



SolidWorks 2001

What's New

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Introduction

About This Book

There are many new features available in the SolidWorks® 2001 software. This book was created to highlight the new functionality and to help you learn these new features. This book is for experienced users and assumes that you have a good working knowledge of an earlier release of the SolidWorks software.

If you are new to using the SolidWorks software, you should contact your reseller for information about SolidWorks training classes.

Using This Book

Use this book in conjunction with the part, drawing, and assembly files provided.

- 1 Install the SolidWorks 2001 software.

Be sure to select the option for installing the **Example Files**. The example files for this book are placed in the *installation directory***samples\what's new** folder.

TIP: To install the *SolidWorks 2001 Getting Started* in .pdf format, select the option for installing the **Manuals** as well. The manuals are placed in the *installation directory***lang\your language\manuals** directory.

- 2 Create a new folder on your system, and copy the sample files into the local folder.

NOTE: Since some of the sample files are used with more than one example, *do not* save changes to these files.

- 3 Follow the instructions.

Go through this book from beginning to end, opening the proper part, drawing, or assembly document for each example, or use the Table of Contents or Index to locate topics of special interest to you.

Late Changes

We have tried to make this book as complete and accurate as possible. Refer to the *Read This First* and *Release Notes* that are shipped with your SolidWorks software for information that could not be included in this printed book. Also, refer to the **Overview of New Functionality in SolidWorks 2001** in the *Online User's Guide*.

Moving to SolidWorks 2001

Backup Copies

We recommend that you save backup copies of all SolidWorks documents (parts, assemblies, and drawings) before opening them in SolidWorks 2001. These documents are automatically converted to SolidWorks 2001 format when opened. Once converted and saved, the documents are not accessible in earlier releases of the SolidWorks software.

Converting Older SolidWorks Files to SolidWorks 2001

Because of changes to the SolidWorks files with the development of SolidWorks 2001, opening a SolidWorks document from an earlier release may take more time than you are used to experiencing. However, once the file has been opened and saved, subsequent opening time returns to normal.

The SolidWorks Conversion Wizard provides a way for you to automatically convert all of your SolidWorks files from an earlier version to the SolidWorks 2001 format. Depending on how many files you have, the conversion process may take a while, but once it is done, the files will open more rapidly.

To access the Conversion Wizard, click the Microsoft **Start** button, select **Programs, SolidWorks 2001**, then **SolidWorks Utilities**. Click **Conversion Wizard**.

When the conversion utility begins, it offers you the choice of backing up all of your files before the conversion. If you choose to backup your SolidWorks files, the Conversion Wizard copies the files to a sub-folder named "Solidworks Conversion Backup." The wizard asks you for the location of the files to be converted, and leads you through the process.

At the end of the conversion process, two report files exist in the folder to which you directed the conversion.

- **Conversion Wizard Done.txt** contains a list of files that converted.
- **Conversion Wizard Failed.txt** contains a list of files that did not convert.

SolidWorks Service Packs

You can take advantage of SolidWorks service packs that are regularly posted on the SolidWorks Web site. These service packs contain software updates and enhancements to the SolidWorks 2001 software. To check for a new service pack, click **Help, Service Packs**, and click the **Check** button. Select the check box if you want the software to automatically check the SolidWorks Web site for a new service pack once a week.

SolidWorks 2001 Getting Started

The *SolidWorks 2001 Getting Started* book is on the SolidWorks CD-ROM in **.pdf** format. If you did not select the **Manuals** option when installing the software, you must reinstall the SolidWorks software to access the manuals.

To access the installed manuals, click **Help, Getting Started Manual** when running the SolidWorks 2001 application.

SolidWorks Fundamentals

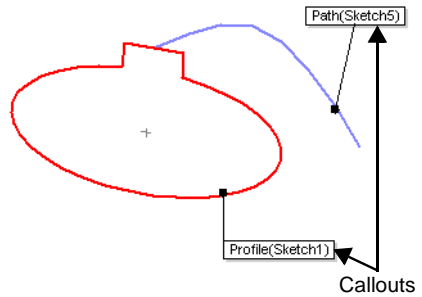
This chapter discusses some basic concepts used throughout the SolidWorks 2001 application. It provides an overview of the enhancements to the following topics:

- Callouts and Handles
- Documentation
- FeatureManager® Design Tree
- Graphics Performance
- Language Support
- Macros
- PropertyManager
- Selection
- Toolbars
- User Interface
- View Animation
- Web Folders

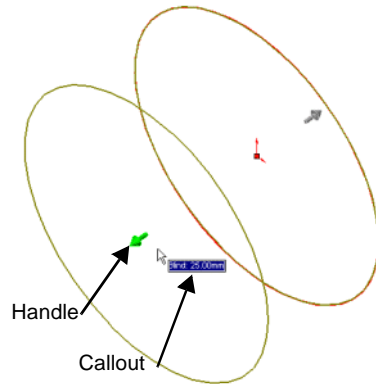
Callouts and Handles

Callouts

There are new callouts that help you easily distinguish between different entities. For example, these callouts show the sweep profile and the sweep path. You can drag these callouts, but you cannot use them to change any properties of the entity to which they refer.



Some callouts, such as those for extrusions, display figures that you can edit to manipulate the size of the entity. You edit the figures by typing in the PropertyManager; the callout and preview update showing the new figures.



Handles

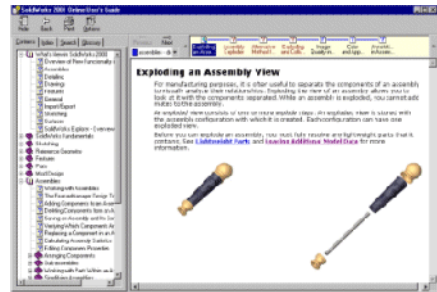
There are new handles that allow you to dynamically click, move, and set certain parameters without leaving the graphics area. Active handles are the highlight color, inactive handles are gray.

You can find more information on callouts and handles throughout the book in individual sections about the functions that use them.

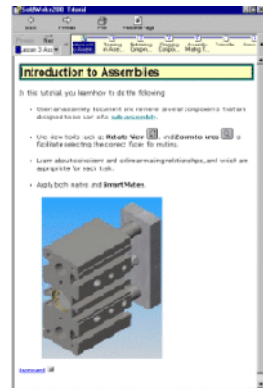
Documentation

There are several improvements and additions to the SolidWorks documentation.

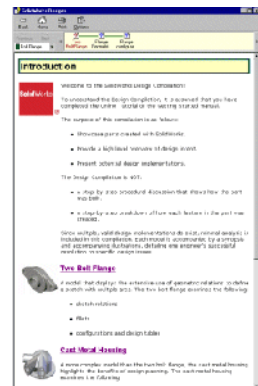
- ❑ **HTML Help** - The SolidWorks Online User's Guide, as well as the other help files shipped with SolidWorks, is now in HTML Help. This offers you improved search and display functions, as well as a consistent look and feel with Microsoft's online help.





- ❑ **Online Tutorial** - An Online Tutorial has been added that contains four chapters. This tutorial provides an online alternative to the Getting Started book, and helps new users build their first models faster.




- ❑ **Design Portfolio** - An online Design Portfolio has been added that contains three machined parts. The Design Portfolio helps you understand how you can combine the various SolidWorks tools to meet your design needs and requirements.





- ❑ **Glossary** - A glossary has been added to the SolidWorks Online User's Guide. It includes terms to assist new users, such as 2D users switching to 3D.
- ❑ **Context-sensitive help** - There is a new context-sensitive help button for getting help about FeatureManager design tree, ConfigurationManager, and toolbar items. Click **What's This**  on the Standard toolbar, then click an item to get help on that item. You must add this button to the Standard toolbar to be able to access **What's This**  help.

To add the button to the Standard toolbar

- 1 With a SolidWorks document open, click **Tools, Customize**.
The **Customize** dialog box opens.
- 2 On the **Commands** tab, click **Standard**, and drag the **What's This**  button to the Standard toolbar in the main SolidWorks window.
- 3 Click **OK** to close the dialog box.

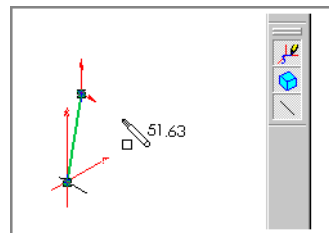
To use What's This Help:

- 1 Click **What's This** .
- 2 Click on a toolbar button, a FeatureManager design tree item, or a ConfigurationManager item.

The pointer turns to an arrow with a question mark .

The *SolidWorks 2001 Online User's Guide* appears containing help information about the item you chose.

- ❑ **Other Additions to Online Help**
 - **Index** - The index has been expanded and made more consistent.
 - **SolidWorks Explorer** - New task-oriented "How do I..." topics have been added to the SolidWorks Explorer online help. Some samples include "How do I revise an assembly document?" and "How do I replace one part with another?"
 - **Animations and real-world images** - There are approximately 15 new animations and over 50 rendered images that have been added to the online help.



FeatureManager Design Tree


There are two new items for the FeatureManager design tree:

- ❑ Flyout FeatureManager design tree
- ❑ FeatureManager design tree color schemes

Flyout FeatureManager Design Tree

There is now a *flyout FeatureManager design tree* function that allows you to view both the FeatureManager design tree and the PropertyManager at the same time. To activate the flyout FeatureManager design tree, click either the title in the PropertyManager, or click the FeatureManager design tree tab. Click anywhere else to hide the flyout FeatureManager design tree.

To see an example of the flyout FeatureManager design tree:

- 1 Open **Flyout_FeatureManager.sldprt**.
- 2 Click **Linear Pattern** .
- 3 Click either the **Linear Pattern** title in the **Linear Pattern** PropertyManager, or click the FeatureManager design tree tab.

The FeatureManager design tree flies out and appears next to the **Linear Pattern** PropertyManager.

With this new flyout FeatureManager design tree functionality, you can select a feature in the flyout FeatureManager design tree without having to leave the graphics area.

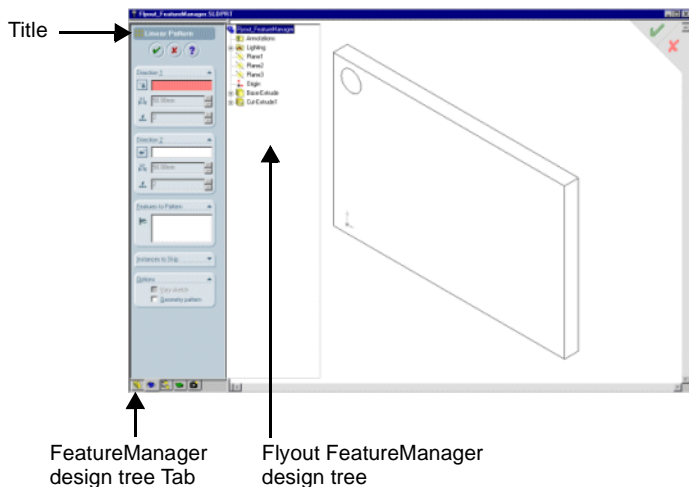
- 4 Select **Cut-Extrude1** as the **Feature to Pattern**.

Cut-Extrude1 appears in the **Features to Pattern**

box and the cut becomes highlighted in the part.

You do not have to build the whole pattern.

- 5 Click anywhere in the graphics area to hide the flyout FeatureManager design tree.



FeatureManager Design Tree Color Schemes

You can now apply different color schemes to the FeatureManager design tree.

To apply a color scheme to the FeatureManager design tree:

- 1 Open **Flyout_FeatureManager.sldprt**.
- 2 Click **Tools, Options**.
- 3 On the **System Options** tab, select **Colors**.
- 4 Click the arrow next to the **FeatureManager Color** box, select a color, then click **OK**.

The new color scheme is applied to the FeatureManager design tree.

Graphics Performance

There is a new performance option that enables software Open Graphics Language (OGL). This option disables graphics adapter hardware acceleration, and instead enables graphics rendering using only software. For many graphics cards, this results in slower performance. You should only enable this option if instructed by technical support.

To enable software OGL:

- 1 Click **Tools, Options**.
- 2 On the **Systems Options** tab, select **Performance**.
- 3 Select the **Use Software OpenGL** check box, then click **OK**.

Language Support

Solidworks software is now supported in the Korean language. This includes the user interface, all printed manuals, and all online documentation.

For add-ins such as FeatureWorks®, PhotoWorks®, and so forth, the user interface is supported. Online

help for these add-ins will be partially supported. Future SolidWorks service packs will feature more comprehensive Korean language support of add-ins' online help.



Macros

SolidWorks software now uses Visual Basic for Applications (VBA) instead of Summit Basic as the engine that records, runs, or edits macros. Recorded macros are now saved as **.swp** VBA project files.

You can read and edit **.swb** and **.swp** (VBA) files using the VBA editor. When you edit an existing **.swb** file, the file will automatically be converted into an **.swp** file. You can export a module to a file that you can use in other VB projects.

For more information on VBA, look to the Help menu in the VBA editor.

