defining dimension to a sketch, a dialog box asks you if the dimension should be *driven*. (The default is to ask.)

- **Set driven by default.** When checked, specifies that when you add an over defining dimension to a sketch, the dimension is set to be *driven* by default.

Note: The above two checkboxes can be used together or alone, resulting in one of four different behaviors when you add an over defining 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.*"

- **Save backup copies in directory:** Lets you name a directory to which all backup copies will be stored by default. Use the **Browse** button to find the directory to use.

- **Maximize document on open.** When checked, each document opens to its largest size within the SolidWorks window.

- **Save AutoRecover info every (*n*) changes.** The software can automatically save information about your active part, assembly, or drawing documents so you will not lose your work in case of a system crash. Use the scroll arrows to specify the number of changes that should occur before information is automatically saved.

   Note: In a part or assembly document, a *change* is a rebuild or an action that requires a rebuild, such as the addition of a feature. In a drawing document a *change* is any action such as changing a dimension, creating a section view, or adding an annotation.

   To disable **AutoRecover**, click to remove the check from the checkbox.

- **Use English language.** If you selected the use of a language other than English during the SolidWorks installation, you can change to English by selecting this check box. Note that you must exit and re-start SolidWorks for this change to take place.

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.)
- If you choose **Fractions** and **Inches** or **Feet & Inches**, you can select **Round to Nearest Fraction**.

Angular Unit

- Select either: **Degrees**, **Deg/Min**, **Deg/Min/Sec**, or **Radians**.

- If you choose **Degrees** or **Radians**, specify the number of decimal places.

Spin Box Increments

- **Length.** The number of units added/subtracted when you click on a spin box arrow to change a linear dimension value.
- **Angle.** The number of degrees added/subtracted when you click on a spin box arrow to change an angular dimension value.

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.

Import Options

Specifies the default response to make when there is an error in importing IGES files.

Disposition: what to do when problems arise...

From the pull-down lists next to the error types, select the way that you want SolidWorks to resolve problems.

- The solutions for **Surface Errors**, **Surface Flaws**, and **Significant Digits** are:
  Ask me what to do
  Try to knit into solid
  Try to form reference surfaces
  Cancel operation
- The solutions for **Blanked Surfaces** are:
  Ask me what to do
  Ignore blanked surfaces
  Do NOT ignore blanked surfaces
  Cancel operation

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.

Reference Geometry

Display Filter

Lets you set a filter for the default display of reference geometry items in all new documents. To display a reference geometry item, click the item's checkbox to add a check.

The display filter manages the default display of the following: **Planes**, **Axes**, **Temporary Axes**, and **Origins**.

> **Note:** Use the options under the View menu to set the visibility of reference geometry in the *current* document only.

Plane Default Names

Lets you specify default plane names for parts and assemblies. For example, you may want to name planes Front, Top, and Right, instead of Plane1, Plane2, and Plane3.

Enter new names in the boxes that correspond to the original plane names.

SolidWorks Application Programming Interface

This appendix contains an overview of the SolidWorks Application Programming Interface (API). The SolidWorks API is an OLE programming interface to SolidWorks. The API contains hundreds of functions that can be called from Visual Basic, VBA (Excel, Access, etc.), C, C++, or SolidWorks macro files. These functions give you direct access to SolidWorks functionality; for example, creating a line, extruding a boss, or verifying the parameters of a surface. Complete online documentation is supplied with the SolidWorks API.

Online Documentation:

The online documentation supplied on your SolidWorks CD documents every object available in the SolidWorks API. Use it as you would use any conventional Windows online help, accessing topics through the table of contents, the index, or by using the Find capability to search for key words or phrases. The complete SolidWorks API documentation is located in the **...\Samples\Appcomm** subdirectory of your SolidWorks installation. Double-click on **API_help.hlp**. The SolidWorks API documentation can also be found on the SolidWorks web page:

http://www.solidworks.com/html/Products/api

Topics in this Appendix:

❑ Recording and playing SolidWorks macros

❑ Getting started and installing with C++ and Visual Basic

❑ Compiling and running your application with C++ and Visual Basic

❑ Programmer's guide and overview of the OLE automation interface

❑ Programming topics

❑ SolidWorks API objects

Recording and Playing SolidWorks Macros

You can record operations performed with the SolidWorks user interface and replay them using SolidWorks Macros. These macros are actually the equivalent calls to the API functions that were made when the operations were performed with the user interface.

To add the Macro toolbar:

Click **View, Toolbars, Macro**.