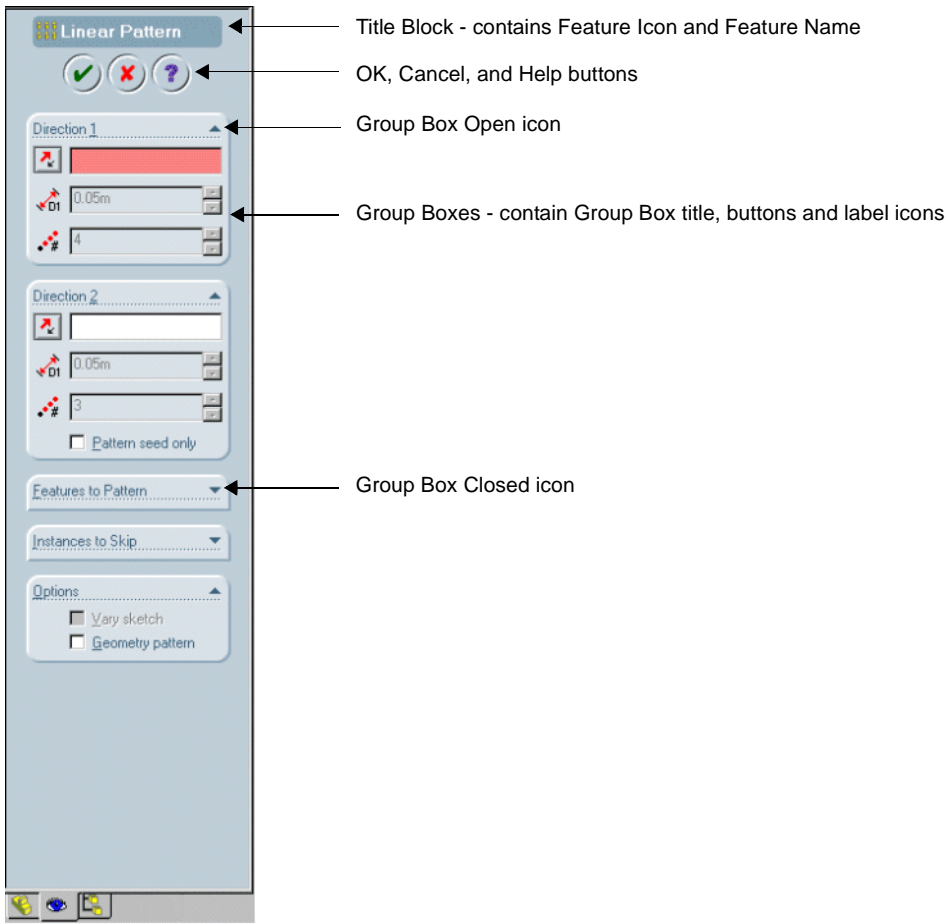
PropertyManager

The PropertyManager has a new design for SolidWorks 2001. More functions now use the PropertyManager instead of dialog boxes, allowing your graphics to be displayed instead of hidden by dialog boxes. The following commands have been moved to or updated in the PropertyManager:

-
- | | | |
|-------------------------------------|---------------------|---------------------------|
| • Balloon | • Face Curves | • Rib |
| • Base Flange | • Fillet | • Revolve |
| • Chamfer | • Flat Pattern | • Sketch Driven Pattern |
| • Circular Pattern | • Fold | • Sketch Offset |
| • Closed Corner | • Linear Pattern | • Sketch Properties (all) |
| • Curve Through
Reference Points | • Loft | • Sketched Bend |
| • Drawing Views (all) | • Mate | • Stacked Balloons |
| • Edge Flange | • Mate Reference | • Surface Extend |
| • Extrude | • Miter Flange | • Surface Trim |
| | • Move
Component | • Sweep |
| | | • Unfold |
-

Description

The new PropertyManager contains the following items:



Functionality

All of the functions found in the SolidWorks 2000 dialog boxes are available in the SolidWorks 2001 PropertyManager. For example, if you create a Linear Pattern, you now use the PropertyManager to set all of the options that you previously set using the dialog box, such as Direction, Spacing, Total Instances, Pattern Features, and so forth.

Color Schemes and Skins

You can now apply a variety of pre-defined color schemes to the PropertyManager. You can also save them as part of a SolidWorks color scheme. Additionally, you can now apply skins as background images to the PropertyManager. Skins are images that appear behind the PropertyManager data. See **Skins** on page 1-17 for details.

To apply a color scheme to the PropertyManager:

- 1 Click **Tools, Options**.
- 2 On the **System Options** tab, select **Colors**.
- 3 Click the arrow in the **PropertyManager Skin** box, and select **None**.

NOTE: Skins take precedence over a color scheme in the PropertyManager.

- 4 Click the arrow in the **Current Color Scheme** box, and select a color scheme from the menu.
- 5 Click **OK**.

When you display the PropertyManager, the new color scheme appears.


Entering Values and Accepting or Cancelling Commands

When you have the PropertyManager displayed, there are several new ways to enter values and accept commands. The following example provides details.

To see an example of entering values and accepting commands:

- 1 Open **PM_commands.sldprt**.
- 2 Right-click **Base-Extrude** in the FeatureManager design tree, and select **Edit Definition**.

The **Base-Extrude** PropertyManager appears.

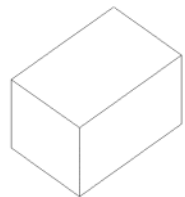
- 3 Under **Direction 1**, type 30mm as the new **Depth** , but do not press **Enter**.
The preview does not change because you have not yet accepted the new value.
- 4 Press **Enter**.

The preview now updates to show the new depth because you accepted the new value by pressing **Enter**.

At this point, if you want to cancel the **Base-Extrude** command and retain the original depth of 20mm, you press **Esc**. However, you want to accept the new value.

- 5 Press **Enter** again.

The **Base-Extrude** command is accepted. The part updates to show the new values, and the **Base-Extrude** PropertyManager closes.



Selection

Box Selection

You can now select all entity types in parts, assemblies, and drawings by dragging a selection box. You can select multiple entities by pressing the **Ctrl** key while dragging multiple box selections.

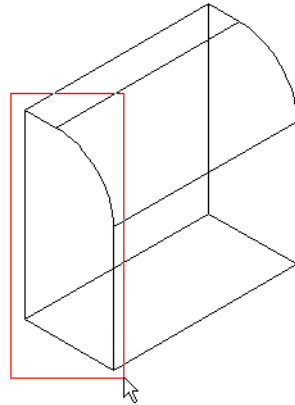
The default geometry type selected is as follows:

- Part documents - edges
- Assembly documents - components
- Drawing documents - sketch entities, dimensions, and annotations

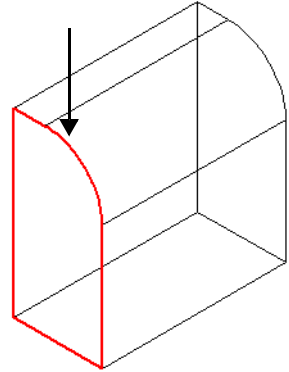
You can set up different selection types with the selection filters. In drawings, hidden edges and faces are not selected.

To use box selection to select multiple edges in a part:

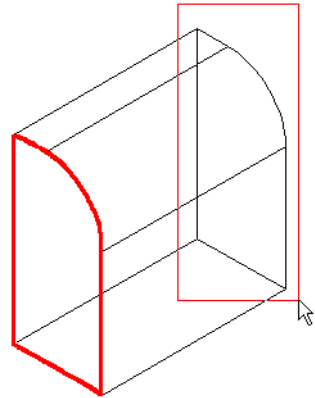
- 1 Open **Box_Selection.sldprt**.
- 2 Drag a selection box around the left face.



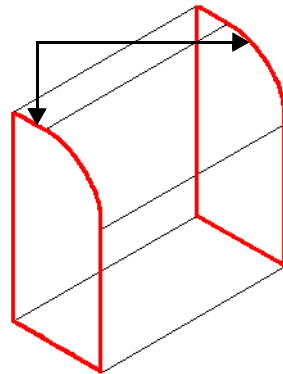
The edges of the left face are highlighted.



- 3 Press **Ctrl** and drag to select the right side of the part.



The left side edges remain selected. The right side edges are highlighted and added to the selection. Note that even the hidden edges are selected.



Loop Selection

You can now select a loop of connecting edges using the right mouse button, and then apply a feature such as fillet or chamfer to the selection.

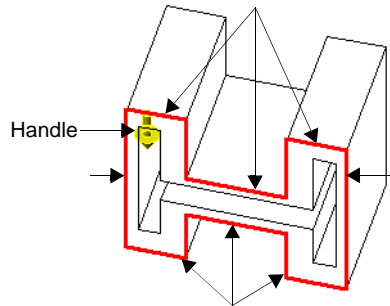
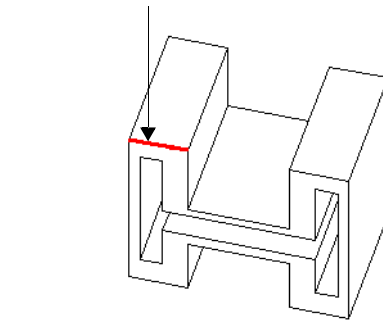
To do a loop selection:

- 1 Open **Select_loop.sldprt** and right-click the edge as shown.

The edge becomes highlighted and the menu appears.

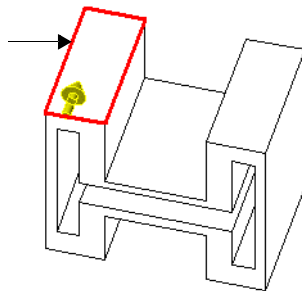
- 2 Click **Select Loop**.

The edges of the entire outer loop of the front face become highlighted. The handle indicates the face whose edge is being loop selected.



- 3 Click the handle to change the loop selection to the edges of the other connecting face.

The edges of the top face become highlighted. The handle changes to indicate the new face whose edge is being loop selected.



Tangent Selection

You can now select a group of tangent curves, edges, or faces, using the right mouse button. You can then apply a feature, such as fillet or a chamfer, to the selection.

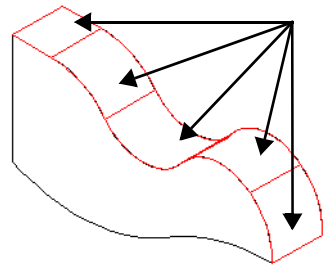
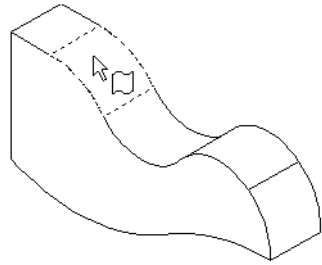
To do a tangent selection:

- 1 Open **Select_tangent.sldprt** and right-click the face as shown.

The face becomes highlighted and the shortcut menu appears.

- 2 Click **Select Tangency**.

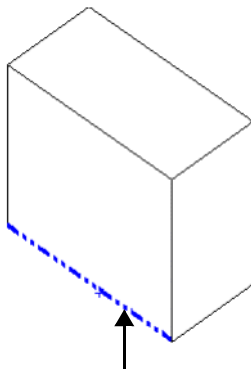
All five of the tangent faces become highlighted.



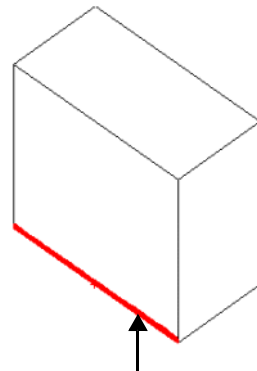
Highlighting

Selected Items

Items that you select are now highlighted using a solid style font instead of a dashed style font.








Old Highlighting - Dashed




New Highlighting - Solid

Edges that you select highlight as thick solid lines; edges of *faces* that you select highlight as thin solid lines.

Dynamic Highlighting

Dynamic highlighting has been improved. The edge and edge of face highlighting scheme works the same way as the selection highlighting when you have enabled **Fast HLR/HLG**  and use **Hidden Lines Removed** , **Hidden in Gray** , or **Wireframe**  view, or, at all times when you use **Shaded View** . When you dynamically move the pointer over an edge or a face, edges highlight as thick solid lines; edges of faces highlight as thin solid lines.

For all drawings, and for all other cases when **Fast HLR/HLG**  is *not* enabled, dynamic highlighting uses the old highlighting scheme.

Toolbars

Improved Docking

The movement and docking of toolbars has been improved. Previously, when you tried to place a toolbar, it sometimes would not dock where you placed it. Movement and docking behavior is now much more reliable.

User Interface

Dynamic and Shaded Previews

You now have two new previews available when working with features: dynamic preview and shaded preview.

Dynamic and shaded previews are supported for the following features:

- Extrude (base and cut)
- Revolve
- Sheet metal edge flange and base flange
- Simple hole
- Surface extend

Dynamic Previews

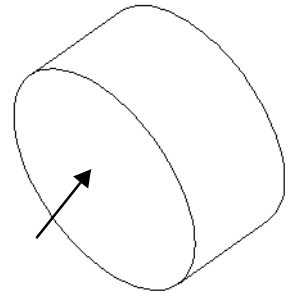
When you create or edit a feature that supports dynamic previews, and then move the pointer, you see a dynamic preview in the graphics area of how moving the pointer changes the model. The dynamic preview snaps in increments that are one-tenth of the spin box increment value set in the PropertyManager.


To disable the snap functionality, hold down **Shift** while moving the pointer. You can then move the pointer to any value.

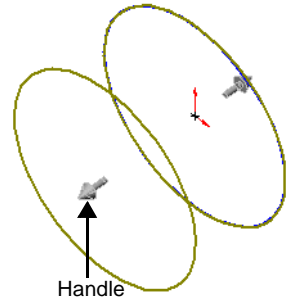
To see an example of the dynamic preview:

- 1 Open **Dynamic_preview.sldprt**.
- 2 Right-click the face as shown, and select **Edit Definition**.


The **Base-Extrude** PropertyManager appears.



- 3 Click the handle and move the pointer forward and back.
The face you are extruding moves dynamically in 1mm increments, one-tenth of the 10mm increment value set in the **Depth**  box on the PropertyManager. The callout also displays the changing values in 1mm increments.



- 4 Move the pointer until the value is 40mm, then click anywhere in the graphics area.

The 40mm value is locked in. The selected face no longer moves. The pointer changes to .



- 5 Right-click in the graphics area.

The 40mm value is accepted. The PropertyManager closes. The part updates to show the new value.

Shaded Previews

Shaded previews are a graphic aid to help you visualize features that you create.

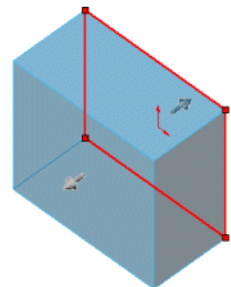
To see an example of shaded previews:

- 1 Open a new part document and sketch a rectangle.
- 2 Click **Extrude** , move the pointer to any **Depth**  value you want, then click to lock the value.

A shaded preview of the extrusion appears, similar to the example shown. You can rotate, pan, zoom, and set standard views while maintaining the shaded preview.

- 3 Click **OK** .

The value you entered is accepted. The part appears in the graphics area. The PropertyManager closes, and the **Base-Extrude** feature appears in the FeatureManager design tree.



Edit With Application Option

You can now right-click to edit OLE items with their related application, such as Visio or Microsoft Word and Excel. Select the OLE item, right-click, and select **Edit with <application name>** from the shortcut menu. The application opens in a separate window with the selected OLE item displayed.

Pop-up Tooltips

There are new pop-up tooltips that help guide you in building your models. The pop-up tooltips appear with an informational message, and then disappear after a few seconds. These tooltips replace the message boxes that required you to click **OK** to close them.

To see an example of pop-up tooltips:

- 1 Open **Popup_tooltips.sldprt**.

The folded sheet metal part appears.

- 2 Click **Insert, Features, Sheet Metal, Fold**.

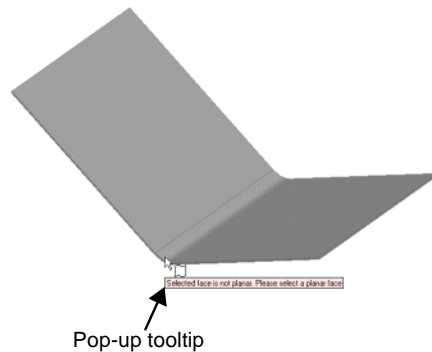
The **Fold** PropertyManager appears.

- 3 Under **Selections**, select **Fixed face**, then select **Bend1** in the part.

A pop-up tooltip appears, warning you that the selected face is not planar.

- 4 Under **Selections**, select **Bends to fold**, then select **Bend1** again in the part.

Another pop-up tooltip appears, warning you that the selected bend is already folded.



Skins

You can now apply skins as background images to the PropertyManager. Skins are images that appear behind the PropertyManager data, similar to this sample. You can also create your own PropertyManager buttons.

NOTE: Only bitmap (.bmp) files are supported as skins.

The skin bitmap files are stored in the **skins** folder, located at *installation directory\data*. SolidWorks offers several sample skins for your use. You can also create your own customized skins using any commercial graphics editing software, and then add your customized skins to the **skins** folder.

To create a background skin bitmap file:

- 1 Open your graphics editing software and create your bitmap file. You can use any color (including true colors), resolution, or size setting.
- 2 Save the skin as *installation directory\data\skins\<skin bitmap file name>.bmp*. For example, you can name the file **background.bmp**.

TIP: The image is tiled, so it is best to use tilable images when applying textures as the background.

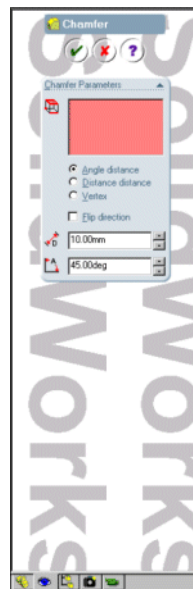
After you create your background skin bitmap file, you can customize any or all of the **OK**, **Cancel**, **Help**, and **Pin** buttons.

You are encouraged to customize all four buttons so they match. If you do not customize a button skin bitmap file, the default SolidWorks button is applied. The button colors are blended to the background color. If you have dark background colors, the default buttons may be difficult to read.

To customize the PropertyManager buttons:

- 1 Create the background skin bitmap file.
- 2 Create the button skin bitmap file with your graphics editing software, using the following guidelines:
 - **File Name.** The button skin bitmap file name must be *<skin bitmap file name>__<button name>.bmp*, for example, **background__ok.bmp**.

NOTE: Note that “__” is a double underscore. Also, the *button name* can be **ok**, **cancel**, **help**, or **pin**.



- **Display States.** You must display the button in four states: off, on, mouse over, and unavailable. Here is an example using the **Cancel** button:



Off On Mouse over Unavailable

- **Pixel Size.** Each individual button state must be 32 by 32 pixels; each group of the four button states must therefore be 32 pixels high and 128 pixels wide



Individual button
32 by 32 pixels



Group of buttons
32 by 128 pixels

- **Colors.** The buttons can have a color palette of 2, 8, 16, or 256 colors. True colors are not supported.
- **Transparency.** To set any part of the button skin bitmap file to transparent, set the RGB color for that area to 255, 0, 255. Transparent areas display the color and design used in the background skin bitmap file. Here is an example using the **OK** button:

Transparent area using
RGB setting 255, 0, 255



Areas displaying
user-defined colors

3 Add the button skin bitmap files to *installation directory\data\skins*.

The button skin bitmap files are automatically integrated into the background skin when you enable the skin.

To enable and view skins:

- 1 Click **Tools, Options**.
- 2 On the **System Options** tab, click **Colors**.
- 3 Click the arrow in the **PropertyManager Skin** box, and select a skin from the menu.

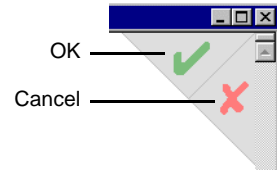
NOTE: To disable skins, select **None**.

- 4 Click **OK** to enable the skin.
- 5 Open a file, and when you activate the PropertyManager, the skin appears as the background image in the PropertyManager.

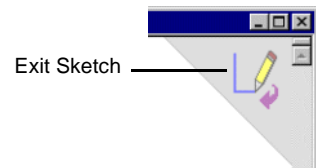
Streamlined Acceptance of Features

You now have several streamlined ways to accept features you create. After creating a preview of a feature, you can do the following:


- ❑ Right-click and select **OK** or **Cancel** from the shortcut menu to accept or cancel the preview
- ❑ Click the **OK** or **Cancel** icons that appear in the Confirmation Corner of the SolidWorks graphics area



- ❑ When you are in sketch mode, click the **Exit Sketch** icon in the Confirmation Corner to finish the sketch.




To turn off the Confirmation Corner, click **Tools**, **Options**. On the **System Options** tab, click to clear the **Enable Confirmation Corner** check box.

- ❑ Right-click when the pointer changes to  to accept the preview, or click to return to the preview without accepting the values

Welcome to SolidWorks Screen

There is a new Welcome screen that appears when you open the SolidWorks application. From the Welcome screen, you can:

- Create documents using the Online Tutorial.
- Read the Getting Started manual.
- View the Design Portfolio for modeling tips.
- Create a new SolidWorks document.
- Open an existing SolidWorks document.
- Click **Cancel**  to close the Welcome screen.



To always show the Welcome screen when you open SolidWorks, select the **Show at startup** check box.

View Animation


You now have the option to animate how changes in view orientation are displayed. With view animation enabled, when you change from one view to another, the change occurs showing an animated sequence of frames between the first view and the new view.

View animation occurs when you change views using the following:


- View orientation commands (For example, **Front**, **Isometric**, and so forth)
- **Zoom to area**, **Zoom to fit**, **Zoom to selection**
- **Shift+** arrows to rotate 90°
- Any command where the system changes the view orientation automatically, such as extruding the first sketch of a part

NOTE: View animation works for part and assembly documents only. It does not work for drawing documents.

To set and use view animation:

- 1 Click **Tools, Options**.
- 2 On the **System Options** tab, click **View Rotation**.
- 3 Move the **View animation speed** slider to set the **View animation speed** to **Slow**, then click **OK**.
- 4 Open the part **Select_loop.sldprt**.
- 5 Click **Isometric**  on the Standard Views toolbar.

The part animates into the isometric view.

- 6 Click one of the top faces of the part to select it, then click **Zoom to selection** .

Note how the change of views is displayed in animated steps.

TIP: To immediately snap to the finished position of an animated view, without viewing the animation, press and hold the **Esc** key before the animation sequence, or press the **Esc** key during the animation sequence.

Web Folders

Web folders is a new SolidWorks tool that allows multiple users to share and work on SolidWorks part, assembly, or drawing documents, as well as other file formats, across the Internet. You must have web folders active on your Microsoft Internet Explorer to use this tool. Web folders is also available through SolidWorks Explorer.

To save a file to your web folder:

- 1 Create a web folder on a computer with a web server. See your systems administrator to help you set up a web folder on a server.
- 2 In SolidWorks, open the file that you want to save to the web folder.
- 3 Click **File, Save to Web Folder**.
The **Save to Web Folder** dialog box appears. The file name of the open file appears in the **File name** box.
- 4 Click the arrow in the **URL** box, and enter the URL of the web folder on the server.
If the web server requires a password, the **Enter Network Password** dialog box appears. If it does not require a password, go to step 6.
- 5 Enter the **User Name** and **Password**, then click **OK**.
The web folder you created opens, displaying default folders.
- 6 Double-click the web folder where you want to save the file.
- 7 Click **Save**.
The file is saved to the web folder.

To open a file from the web folder:

- 1 Click **File, Open From Web Folder**.
The **Open From Web Folder** dialog box appears.
- 2 Click the arrow in the **URL** box, and enter the URL of the web folder on the server.
If the web server requires a password, the **Enter Network Password** dialog box appears. If it does not require a password, go to step 4.
- 3 Enter the **User Name** and **Password**, then click **OK**.
The web folder you created opens, displaying the default folders.
- 4 Double-click the web folder that contains the file.
All files in that folder appear in the dialog box window.
- 5 Select the desired file and click **Open**.
The file opens in SolidWorks.

Sketching

There are new 2D and 3D sketching functions in SolidWorks 2001.

- Automatic PropertyManager
- Sketch Modes
- 3D Sketches
- Sketches in Part and Assembly Documents

Automatic PropertyManager



The PropertyManager now opens automatically with many sketch operations such as points, lines, arcs, circles, and other geometric entities. In previous releases of SolidWorks, you enabled the PropertyManager by selecting the **Enable PropertyManager** check box under **Tools, Options, System Options, General**. This option has been removed.

Double-clicking a sketch entity opens the PropertyManager and displays the entity's relations and parameters. At the same time, callouts in the graphics area name the entities in the relations. If you click on the relation in the PropertyManager, the related callout is highlighted.



Sketch Modes

For new dimension functionality available for sketching, see **Dimensions** on page 5-9 and **Ordinate Dimensions** on page 2-7.


Sketching without Preselection

You can now select a sketch, plane, face, or edge before *or* after selecting the **Sketch**  tool or **3D Sketch**  tool to create or edit a sketch. Previously you were required to preselect the entity.

To start a sketch without preselecting an entity:

- 1 Open a new part.
- 2 Click **Sketch** .
- 3 Select a tool on the Sketch tool toolbar and proceed to create a sketch.
- 4 Click **Sketch**  again to exit from the sketch.

To edit an existing sketch or create another sketch:

- 1 In a part that contains a sketch, click **Sketch** .
The **Edit Sketch** PropertyManager appears and prompts you to select a plane, face, or edge to create a new sketch, or to select an existing sketch to edit.
- 2 Click a plane in the FeatureManager design tree and create another sketch.

Alternate Geometry Creation

A new mode for creating 2D sketch entities, called click-click, has been added. SolidWorks responds to cues from you to determine whether you wish to be in click-drag mode (the previous behavior) or the new click-click mode as follows:

- If you click the first point and drag, you are in click-drag mode.
- If you click the first point and release the pointer, SolidWorks recognizes the click-click mode.



The click-click mode is not available for 3D sketching.

When the line and arc tools are in click-click mode, they create a chain of segments as you continue to click. To terminate a sketch chain, do one of the following:

- Double-click to terminate the chain of entities and leave the tool active.
- Right-click and select **End chain**, which is the same as double-clicking.
- Press **Esc** to terminate the chain and also release the tool.
- Move the pointer outside the view window to stop dragging. You can then choose another tool.

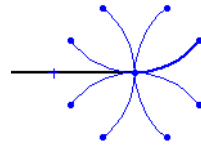
NOTE: In previous releases, the default method of creating sketch entities was to click-drag. In SolidWorks 2000, a sketch option, **Alternate spline creation**, allowed you to use a click-click mode for creating splines. The **Alternate spline creation** sketch option has been removed from the Tools, Options dialog box.

To create sketch lines in click-click mode:

- 1 Open a new part.
- 2 Click **Sketch** .
- 3 Click **Line**  or **Tools, Sketch Entity, Line**.
- 4 Click in the graphics area to start a line.
- 5 Release the mouse button, move the pointer to another position in the graphics area, and click to form a line.
- 6 Repeat step 5 to continue to create line segments.
- 7 Double-click in the graphics area to terminate the chain. Start another line. Draw one horizontal line segment to use in the next example.
- 8 Press **Esc** to terminate the chain and release the **Line** tool.