To record a SolidWorks macro

1 Click ▮● or **Tools, Macro, Record**.

2 Perform the operations you wish to have in your macro.

3 Click **Tools, Macro, Stop** and enter a filename for the macro. (Macros have a **.swb** extension.)

To run a SolidWorks macro:

1 Click ▶ or **Tools, Macro, Run**.

2 Choose the filename of the macro you wish to run.

To edit a SolidWorks macro

1 Click ▦ or **Tools, Macro, Edit**.

2 Choose the filename of the macro you wish to edit.

3 You can now modify, run, and debug your macro.

Note: SolidWorks Macros recorded in this way usually have some context-dependent elements that require minor changes before they can be used with other documents. For example, a macro recorded with Part1 may need to be changed so that it plays back with another document.

Installing and Getting Started with C++

Installing SolidWorks to compile and run your C++ applications requires the following:

MSDEV

MSDEV, Microsoft Developer Studio, must be installed with all appropriate libraries. It is recommended that you perform a Full Install to avoid unforeseen problems. However, if you wish to customize your setup, refer to the requirements below. Your installation should be for Visual C++ revision 5.0.

❑ **Windows NT** must have all UNICODE libraries installed

❑ **Windows 95** must have MBCS libraries installed (these are the default libraries for MSDEV)

❑ **Alpha** must have all UNICODE libraries installed

SolidWorks recommends that all operating systems also have the Shared MFC and MSVCRT Libraries installed.

SolidWorks

Development using the SolidWorks API is done with a regular installation of SolidWorks.

Note: The /API installation option is no longer available when you install SolidWorks because the debug libraries are no longer supplied on the SolidWorks CD. If you wish to DEBUG your code in a dynamic link library, then you can still build the debug information in your DLL file. (See the section Compiling Your C++ Applications.)

Compiling Your C++ Applications

Visual C++ 5.0 should be used to develop your C++ applications. SolidWorks allows you to create and run your project as a standalone **.exe** file or as a User DLL or Extension DLL.

1 Use a **MAK** file from a project in the *Samples* subdirectory to determine the build properties or simply load an existing project (**.mdp** file) and begin cutting and pasting your own code.

2 If you chose not to use the setups from one of the existing projects, you need to bring in the SolidWorks API declarations by yourself. You can generate these from the **\SldWorks\solidworks.tlb** type library, or you can simply include **\SldWorks\Samples\AppComm\swdisp.cpp** and **swdisp.h** into your project. The **swdisp.cpp** and **swdisp.h** files expose all objects available in SolidWorks. If you use a COM interface instead of Dispatch, include **amapp.h** instead of **swdisp.h** and **swdisp.cpp** .   The **amapp.h** file also includes the header information needed for anyone using event notification.

3 Choose the correct build configuration (Win32 Release, Win32 Debug, Win32 MBCS Release, or Win32 MBCS Debug):

- Windows 95 should use the MBCS configurations (Win32 MBCS Release or Win32 MBCS Debug.):

- Windows NT should use the Unicode configurations (Win32 Release or Win32 Debug.)

4 If you choose a Debug configuration you *must* also change project settings for that configuration as follows:

- Click **Project, Settings** to display the settings dialog.

- Click the **C/C++** tab and select **Code Generation** from the **Category** list. Select **Multithreaded DLL** from the Use run-time library (the previous setting should be **Debug Multithreaded DLL**).

- Click **Preprocessor** from the **Category** list and change _DEBUG in the **Preprocessor** definitions text box to _NDEBUG.

- If you wish to use ASSERTions or TRACE, then select **Preprocessor** from the **Category** list and at the end of the **Preprocessor** definitions text add _PSEUDO_DEBUG. You need to add two Microsoft files to your project, debug.cpp and debug.h, which can be downloaded from the SolidWorks web site at:

   http://www.solidworks.com/html/Products/api/solpartner.htm.

- Click the **Link** tab and select **General** from the **Category** list. Make sure the libraries in the Object/library modules text box are release versions and not debug. As an example, use mfc42u.lib and not mfc42ud.lib.

5 Add your own code.

6 Build your project.

Note: SolidWorks uses the function `InitUserDll3` to initialize your DLL. Please refer to the sample projects in the **../Samples** directory for implementation guidelines. If your DLL is not initialized properly or you do not use `InitUserDll3`, then you will receive *invalid add-in* and *incompatible version* messages.