Tangent Arc Intent Zones

When you sketch a Tangent Arc, the SolidWorks software infers from the pointer motion whether you want a tangent or normal arc. There are four intent zones, with eight possible results as shown.



Tangent and normal arc intent zones

To sketch a tangent arc:

- 1 Click **Tangent Arc**  or **Tools, Sketch Entity, Tangent Arc**.
- 2 Select the end point of the horizontal line from the previous example.
You can start from the end point of any existing sketch entity (line, arc, spline, and so on).
- 3 Move the pointer away from the end point, either dragging or in the alternate click-click mode.
 - Moving the pointer in a tangent direction creates a tangent arc.
 - Moving in a normal direction creates a normal arc.
 - You can change from one to the other by returning to the end point and moving away in a new direction.
 - Preview both tangent and normal arcs.
- 4 Return to the end point, and now move the pointer tangent to the line until a 90° angle is inferenced, and complete a tangent arc as shown.
- 5 Continue to the example in the next section.




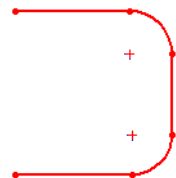
Tangent arc added to a line

Autotransitioning between Lines and Arcs

You can transition from sketching a line to sketching an arc, and vice versa, without selecting an arc tool. You can also toggle between line and arc with the **A** key.

To continue from the previous section:


- 1 Click **Line**  and click the end point of the arc.
- 2 Move the pointer away from the end point, sketching a line.
- 3 Move the pointer away from the line end point, then return and move away again, forming a 90° arc.
- 4 Move the pointer away from the arc end point. Press **A** several times, noticing that line and arc alternate each time you press the key.
- 5 Sketch a horizontal line, completing the sketch as shown.



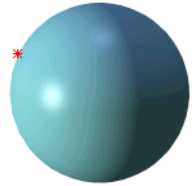
Autotransitioning between lines and arcs

3D Sketching

Relations in 3D Sketches

When you sketch a **3D Point**  on a face or surface, the point automatically has a coincident relation.

You can also add the relation manually.



3D point coincident with sphere

Proportional Splines

You can now add dimensions and relations to internal points of 2D and 3D proportional splines.

Iso-parametric Curves

You can extract iso-parametric (UV) curves from a face or surface. You can specify evenly spaced locations (mesh) or one location (position). If the face or surface is changed and the **Constrain to model** option is selected, the extracted curves update.


Applications of this functionality include extracting the curves for imported surfaces and then performing localized cleanup using the face curves. The **Ignore holes** option is useful for imported surfaces.

To extract iso-parametric curves:

- 1 Open **surface_for_iso.sldprt**
- 2 Click **Face Curves**  on the Sketch Tools toolbar or **Tools, Sketch Tools, Face Curves**.

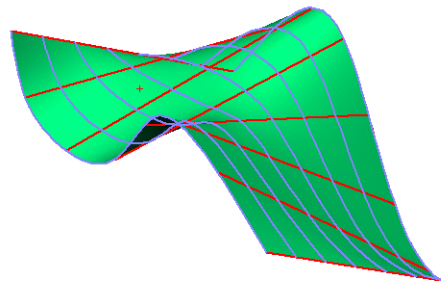
The **Face Curves** PropertyManager appears.

- 3 Click the surface.


A preview of the iso-parametric curves appears on the surface. Under **Selections** in the PropertyManager, **Face<1>** appears in the **Face**  box.

The curves on the surface are one color in one direction and another color in the other direction. The colors correspond to the colors under **Selections** in the **Face Curves** PropertyManager.

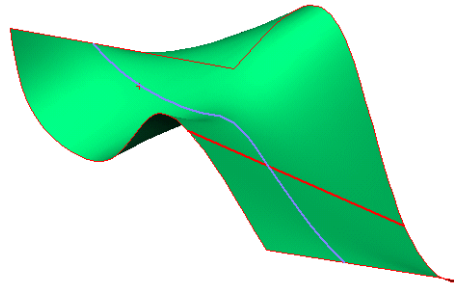
- 4 Under **Selections**, change the number of integer **Mesh** curves to 8 in each direction, as shown.



Mesh face curves

- 5 Select **Position** and drag the curves at the intersection point, or change the percentages, measured from the bottom edge (**Direction 1 Position**) and the right edge (**Direction 2 Position**), in the boxes.
- 6 When you are satisfied with the preview, click **OK** .

Each separate curve becomes a 3D sketch. If you are editing a 3D sketch when you invoke the command, all extracted curves are added to the active 3D sketch.







Position face curves

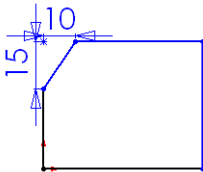
Sketches in Part and Assembly Documents

Chamfer

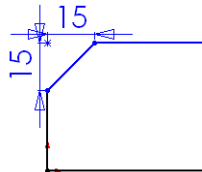
A chamfer tool is now available for 2D sketches. You can create **Angle-distance** or **Distance-distance** chamfers between two adjacent sketch entities.

To create a sketch chamfer:

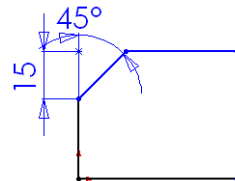
- 1 Starting at the Origin, sketch a rectangle without dimensions.
- 2 In the graphics area, select the vertical line from the origin and the top horizontal line.
- 3 Click **Chamfer**  on the Sketch Tools toolbar, or **Tools, Sketch Tools, Chamfer**.
- 4 Under **Chamfer Parameters**, do the following:
 - a) Click to clear the **Equal distance** check box.
 - b) Make sure **Distance-distance** is selected.
 - c) Enter 15mm as the **Distance 1** .
 - d) Enter 10mm as the **Distance 2** .
- 5 Click **OK**  to create the sketch chamfer.



Distance-distance chamfer



Distance-distance chamfer with equal spacing




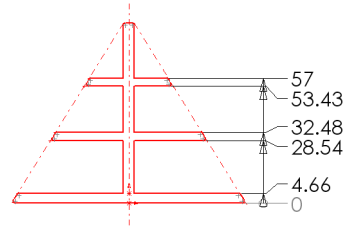
Angle-distance chamfer

Ordinate Dimensions

You can now add ordinate dimensions to sketches. Unlike ordinate dimensions in drawings, ordinate dimensions in part and assembly sketches are driving dimensions.

To add ordinate dimensions to a sketch:

- 1 Click **Dimension** .
- 2 Right-click in the graphics area and select **Ordinate Dimension**.
- 3 Select a line for the zero ordinate, then click to place the zero ordinate dimension.
- 4 Select other entities for ordinate dimensions.



Ordinate dimensions in a sketch



All the capabilities of ordinate dimensions in drawings are also available in sketches. If the zero ordinate is moved, all other dimensions update automatically (see **Dimensions** on page 5-9). You can choose whether to display the dimensions as chain dimensions in the **Dimension Properties** dialog box.

Offset Entities


The **Offset Entities** tool now includes:

- Dynamic preview
- Bidirectional option
- Select chain option
- Offset of construction geometry


To create an offset entity:

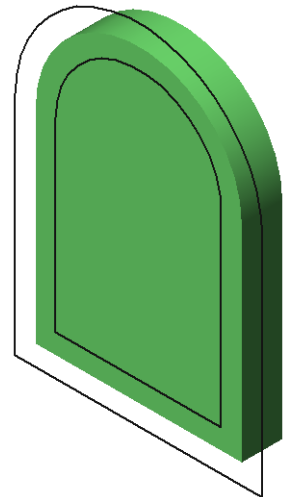
- 1 Open **sketch_offset.sldprt**.
- 2 Click **Sketch** .
- 3 Select the front face of the part.
- 4 Click **Offset Entities**  on the Sketch Tools toolbar.

The **Offset Entities** PropertyManager appears. The selection filters for **Edges**, **Faces**, and **Sketch Segments** are activated automatically.

- 5 Under **Parameters**, do the following:
 - Select **Bidirectional**.
 - Clear **Select chain**.
 - Set **Offset Distance**  to 5mm.


You can also set the distance by moving the pointer.

- 6 Click **OK**  to create the offset entity.



Bidirectional offset entity


Redo

Redo  is now available for sketches in parts and assemblies. **Redo** is also available on the **Edit** menu. The shortcut key is **Ctrl+Y**.




Reference Plane Created Normal to an Edge

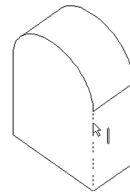
You can sketch on a reference plane normal to an edge. The plane is created automatically.

To sketch on a reference plane normal to an edge:

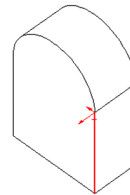
- 1 Open the part **sketch_normal_plane.sldprt**.
- 2 Select a point on the edge, as shown, then click **Sketch**  .

A reference plane normal to the selected edge is created at the edge's end point closest to the selected point. Note the origin symbol for the new plane.

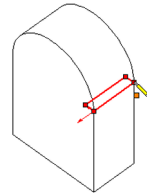
- 3 Sketch a rectangle on the new plane, as shown.
- 4 Click **Extruded Cut**  .
The **Cut-Extrude** PropertyManager appears.
- 5 Under **Direction 1**, set the **End Condition** to **Through All**.
If necessary, select **Reverse direction**  so that the cut points up.
- 6 Click **OK**  .
The cut is made normal to the new plane, as shown.



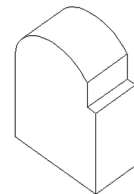
Select edge



Plane normal to edge



Sketch rectangle



Cut normal to plane

Features, Parts, and Surfaces

Features are the individual shapes that, when combined, make up the part. The 3D part is the basic building block of the SolidWorks mechanical design software. Surfaces are zero-thickness geometry that can be used to create solids. Changes include the following:

- Apply user interface through the PropertyManager
- Create sweeps with multiple contours
- Implement thin features for sweeps
- Implement thin features for lofts
- Skip pattern instances without editing
- Create hole wizard holes on planes
- Control tangency magnitude for lofts (solids or surfaces)
- Display configuration-specific information, using embedded custom dimensions and mass property custom parameters
- Delete interior holes on a surface body
- Create Mid Surface between existing surface bodies

PropertyManager User Interface

Many commands have shifted from a dialog box to the PropertyManager, offering a clearer user interface. The PropertyManager includes the following capabilities:

- Graphics area that remains clear of dialog boxes, facilitating feature manipulation and dynamic previews
- Functions that are intuitive, with visible, icon-driven access
- Feature parameters that can be set in multiple ways

Features that use the PropertyManager include:

- Extrude - base/boss and cut
- Revolve
- Chamfer
- Fillet
- Rib
- Shell
- Sweep
- Loft
- Linear pattern
- Circular pattern
- Sketch driven pattern

Using the PropertyManager

Functionality in the PropertyManager includes the following enhancements for creating or editing features:


- **Flyout FeatureManager design tree** - feature selection is facilitated when the PropertyManager is active (see **FeatureManager Design Tree** on page 1-5)
- **Dynamic handles** - dynamic display in the graphics area when you drag the handles to change the dimension (see **Handles** on page 1-2)
- **Callouts** - dynamic dimensions that you can edit as you drag the feature (see **Callouts** on page 1-2)

When creating features such as an extrude or a sweep, you have the option to either:


- Enter data through the PropertyManager
- Enter data in the callouts
- Modify the feature graphically, using the handles to drag the feature
- Use shortcut menu functions to set the parameters of the feature
- Expand or collapse the different areas in the PropertyManager

Extrude Boss/Base Features

You can create a base extrude from the graphics area as follows:

- 1 Sketch a rectangle, dimension, and click **Extruded Boss/Base** , or **Insert, Boss/Base, Extrude**.

The **Base-Extrude** PropertyManager is displayed.

- 2 To change the extrusion type from the default **Blind**, right-click in the graphics area and select another **End Condition**.
- 3 In the graphics area, drag the sketch handle to adjust to the desired extrude dimension, as displayed by the value in the callout.
- 4 To accept the extrude dimension click to display .

This locks your selection.

To unlock, click the handle again and drag to change the extrude dimension.



- 5 Right-click and select **OK**  to create the boss/base extrude.

You can also combine the dynamic functionality in the graphics area with the PropertyManager, as follows:



- 1 Click **Extruded Boss/Base**  to extrude the sketch.

The **Base-Extrude** PropertyManager is displayed.



- 2 Under **Direction 1**, set the following, as needed:


- **End Condition**
- **Depth** 
- **Draft On/Off**  - **Draft Angle**

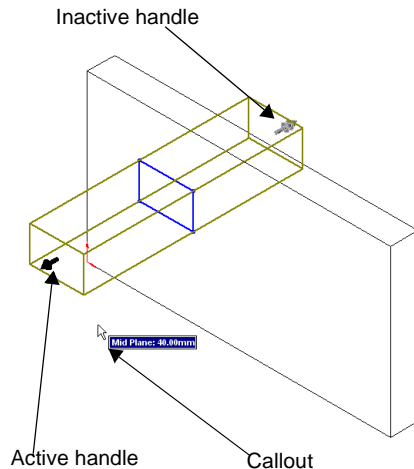
- 3 If necessary, select the **Direction 2** check box and set the following:

- **End Condition**
- **Depth** 
- **Draft On/Off**  - **Draft Angle**

- 4 If necessary, select the **Thin Feature** check box and set the following:

- **Type** - select **One-Direction**, **Mid-Plane**, or **Two-Direction**
- **Thickness** - set the **Direction 1 Thickness** , and if appropriate, the **Direction 2 Thickness** 
- **Cap Ends** - select the check box to create cap ends

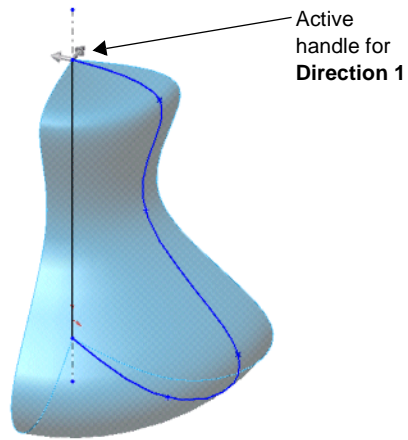
- 5 Click **OK**  to extrude the feature.



Revolve Features



The Revolve feature now displays on the PropertyManager. As with other features that now use the PropertyManager, you can use:

- **PropertyManager** to enter data for **Revolve Type**, **Direction 1**, or **Direction 2**
- **Graphics area** to select a handle, drag and change the angle of the revolve, or type to edit the value in the callout. Right-click to access menu functions.

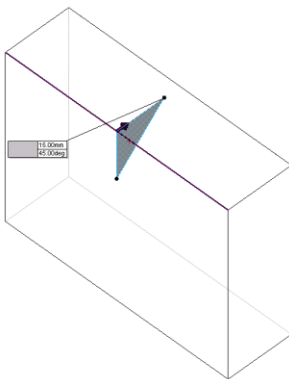


Chamfer Features

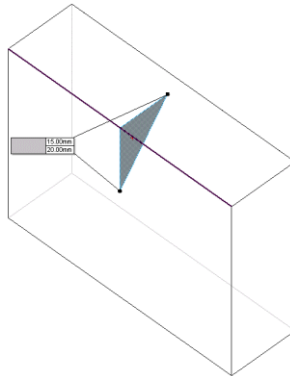
You can use the handles in the graphics area to change the **Angle distance**, the **Distance distance**, or the **Vertex** by selecting each handle in turn, and dragging to the desired value. You can also enter the value in the callout

- 1 Sketch a rectangle, extrude, and click **Chamfer** , or **Insert, Features, Chamfer**.
- 2 Under **Chamfer Parameters**, do the following:
 - a) Select the type of chamfer you want to apply: **Angle distance**, **Distance distance**, or **Vertex** (or right-click in the graphics area to select the chamfer type).
 - b) In the graphics area, select the edges, faces, or the vertex you want to use in the chamfer to display under **Edges and Faces or Vertex** .

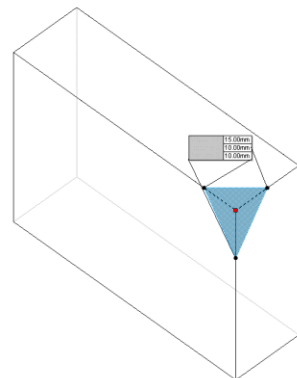
A dynamic triangle displays on the model. The shape and position of the triangle depends on the type of chamfer you select, and the dimensions you input.









Distance angle chamfer



Distance distance chamfer



Vertex chamfer

- 3 Enter the dimensions in the callout. Note that the values change in **Chamfer Parameters (Distance 1 , Distance 2 , Distance 3 , or Angle )**.
- 4 Click  to lock the values.
- 5 Click **OK**  to create the chamfer.

Fillet Features




Fillet features include two enhancements.

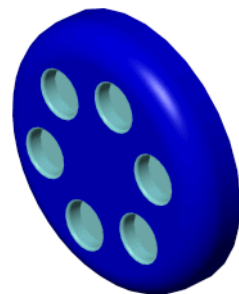
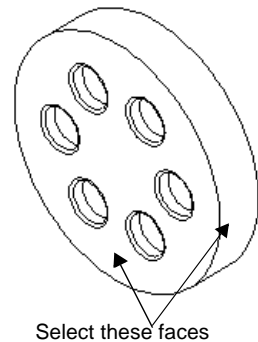
- All Fillet features now use the PropertyManager.
- A new default function, **Keep features**, was added to fillets under **Options**.

Fillet Feature in the PropertyManager

When you select the **Fillet type (Constant radius, Variable radius, or Face fillet)**, **Items to Fillet** and **Fillet options** change dynamically in the PropertyManager for the fillet type.




To create a Face Fillet:

- 1 Open **fillet_face_PM.sldprt**.
- 2 Click **Fillet** , or **Insert, Features, Fillet/Round**.
- 3 Under **Fillet Type**, click **Face fillet**.
- 4 Under **Items to Fillet**, do the following:
 - a) Set the **Radius**  to 5mm.
 - b) Click **Face Set 1** and select the flat face from the graphics area.
 - c) Click **Face Set 2**, and select the narrow circular face in the graphics area.
- 5 Click **OK**  to create the fillet.



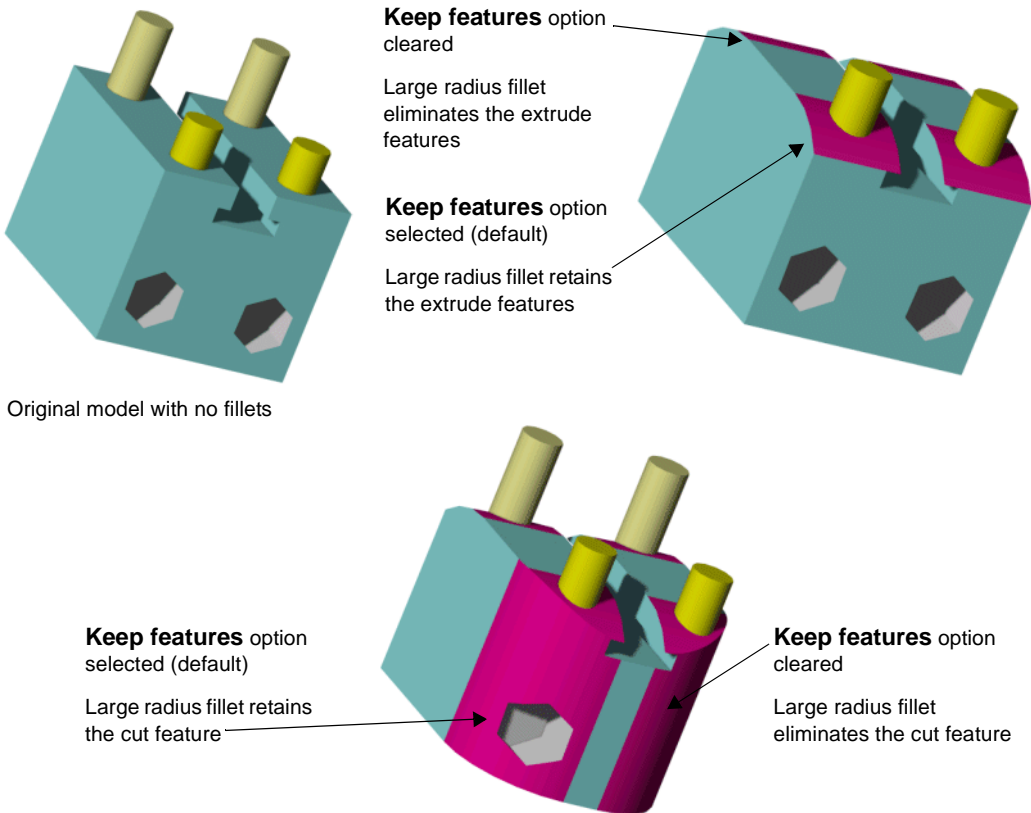
Other fillet functions

Other fillet functions now using the PropertyManager include the following:

- ❑ **Setback fillet** - Under **Setback Parameters**, select the **Setback Vertices**  from the graphics area where the three edges you selected to fillet meet. Then assign **Setback Distances**  to each edge (or if all setbacks are equal, select a **Distance**  and click **Set All**).
- ❑ **Fillet Options** - **Fillet Options** changes dynamically to reflect the type of fillet. For example, when you create a **Face fillet**, the options listed are **Hold line** and **Help point**. When you select a **Variable radius** fillet, the options listed control the **Overflow type**.

Keep Features Option







A new default option, **Keep features**, keeps cuts and extrudes intact when large radius fillets are applied to an edge. Previously, a large radius fillet that overlapped a cut or extrude feature would eliminate that feature from the model. This function is only active with **Constant radius** fillets.

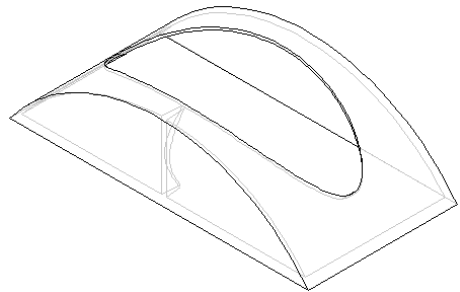
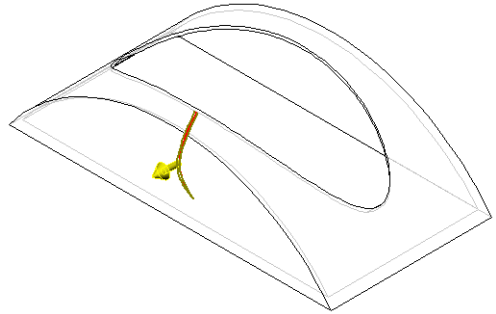


Rib Features

Functionality for rib features has moved to the PropertyManager.

To create a rib:



- 1 Open **rib_PM.sldprt**.
- 2 Select **Sketch3** in the FeatureManager design tree.
- 3 Click **Rib** , or **Insert, Features, Rib**.
- 4 Under **Parameters**, do the following:
 - a) Click **Create rib on both sides of the sketch**  as **Thickness**.
 - b) Enter 2mm as **Rib Thickness** .
 - c) Click **Normal to Sketch**  as **Extrusion direction**.
 - d) Click **Draft On/Off**  and enter 5°.
- 5 Click **OK**  to create the rib.





Shell Features

Functionality for shell features has moved to the PropertyManager.

Under **Parameters**, select the following:

- **Thickness**  - use the handles in the graphics area, or type in a value.
- **Faces to Remove**  - select the faces to remove in the graphics area (leave blank to create a hollow feature).
- Select the **Shell outward** check box if necessary.

Under **Multi-thickness Settings**, select the following:

- **Multi-thickness(es)**  - enter the thickness values.
- **Multi-thickness Faces**  - enter the faces.

Sweep Features

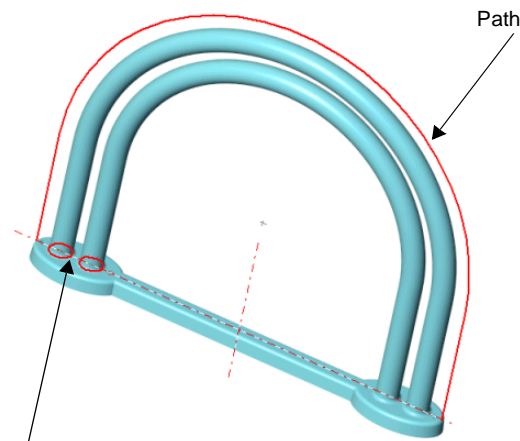
Sweep features include the following new functionality:

- move to the PropertyManager
- create sweeps with multiple contours
- create sweeps using thin features

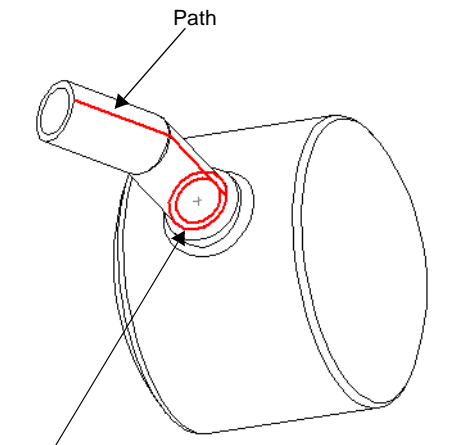
Sweeps with Multiple Contours

You can create sweeps with multiple contours. The requirements include:

- define a section and a path
- create one or more contour profiles
- close the contours for solid sweeps



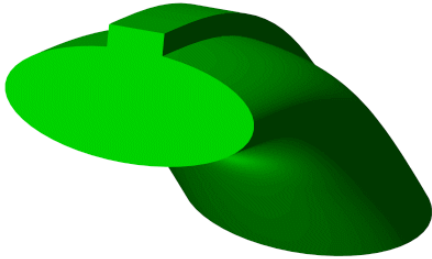
Multi-section closed contour with separate curves



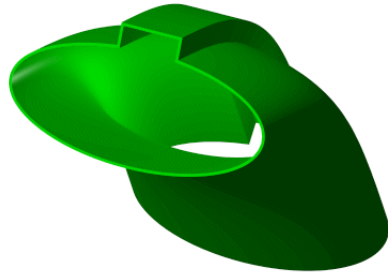
Multi-section closed contour with nested curves

Sweeps with Thin Features

When you select **Thin Feature** in the PropertyManager, you can create sweeps with thin walls.







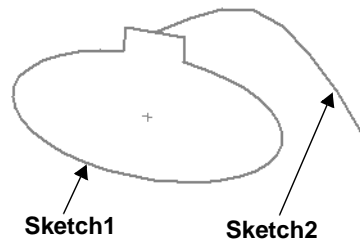
Sweep with solid feature



Sweep with thin feature





To create a sweep using thin feature:

- 1 Open **sweep_PM_thin.sldprt**.
- 2 Click **Sweep** , or **Insert, Base, Sweep**.
- 3 Under **Profile and Path**, do the following:
 - a) Click **Profile**  and select **Sketch1** in the graphics area.
 - b) Click **Path**  and select **Sketch2** in the graphics area.
- 4 Select the **Thin Feature** check box, and accept the default thickness.
- 5 Click **OK**  to create the sweep with thin feature.