Running Add-In Applications

Running your application as a Release DLL

Loading 3rd party applications is handled by the SolidWorks Add-In manager, which is accessed from the Tools menu in SolidWorks. To add an application to the Add-Ins available to the user you have to register the application in the windows registry (this is described in the section *Using the Add-In Manager* in the API Help documentation.)

Once the application is registered, follow these steps to load (or unload) it:

1   Start a SolidWorks session.

2   Open any document or create a new document.

3   Select **File, Tools, Add-Ins...** .

4   Check (uncheck) the checkbox for the Application to be loaded (unloaded).

5   Select **OK**.

Once the Add-In is loaded then it will be automatically loaded whenever SolidWorks starts up.

Note: As of SolidWorks 98, the \APPS directory is no longer supported for automatic loading of applications.

Debugging Add-In Applications

If you are debugging your application, then you should start SolidWorks from your development environment. This allows you to set break points in your application and step through your code. If your application was built as a Debug DLL (see above), follow these steps:

1   From your development environment, select **GO**.

2   When you are prompted for the executable name, enter *SldWorks.exe* with its path name.

   If your application is registered in the windows registry, make sure it is loaded:

3   Select **File, Tools, Add-Ins...** .

4   Check the checkbox for the application to be loaded.

If your application is not registered you can still debug it once SolidWorks is running.

1   Click **File, Open** and change your file selection filter to **Add-Ins(*.dll)**.

2   Select the desired DLL file and click **OK**. This brings your application into SolidWorks.

Note: Only debug information for your DLL will be available because the release MFC libraries are used, so you will not be able to step into MFC functions or see stack information outside of your DLL.

Installing and Getting Started with Visual Basic

Installing SolidWorks to compile and run your Visual Basic applications requires no special steps. Simply install SolidWorks from the SolidWorks CD.

Compiling Your Visual Basic Applications

SolidWorks supports programs through any OLE controller (Visual Basic, Visual Basic for Applications, etc.).

Applications written in Visual Basic can be started from many different points. In any of the cases below, the code generated by you should be similar to code generated by the SolidWorks Macro utility. To get started, it may be helpful to generate a macro from within SolidWorks (**Tools, Macro, Record**) and then use that code as the foundation for your application.

❑   Programs to Run From SolidWorks

There is no need to compile your application. If your routine uses only SolidWorks API calls, create your program in Visual Basic and use a file extension of **.swb** instead of **.bas**.

Note: SolidWorks Macro files (*.swb) only recognize Visual Basic 3.0 commands. If you wish to use Basic commands in Visual Basic 4.0 or higher, then you must compile and run the program as a Visual Basic executable or from a VBA application.

❑ Programs to Run as Separate **.exe** File

Build your project with a standard utility such as Microsoft Visual Basic. From the Microsoft Visual Basic application, select:

File, Make EXE File…

❑ Programs to Run as a DLL

SolidWorks does not support Visual Basic DLL implementations.

❑ Programs to Run From Other Applications

1 Load the application (Access, Excel, etc.).

2 Use the embedded VBA to generate your utility or script.

3 Use the compile utilities within the application to build your project.

Running Your Visual Basic Applications

Applications written in Basic can be started from many different points.

❑ Running From SolidWorks

1 Click **Tools, Macro, Run**.

2 Choose the desired BASIC source file and click **OK**.

❑ Running From Customized SolidWorks Menu Buttons

Applications written as .swb files can be run from custom buttons in the SolidWorks user interface.

1 Place the .swb file in the Macros folder of the SolidWorks installation. (You may have to create it.)

2 Customize the user interface by clicking **Tools, Customize...** .

3 Click the **Menus** tab.

4 Click **Macros** under **Categories** and the Macro you added under **Commands**.

5 Choose the menu, position, and text for the new button, then click **Add**.

6 To run the macro, click the button you just added.

Note: Once added, the button will always appear in subsequent SolidWorks sessions.

❑ Running as a Separate **.exe** File

Execute your **.exe** file.

If a SolidWorks session is already running, then your program will attach to it. If not, then a new SolidWorks session will be started.

❑ Running From Other Applications

Load the application (Access, Excel, etc.) and run your program or script from the application.

If a SolidWorks session is already running, then your program will attach to it. If not, then a new SolidWorks session will be started.

Programmer's Guide

OLE Automation Interface

The functions listed in the following pages are specified in C++ format. They may be called from Visual BASIC, VBA (Excel), C or C++.

- C++ classes are generated in a header file and an implementation file by importing the type library for SolidWorks using the Visual C++ class wizard.

- In C implementations, you need to determine the program ID from the type library and set up an invoke handler.