Other sweep options:

Other sweep functions now using the PropertyManager include the following:

- Options** - Select the appropriate **Orientation/twist control**. If necessary, click **Maintain tangency** and **Advanced smoothing**.
- Guide Curves** - Click **Guide Curves**  and select sketches from the graphics area to use as guide curves. To reorder the sketches, use the **Move Up**  and **Move Down**  arrows. Use **Show Sections**  to view and troubleshoot the sections of a sweep that uses guide curves.

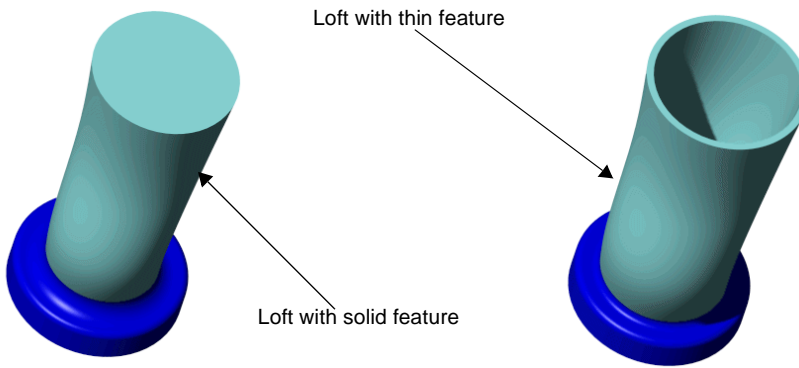
Loft Features

Loft features include the following new functionality:

- move to the PropertyManager
- create lofts using thin features
- control tangency for solid and surface lofts


Lofts with Thin Features



When you select **Thin Feature**, you can select the **Type** and the **Wall Thickness**.

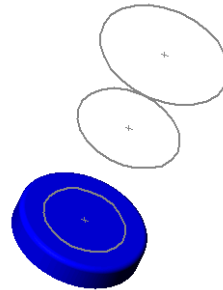


To create a loft:

- 1 Open **loft_PM_thin.sldprt**.
- 2 Click **Loft** , or **Insert, Boss, Loft**.
- 3 Click  at the bottom of the **Loft** PropertyManager to open the flyout FeatureManager design tree.
- 4 In the FeatureManager design tree, click **Sketch2**, **Sketch3**, and **Sketch4**.




The profiles are added to **Profile** .

- 5 Select the **Thin Feature** check box, and keep the default value of 1.00mm under **Direction 1 Thickness** .
- 6 Click **OK**  to create the loft with thin feature.



Other loft functions:



Other loft functions now using the PropertyManager include the following:

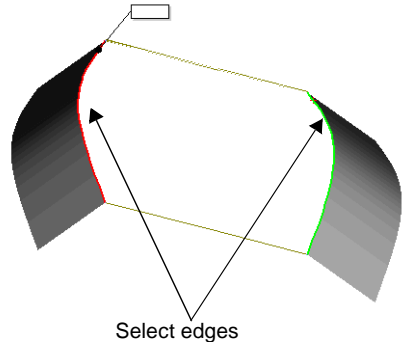
- Guide Curves** - Click **Guide Curves**  and select sketches from the graphics area. Use the **Move Up**  and **Move Down**  arrows to reorder the sketches.
- Options** - Select the **Maintain tangency**, **Advanced smoothing**, and **Close loft** check boxes.


Control Tangency in Lofts


You can control the tangency magnitude and direction of a loft. You can control the tangency magnitude between sketches, faces, or edges on surfaces.

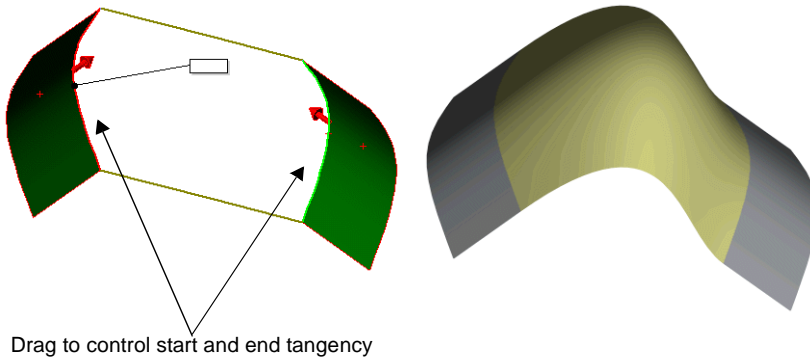
To control tangency magnitude in a loft:

- 1 Open `loft_control_tangency.sldprt`.
- 2 Click **Lofted Surface** , or **Insert, Surface, Loft**.
- 3 Under **Profile**  ⁰, select the inner edge of each surface, as shown.
- 4 Under **Start/End Tangency**, do the following:
 - a) For **Start tangency type** and for **End tangency type**, select **All Faces**.
 - b) Set **Start Tangent Length** to 4.00, and set **End Tangent Length** to 1.00. Use the handles in the graphics area and drag to change the value, or enter the values directly in the PropertyManager.



You can also control the direction of the loft. Click **Reverse Tangent Direction**  for either **Start tangency type** or for **End tangency type**.

- 5 Click **OK**  to create the loft with controlled tangency.



Feature Patterns











In SolidWorks 2001, feature patterns include the following enhancements:

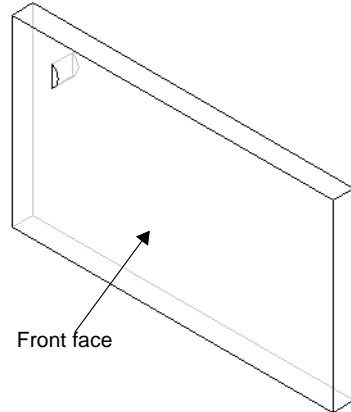
- Linear, circular, and sketch driven patterns now use the PropertyManager.
- You can select to skip pattern instances while building the pattern. Previously, you had to create the feature pattern and then delete pattern instances by selecting them and pressing **Delete**.


Linear Patterns

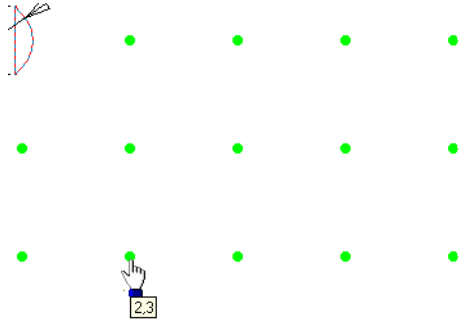
Linear patterns now use the PropertyManager. You can skip pattern instances during pattern propagation.


To create a linear pattern and skip instances:

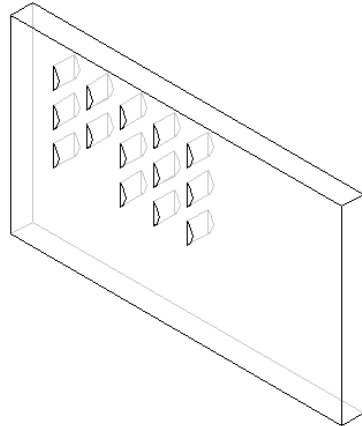
- 1 Open **linear_pattern_skip_PM.sldprt**.
- 2 Select the front face and click **Normal To** .
- 3 In the FeatureManager design tree, select **Cut-Extrude1**.
- 4 Click **Linear Pattern** , or **Insert, Pattern/Mirror, Linear Pattern**.
- 5 Under **Direction 1**, do the following:
 - a) Click **Pattern Direction** and select either horizontal edge. Note how this box is already active before you select it.
 - b) Make sure the pattern direction in the graphics area points right . Click **Reverse Direction** , if necessary.
 - c) Set **Spacing**  to 8mm (using either the callout or the PropertyManager).
 - d) Set **Number of Instances**  to 5.
- 6 Under **Direction 2**, do the following:
 - a) Click **Pattern Direction** and select either vertical edge.
 - b) Make sure the pattern direction in the graphics area points down . Click **Reverse Direction** , if necessary.
 - c) Set **Spacing**  to 8mm.
 - d) Set **Number of Instances**  to 3.



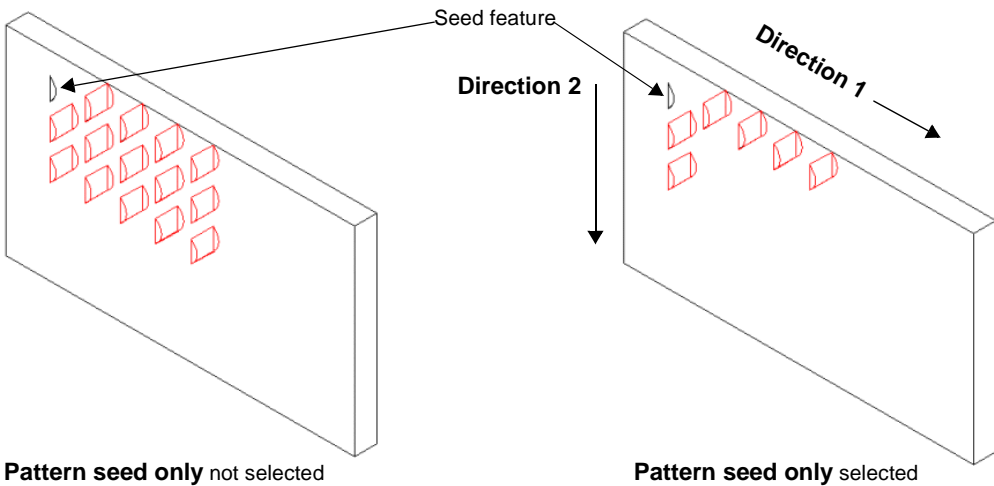
- 7 Click **Instances to Skip** , and in the graphics area select a pattern instance to skip, as shown. Note that the pointer includes a callout with the coordinates of the pattern instance.



- 8 Click **OK**  to create the linear pattern.







You can also pattern only the seed feature by clicking the **Pattern seed only** check box under **Direction 2**. Selecting the **Pattern seed only** check box, creates a linear pattern under **Direction 2**, without replicating the linear pattern created under **Direction 1**.

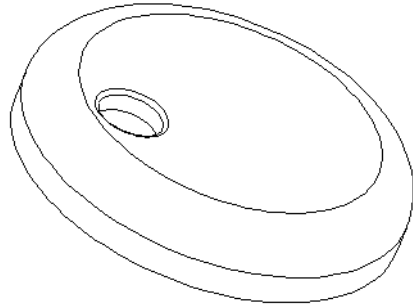


Circular Patterns

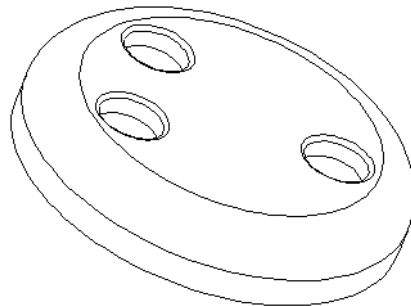
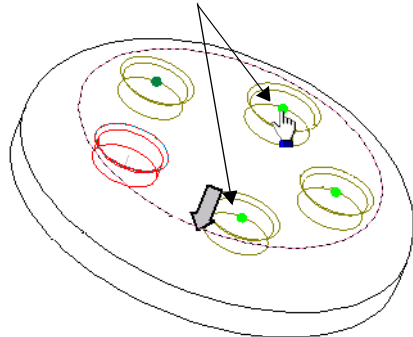
Circular patterns now use the PropertyManager. You can skip pattern instances during pattern creation.

To create a circular pattern and skip instances:

- 1 Open **circular_pattern_skip_PM.sldprt**.
- 2 Click **View, Temporary Axes**.
- 3 Press **Ctrl** and select **Cut-Extrude1** and **Fillet1** in the FeatureManager design tree.
- 4 Click **Circular Pattern** , or **Insert, Pattern/Mirror, Circular Pattern**.
- 5 Under **Parameters**, do the following:
 - a) Click **Pattern Axis** and select the axis of the base revolve in the graphics area.
 - b) Select the **Equal spacing** check box.
 - c) Set the **Number of Instances**  to 5.
- 6 Click **Instances to Skip** , and in the graphics area select the third and fifth pattern instances to skip, as shown. Note that the pointer includes a callout with the number of the pattern instance.
- 7 Under **Options**, select the **Geometry pattern** check box.
- 8 Click **OK**  to create the circular pattern.



Selected pattern instances to delete

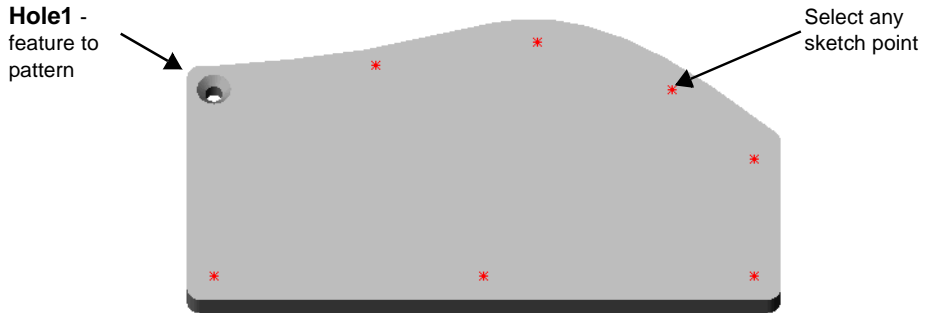




Sketch Driven Patterns

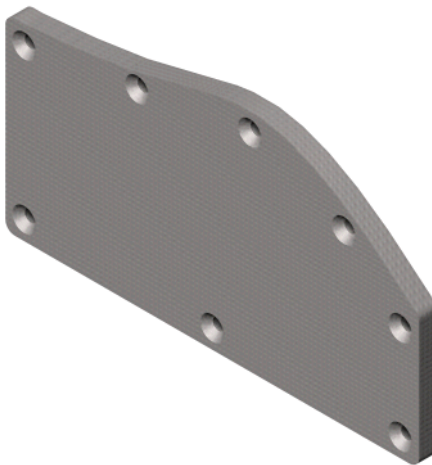
Sketch driven patterns have moved to the PropertyManager.

To create a sketch driven pattern:

- 1 Open **sketch_pattern_cover.sldprt**.

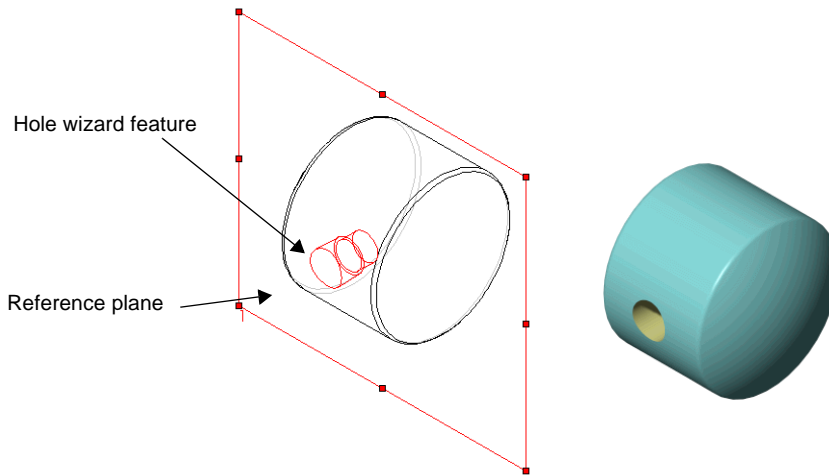


- 2 Click **Insert, Pattern/Mirror, Sketch Driven Pattern**.
- 3 Under **Selections**, do the following:
 - a) Click **Reference Sketch** , and select any of the sketch points in the graphics area.
 - b) Accept **Centroid** as the default **Reference Point**.
- 4 Click **Features to Pattern**, and in the graphics area, select **Hole1**.
- 5 Under **Options**, select the **Geometry pattern** check box.
- 6 Click **OK**  to create the sketch pattern.



Hole Wizard on a Plane

You can now create holes on a plane using the Hole Wizard. Previously, you could create them only on planar faces. This new functionality allows you to adapt Hole Wizard to non-planar faces, and to create holes at an angle to the feature.



See **Hole Wizard Holes as Assembly Features** on page 4-12, for using the Hole Wizard with assembly features.

Configurations

Several enhancements involving configurations and the display of summary information were added to SolidWorks 2001. These enhancements include:

- dimensions and mass custom properties
- custom properties in design tables
- dimension values for specified configurations

Dimensions and Mass Property Parameters

This function allows you to add dimensions and mass property parameters into values of custom properties.

Dimensions in Custom Properties

Formerly, if you wanted values reflected in a Bill of Materials, you needed to cut and paste these values. Now, changes to the dimensions in the part are associative, so values in the BOM are updated.

To add dimensions into custom properties:

1 Open **base_revolve_mass.sldprt**, and double-click **Base-Revolve** in the FeatureManager design tree to display the dimensions.

2 Click **File, Properties**, and in the **Summary Information** dialog box, select **Custom**.

3 Under **Name**, type **revolve1**, and click the 25mm dimension in the graphics area.

D4@Sketch1@Base_revolve_mass.SLDPRT is displayed in **Value**.

4 Place your cursor at the end of the dimension expression displayed in **Value**, press the spacebar, and type **millimeters**.

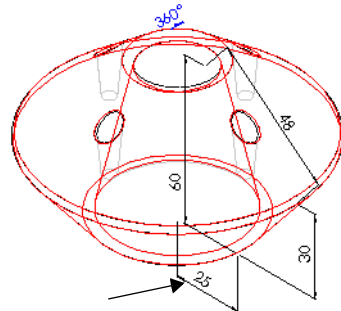
5 Click **Add**.

The name and the value, including what you typed in, is listed under **Properties**. Optionally, you can include more than one dimension in each value.

6 Click **OK**.

7 Change the dimension from **25** to **24**.

8 Open the **Summary Information** dialog box, and note the dimension value has updated.



Mass Properties

You can also insert system-defined, configuration-specific mass properties as a custom property. You can add the value using either the **Custom** or the **Configuration Specific** tabs of the **Summary Information** dialog box. Mass properties use the following, system-generated format: **SW-<mass property>@@configuration@model**

To add mass properties:

- 1 Click **File, Properties**, and in the **Summary Information** dialog box, select **Configuration Specific** (to display with other configuration-specific properties).
You can also select the **Custom** tab.
- 2 Under **Name**, type **Mass**.
- 3 Click **Mass Properties** and select **Mass**.
- 4 Click **Add** to display the mass property under **Properties**, along with the configuration-specific dimensions and the volume.
- 5 Click **OK**.

Custom Properties in Design Tables

To add configuration-specific properties into a custom property, you specify the configurations by typing the custom properties into the design table. The SolidWorks software then adds the properties automatically to the **Configuration Specific** tab in the **Properties** dialog box.

To create custom properties in a design table:

- 1 In the design table, create a row for as many configurations as you have of your model.
- 2 Create a column for each custom property you intend to include in the design table. Use the following format for each column header:
\$prp@<name of the custom property> (for example: **\$prp@length** to add the **length** custom property)
- 3 Enter the dimension associated with each custom property in the appropriate cell. Use the following format for each dimension:
"<dim name>@@configuration@model" (for example: **"D2@Sketch1@@First instance@box.SLDPRT"**)

NOTE: Make sure to include opening and closing quotes (" ... ") when entering custom properties in a design table. Also, make sure the file extension (SLDPRT) is uppercase.

- 4 After you have set up the custom property-specific information, add the standard design table information by entering the dimension names and values in new columns.

Custom Property names: \$prp@<user defined term>

Standard dimension values

	A	B	C	D	E	F	G
1	\$prp@length		\$prp@height	\$prp@depth	D2@Sketch1@D1@Base-Extrude	D1@Sketch1@D1@Base-Extrude	D1@Sketch1@D1@Base-Extrude
2	First Instance "D2@Sketch1@@First Instance@box.SLDPRT"	"D1@Sketch1@@First Instance@box.SLDPRT"	"D1@Base-Extrude@@First Instance@box.SLDPRT"	80	50	20	
3	Second "D2@Sketch1@@second@box.SLDPRT"	"D1@Sketch1@@second@box.SLDPRT"	"D1@Base-Extrude@@second@box.SLDPRT"	100	25	10	
4	Third "D2@Sketch1@@third@box.SLDPRT"	"D1@Sketch1@@third@box.SLDPRT"	"D1@Base-Extrude@@third@box.SLDPRT"	60	40	30	


\$prp@length

First instance "D2@Sketch1@@First instance@box.SLDPRT"

Second "D2@Sketch1@@second@box.SLDPRT"

Third "D2@Sketch1@@third@box.SLDPRT"

To use custom properties in design tables:

- 1 Open **box.SLDPRT**.
- 2 Click **Edit, Design Table** to display the design table.
The design table includes the custom property names, followed by the standard dimension values used in all design tables.
- 3 Click the **ConfigurationManager**  tab, and note the three configurations.
- 4 Double-click **second** to display the second configuration, and click anywhere in the graphics area to close the design table.
- 5 Click **File, Properties**, and in the **Summary Information** dialog box, select **Configuration Specific**.
Under **Properties**, the dimensions for the **second** configuration are retrieved from the design table and automatically displayed.
- 6 Click **OK**.
- 7 Open **box.sldrw** to see custom properties applied to a drawing.

Values for Specified Configurations

When creating a new model configuration and editing the dimensions for the new configuration, you can now set the feature properties for:

- **This configuration** - applies the dimension to the current configuration.
- **All configurations** - applies the dimension to all configurations of the model.
- **Specified configurations** - allows you to apply the dimension to selected configurations.

You can also select the current configuration, all configurations, or specific configurations when you suppress, or unsuppress a feature (or unsuppress with dependents). Select the feature, and click **Edit**, and then select to **Suppress**, **Unsuppress**, or **Unsuppress with Dependents** by configuration.

Surface Features

Changes to surface features include the following:

- Move user interface to the PropertyManager
- Delete hole in a surface
- Create Mid Surface between reference faces

PropertyManager Interface

The following surface features now use the new PropertyManager user interface:

- Surface extrude
- Surface loft
- Surface extend
- Surface trim

Functionality using the PropertyManager for surfaces is similar to other features. You can:

- use the handles in the graphics area
- enter values using the callouts or through the PropertyManager
- use shortcut menu functionality

Delete Hole

You can now select any closed profile hole on a surface, and delete the hole by clicking **Delete**, using the **Choose Option** dialog box, and selecting **Delete Hole(s)**.

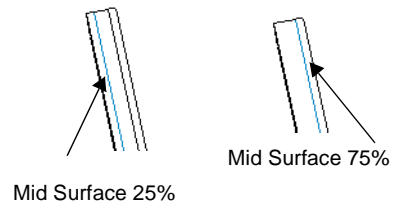


Mid Surface

The Mid Surface tool is useful with finite element modeling. It enables you to create mid-surfaces between suitable selected face pairs. Suitable face pairs should be offset from each other. Examples of suitable face pairs include two parallel planes, or two concentric cylindrical faces.

The resulting surface includes all the same attributes as any surface created in SolidWorks. Mid Surface parameters include:

- Select an individual or multiple face pairs in the graphics area. Individual faces are displayed in **Face 1** and **Face 2**; face pairs are listed in **Face pairs**.
- Locate all selected face pairs in a part, by using **Find Face Pairs** to have the system scan the model for face pairs to use. **Find Face Pairs** automatically filters out any unsuitable face pairs.
- Edit an existing face pair by selecting the set in **Face pairs**. The selected faces are displayed in **Face 1** and **Face 2**. Select new faces and click **Update Pair** to update **Face pairs** with the new set.
- Select where to create the Mid Surface, relative to the two selected faces, using **Position**. The default is 50% (halfway between **Face 1** and **Face 2**).
- Specify a **Recognition threshold** for selected faces. This is based on combining the **Threshold Operator** (= equal, < less than, <= less than or equal to, > greater than, >= greater than or equal to), and the **Threshold Thickness**. The **Recognition threshold** function is used to filter what is found when you select **Find Face Pairs**.
- The **Knit surfaces** check box is set by default to create a knit surface.




Assemblies

This chapter describes general information about working with assemblies of any kind. The following enhancements are discussed:

- ❑ Hiding *update holders*
- ❑ *Mirroring* components
- ❑ Setting components to *lightweight*
- ❑ Setting sub-assemblies to be *flexible*
- ❑ *Moving and rotating* components
- ❑ Mating components, including *new mates*, enhancements to existing mate types, and the ability to *switch mates*
- ❑ Stopping components at a *specified distance*
- ❑ Adding *hole wizard holes* as assembly features
- ❑ Deriving *component patterns* from assembly features


General Enhancements

Hide Update Holders


When you create a feature or add a relation in the context of an assembly, an icon  appears in the FeatureManager design tree to indicate an in-context reference. To reduce the size of the FeatureManager design tree, you can hide all of these in-context icons.

NOTE: This does not break or lock the external references; this only hides the markers.

To hide or show update holders:

To hide the holders, right-click the top-level assembly icon  and select **Hide update holders**.

- or -

To show the holders, right-click the top-level assembly icon  and select **Show update holders**.

Mirror Components

You can create new components by mirroring existing part or sub-assembly components. The new components can either be a copy or a mirror of the original components. If the original component changes, so does the copied or mirrored component. The differences are as follows:

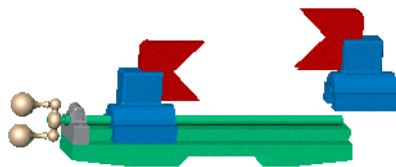
Copy	Mirror
No new part or assembly document is created.	A new document is created.
The geometry of the new component is identical to the original component; only the orientation of the component is different.	The geometry of the new component is mirrored; thus it is different from the original component.

Additionally, you can preserve any mates between the selected components when you mirror more than one component. Select the **Recreate mates to new components** check box. If the mates cannot be recreated due to geometry conditions, a dialog box appears.

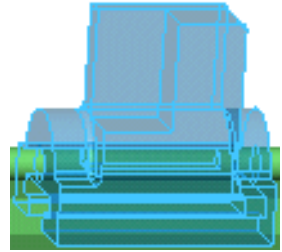
To copy a component:

- 1 Open `\vise\vise.sldasm`.
- 2 Delete the **support1<1>** component from the FeatureManager design tree.
- 3 Click **Insert, Mirrored Components**.