- Calling from Visual Basic or other varieties of BASIC which support OLE Automation requires only that an object is defined (Set Object =) and the implementation of the Basic OLE implementation takes care of the rest.

The SolidWorks API interface uses an object oriented approach. All of the functions described in this document are methods or properties which apply to an object. Thus, there is an assumption that these methods will apply to the current state. For example, there is a method EditDelete with effectively no arguments. Instead of passing in the items to delete, the EditDelete method acts on the current set of selected items. This style of interface may be different from what you have encountered in previous products, however, it is consistent with the Windows-based approach of SolidWorks. By way of reference, all of the SolidWorks user interface is based on Microsoft Foundation Classes.

COM vs. Dispatch

SolidWorks exposes functionality through OLE automation using IDispatch and also through standard COM objects.

❑ The Dispatch interface packages arguments and return values as Variants so they can be handled by languages such as Basic.

❑ A COM implementation gives your application more direct access to the underlying objects, and subsequently, increased performance.

COM implementations also have an HRESULT return value for each API function to indicate successful or unsuccessful calls and provide slightly more functionality with operations such as enumeration.

Sample Projects

SolidWorks provides you with several sample applications to get started. These Visual Basic and C++ projects can be found in the **Samples** subdirectory of your SolidWorks installation. Each project directory includes a **readme.txt** file so you can become familiar with each project and the highlighted functionality within that project. Refer to **...\Samples\comuserdll**, **...\Samples\userdll**, **...\Samples\TestApp**, and **...\Samples\VisualBasic**.

In most cases it is necessary to recompile each project on your system before attempting to run. For instructions on compile settings and requirements, see Compiling Your C++ Applications on page B-3 and Compiling Your Visual Basic Applications on page B-6.

For C++ applications you have the option of using either a COM or a Dispatch interface. A sample project exists for each interface type in the **...\Samples\ComUserdll** and **...\Samples\Userdll** directories respectively. The **...\Samples\TestApp** project is a second Dispatch example. Many developers simply load the **.MDP** file from the appropriate directory and cut and paste their code into the sample project. In the least, you should refer to the **.MAK** file for appropriate build settings.

Note: The sample projects are supplied on an as is basis, and are intended to demonstrate the method of using the OLE capabilities of SolidWorks. SolidWorks Corp. makes no representations or warranties regarding these samples.

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-12). The following C++ example shows you one method of checking for an empty VARIANT:

```
VARIANT v = m_ModelDoc.GetMassProperties();        // Get the Mass Properties
if (v.vt == VT_EMPTY)  ||  (V_VT(&v) == VT_NULL))// Error occurred
return;
```

SolidWorks API, Programming Topics

Implementation Guidelines

As a general rule of OLE programming, the caller is responsible for allocating and deallocating memory. This includes data returned by a SolidWorks function.

Interface Pointers

Interface pointers can also be an area of concern with C++ programming. Each SolidWorks API method which returns an interface pointer will automatically increment the Reference Count on the interface pointer by 1.

❑ For COM implementations, you may call a SolidWorks API which returns an interface pointer. You can then use this pointer as you wish, but you are responsible for releasing it.

This C++ COM example demonstrates how to handle interface pointers:

```
{ LPMODELDOC m_ModelDoc = NULL;     // Retrieve IModelDoc pointer
HRESULT res = UserApp->getSWApp()->get_IActiveDoc( &m_ModelDoc );
if( m_ModelDoc == NULL )
     return;
LPPARTDOC m_PartDoc = NULL;     // Retrieve IPartDoc pointer
res = m_ModelDoc->QueryInterface(IID_IPartDoc, (LPVOID *)&m_PartDoc);
ASSERT( res == S_OK );
...                             // Use the interface pointers within your code
m_ModelDoc->Release();          // Release the IModelDoc pointer
m_PartDoc->Release();           // Release the IPartDoc pointer
     }
```

❑ For Dispatch implementations, the release of the interface pointer is hidden in the destructor of the dispatch objects (`IModelDoc`, `IFace`, etc.). This implies that attaching an interface pointer to more than one of these dispatch objects, would cause a release to be performed by each of the objects as they go out of scope. This would cause a problem since the Reference Count is only incremented once when the interface pointer is returned to you. To avoid this problem, you must manually increment the Reference Count (`pdisp->AddRef();`) if you are attaching the interface pointer to more than one object.

This C++ Dispatch example demonstrates how to handle the Reference Count on interface pointers:

```
{LPDISPATCH modDisp;
modDisp = UserApp->getSWApp()->GetActiveDoc();
                          //  Get interface pointer to the active document
                          //  Ref Count on modDisp automatically incremented by 1
if( modDisp == NULL )
     return;
IModelDoc m_ModelDoc( modDisp );        //  Attach to the IModelDoc object
IPartDoc m_PartDoc( modDisp );          //  Attach to the IPartDoc object
modDisp->AddRef();          //  Manually increment the Ref Count on modDisp
                            //  because we use modDisp a second time
...                         //  Use objects within your code
     }                      //  Variables go out of scope and destructor called for
                          //  IModelDoc and IPartDoc which will decrement
                            //  the Ref Count on modDisp by two.
```

SafeArrays

C++ programmers who use API functions with SafeArrays, must be careful when managing SafeArray memory. If you receive a SafeArray from SolidWorks, then you are responsible for destroying it. Also, if you are passing a SafeArray to SolidWorks, the SafeArray will not be destroyed by SolidWorks and you are again responsible for destroying it.

Return Values

All C++ COM implementations will have an HRESULT return value. The API Help documentation will only show the COM syntax when necessary (see **Programmer's Guide** on page B-8 for more details). If you are using COM, it is implied that the SolidWorks API function will return an HRESULT and that any additional return values should be passed by reference as arguments.

HRESULT return values are used by SolidWorks to indicate that the code was called successfully. It is not meant as an indication that your call achieved its objective. For example, calling IGetFirstFace will have an HRESULT return value of S_OK if the code for IGetFirstFace was called successfully. It does not mean that the code succeeded in finding the first face. In this case, you should check the LPFACE return value for a NULL condition.

If an API method returns an object, it is always a good idea to verify that the object returned is not NULL. This type of error checking is good programming practice and will avoid crashes in your code for unexpected conditions.

For example:

```
LPFACE m_Face = NULL;
HRESULT hres = S_ERROR;
hres = m_Body->IGetFirstFace( &m_Face );
if (hres != S_OK || m_Face == NULL)
{
AfxMessageBox( _T("Error in call to IGetFirstFace.") );
return;
}
```

This type of error checking is also a good habit for Visual Basic programmers. For example:

```
Dim FaceObj As Object
Set FaceObj = BodyObj.GetFirstFace
If FaceObj Is Nothing Then                 // See if we found a Face
Msg = "Error in call to GetFirstFace"   // Define message
Style = vbOKOnly                         // Show OK button only
Title = "Error !"                        // Define title
Call MsgBox(Msg, Style, Title)
Exit Sub                                 // If no face, then exit
End If
```

For more information about NULL `Variant` return values, see **Helpful Hints** on page B-10.

SolidWorks API Objects

Accessing Objects

To call any of the methods or properties that are found in the API, you first need to obtain the object. For example, to use the `Face::GetArea` method you need to have a Face object.

There are many different ways to access objects within the SolidWorks API. For example, to obtain the Face object mentioned above, you could:

- Obtain the Body object using `PartDoc::Body` and then traverse the faces on the body using the `Body::GetFirstFace` and `Face::GetNextFace` methods.

- Or you could get the Face object from the current set of selected items using `SelectionMgr::GetSelectedObject`.

- Or maybe you prefer to get the face object by its name using `PartDoc::GetEntityByName`.

In either case, once you have the Face object, you are then able to access the properties and methods found in the Face Class. For example, once you have the Face object, you could get the number of edges on the face using the `Face::GetEdgeCount` method, or you could get the normal vector for the face using the `Face::Normal` property.

Do not get confused with API methods and properties which also require a selected item, such as `AssemblyDoc::OpenCompFile`. To call `OpenCompFile` you need to have the `AssemblyDoc` object AND have a component selected. The component can be selected in one of two ways:

- The user can interactively select the component.

- Or you can programmatically select the component using `ModelDoc::SelectByID`.

For more information on object relationships, please refer to the API Object Diagram.

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.
BOM Table	Allows access to the BOM table information and values.
CoEdge	This "edge with a defined direction" allows access to the underlying edge and loop as well as various CoEdge data.
Configuration	This object gives you access to the current configuration and certain properties, along with access to the component interface for assembly traversal.
Component	This object is used for assembly traversal. Information returned from the component methods and properties is based on the current configuration and may vary if the user has activated a different configuration.
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	Obsolete.
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.
SelectionMgr	Allows you to get information about selected objects and to get your selection coordinates interpreted in model or sketch space.
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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