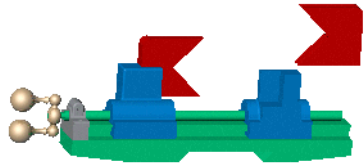
The **Mirror Components** dialog box appears.



- 4 Select the **Center** plane as the **Mirror plane**, and select **support2<1>** as the **Components to mirror**.
- 5 Make sure the icon next to **support2-1** is indicating the component will be copied.
- 6 Click **Next**.
- 7 Click or to toggle through the possible alignments of the component until the preview appears as shown, then click **Finish**.



A copy of **support2** appears in the assembly in the orientation you specify. The geometry of the component is unchanged from the original, only the orientation of the component is different.

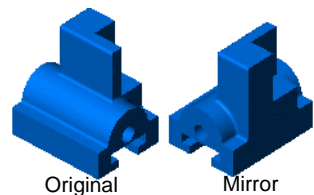


To mirror a component:

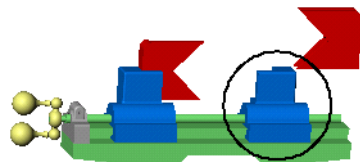
- 1 Delete **support2<2>** from the assembly.
- 2 Repeat steps 3 and 4 from the previous example.
- 3 Click the icon next to **support2-1** to toggle it to indicating the component will be mirrored.
- 4 Click **Next**.
- 5 In the next dialog box, do the following:
 - a) Set the file name to **newsupport**.
 - b) Make sure **Create New File** is selected to create a new file of the mirrored component.
 - c) Click **Finish**.

A new part document appears with the mirrored component. Note that the geometry of the component has been altered to create a truly mirrored component.

Additionally, the new component appears in the assembly.



- 6 Close the assembly *without* saving. You use this assembly in other examples in its original version.



Lightweight Components

There are new functions for lightweight components available in SolidWorks 2001:

- ❑ **Set lightweight option.** While an assembly is open, click **Tools, Options**. On the **System Options** tab, under **Performance**, select the **Automatically load parts lightweight** check box. In earlier versions, you had to close all assemblies before toggling this check box.
- ❑ **Open as lightweight.** There is a check box on the **Open** dialog box allowing you to open assemblies as lightweight. When you select the check box and open the assembly, all parts are loaded lightweight. The default status of the check box depends on the **Automatically load parts lightweight** check box.

NOTE: The same exceptions that existed in SolidWorks 2000 for opening parts lightweight still exist. For example, a component that is in the feature scope of an assembly feature cannot be loaded lightweight.

- ❑ **Re-lightweight.** In SolidWorks 2000, you had to close and re-open an assembly to set a resolved component back to lightweight. In SolidWorks 2001, you right-click the component and select **Set to Lightweight**. To set all the components of the assembly back to lightweight, right-click the top-level assembly icon and select **Set Resolved to Lightweight**. These options are available for all of a sub-assembly's components when you right-click the sub-assembly.
- ❑ **Suppress lightweight components.** You can right-click a lightweight component in the FeatureManager design tree or the graphics area and select **Suppress**. In earlier versions, you had to resolve the component first, then suppress the component.
- ❑ **Right-click lightweight components.** Right-clicking a lightweight component no longer resolves the component. Now it brings you to a shortcut menu where you can select **Set to Resolved**, if you want.

Assembly Dimensions in Custom Properties

You can add dimensions for mates, assembly features, and assembly sketches to an assembly's custom properties. For more information on adding dimensions to custom properties, see **Dimensions in Custom Properties** on page 3-17.

Mate Reference

The **Mate Reference** functionality is now located in the PropertyManager.

Additionally, you can select a primary, secondary, and tertiary reference entity. When you insert a part with a **Mate Reference** into an assembly, the SolidWorks software first tries to use the primary entity when looking for potential mate partners. If the primary entity is not valid for the entity your pointer is over, then the secondary entity is used. If neither the primary nor secondary entities are valid, then the tertiary entity is used.

Flexible Sub-assemblies


In SolidWorks 2000 and earlier, you could only move a sub-assembly as a single, rigid component in the parent assembly. In SolidWorks 2001, you can make sub-assemblies flexible. This allows movement of the individual components of a sub-assembly in the parent assembly.

An example of the benefit of this function is when you want to move the components of a piston sub-assembly in a motor assembly. You can move the individual components of the piston while still grouping the components as a sub-assembly.

Please note the following regarding flexible sub-assemblies:

- ❑ **Mates.** The mates in a flexible sub-assembly are solved simultaneously with the mates of the parent assembly. Therefore, moving or rotating a sub-assembly component does not violate the mates in the sub-assembly or its parent. A component moves only within its degrees of freedom according to its mates.
- ❑ **Configurations.** Each flexible sub-assembly must have its own unique configuration. This is because the position of a component in one flexible sub-assembly could be in a different position in another configuration of the flexible sub-assembly.



To make a sub-assembly flexible:

- 1 Open `\viselwise.sldasm`.
- 2 Right-click the rest handle sub-assembly and select **Component Properties**.
- 3 Under **Solve as**, select **Flexible**, then click **OK**.
- 4 Using **Rotate Component** , move one of the knobs of the rest handle.

Notice that in addition to the movement of the entire rest handle sub-assembly, the knob moves independently of the rest handle sub-assembly.


Assembly Configurations

You can create different configurations that toggle the visibility of selected components. In earlier versions of SolidWorks, you could toggle the suppression state of a component in a configuration, but not the visibility.

You can create different configurations placing assembly components in different locations using the **Move Component**  or **Rotate Component**  tools. On the **Move Component** and **Rotate Component** PropertyManagers, under **Options**, select the **This configuration** check box to apply the component movement to the current configuration only.


Move and Rotate Components

The user interface for moving and rotating components is new. In addition, some new functionality has been added, including the ability to move or rotate components in specified directions by dragging.

Using the **Move Component**  tool, here are the available options:

- Free Drag** - Select a component and drag in any direction. This function was available in SolidWorks 2000.
- Along Assembly XYZ** - Select a component and drag in the X, Y, or Z direction of the assembly. The coordinate system appears in the graphics area to help orient you. To select the axis you want to drag along, click near that axis before dragging.
- Along Entity** - Select an entity, then select a component to drag along that entity. If the entity is a line, an edge, or an axis, the component you move has one degree of freedom. If the entity is a plane or a planar face, the component you move has two degrees of freedom.
- By Delta XYZ** - Select a component, type an X, Y, or Z value in the **Move Component** PropertyManager, and click **Apply**. The component moves by the amount you specify. This function was available in SolidWorks 2000.
- To XYZ Position** - Select a point of a component, type an X, Y, or Z coordinate in the **Move Component** PropertyManager, and click **Apply**. The component's point moves to the coordinate you specify. If you select something other than a vertex or point, the component's origin is placed at the coordinate you specify.

Using the **Rotate Component**  tool, here are the available options:





- Free Drag** - Select a component and drag in any direction. This function was available in SolidWorks 2000. However, in SolidWorks 2001, you can now rotate a component that has mates.
- About Entity** - Select a line, an edge, or an axis, then drag a component around the selected entity. This function was available in SolidWorks 2000 using the **Rotate Component Around Axis**  tool; this tool has been removed.
- By Delta XYZ** - Select a component, type an X, Y, or Z value in the **Rotate Component** PropertyManager, and click **Apply**. The component moves around the assembly's axes by the angular value you specify.

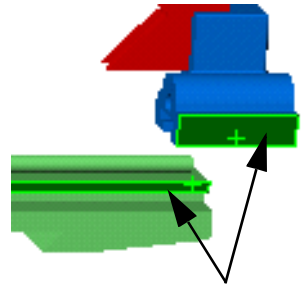
Mates

Mates in the PropertyManager


The **Assembly Mating** dialog box includes the same options, but is now in the PropertyManager.

To add a mate:

- 1 Open `\viselwise.sldasm`.
- 2 Click **Mate**  or **Insert, Mate**.
The **Mate** PropertyManager appears.
- 3 Select the two faces shown.
Note that the selections appear in the **Selections**  box automatically. Also note that only the valid mate types for two planar faces are shown.
- 4 Click **Distance** , and set the distance to 7in.
- 5 Click **OK**  to set the distance between the components.
- 6 Save the assembly as you will use this for other examples.



Other items of note in the *Mate PropertyManager*:

- Click **Keep Visible**  to keep the **Mate** PropertyManager open while you add multiple mates.
- You can control the alignment of the mate with the **Mate Alignment** options.
- You can delay the mate with the **Defer mate** check box.

Switch Mate Types

You can switch the mate type when you edit the definition of a mate. In earlier versions, it was necessary to delete the old mate, then add a new mate to switch the mate type. Only the valid mate types for the selections are shown.

To switch mate types:

- 1 Open `\viselwise.sldasm`.
- 2 If you have not completed the **Mates in the PropertyManager** section, you must do so first before proceeding.
- 3 Expand **MateGroup1** in the FeatureManager design tree to view all the mates.

- 4 Right-click the last **Distance** mate between **base<1>** and **support<1>** in **MateGroup1**, and select **Edit Definition**.

The **Mate** PropertyManager appears with the default values of the **Distance** mate.

- 5 Click **Parallel** , then click **OK** .

The mate type changes from **Distance** to **Parallel** as seen in the FeatureManager design tree.

- 6 Save the assembly as you will use this for other examples.


Animation of Mates

When previewing or applying a mate, you can view an animation showing the mated components moving into their final position. In some cases, this helps you visualize the effects of your new mate.

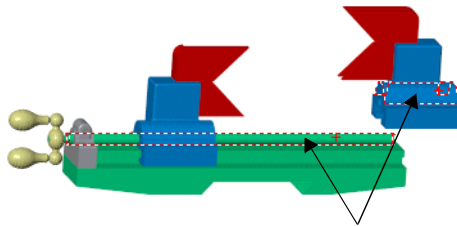
If you decide to view the mate animation, you can control the speed of the animation.

To view an animation of mates:

- 1 Open **\visel\wise.sldasm**.
- 2 If you have not completed the **Switch Mate Types** section, you must do so first before proceeding.
- 3 Click **Tools, Options**.
- 4 On the **System Options** tab, click **Performance** and move the **Mate animation speed** slider to the middle between **Fast** and **Slow**, then click **OK**.
- 5 Add a concentric mate between the two faces shown.

Note that when you click **Preview** or **OK** , you see an animation of the jaw and bracket moving into place.

- 6 Save the assembly as you will use this for other examples.



Symmetry Mates

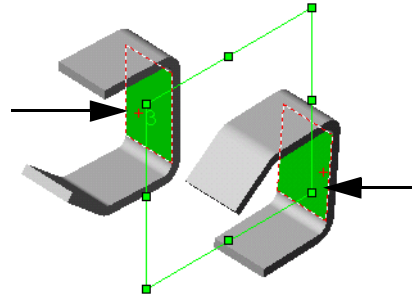
A symmetry mate forces two similar entities to be symmetric about a plane or planar face of a component. The following entities are allowed in symmetry mates:

- Points such as vertices or sketch points
- Lines such as edges, axes, or sketch lines
- Planes or planar faces
- Spheres of equal radii
- Cylinders of equal radii



The following items are not supported for symmetry mates:

- ❑ Symmetry mates do not mirror the entire component about the plane of symmetry.
- ❑ Symmetry mates only relate the selected entities to one another.

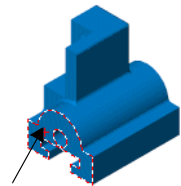
In the illustration at right, the two highlighted faces are symmetric about the highlighted plane. Notice the two components are upside down with respect to one another. That is because the highlighted faces only are symmetric, not all of the faces of both components.






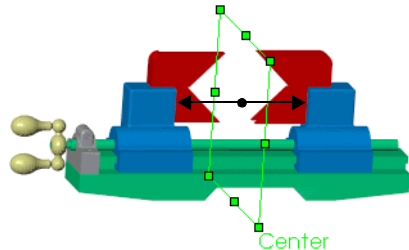
To add a symmetry mate:

- 1 Open `\visel\wise.sldasm`.
- 2 If you have not completed the **Animation of Mates** section, you must do so first before proceeding.
- 3 Click **Mate**  or **Insert, Mate**.
- 4 Click the FeatureManager tab  then select the **Center** plane from the flyout FeatureManager design tree. This is the plane of symmetry.

TIP: When creating a symmetry mate between two planes or planar faces, you must select the plane of symmetry first.



- 5 Select the face shown on each of the supports in the graphics area as the symmetric items.
- 6 Click **OK**  as **Symmetric**  is already selected.
- 7 Move one of the supports using **Move Component**  to see that the two supports maintain the same distance from the plane of symmetry.
- 8 Save the assembly as you will use this for other examples.



Enhanced Mate Selections

SolidWorks 2001 includes additional selections for several mates. The new combinations include:

Mate Types	Selections	Mate Types	Selections
Angle	Line and Cylinder	Parallel and Perpendicular	Cylinder and Line
Angle	Line and Extrusion	Parallel and Perpendicular	Cylinder and Cylinder
Angle	Cylinder and Cylinder	Parallel and Perpendicular	Cylinder and Extrusion
Angle	Cylinder and Extrusion	Parallel and Perpendicular	Line and Extrusion
Angle	Extrusion and Extrusion	Parallel and Perpendicular	Extrusion and Extrusion
Coincident	Point and Sphere	Tangent	Cylinder and Extrusion
Coincident	Point and Extrusion	Tangent	Line and Sphere
Distance	Line and Cylinder	Tangent	Plane and Extrusion
Distance	Point and Cylinder		
Distance	Plane and Cylinder		
Distance	Cylinder and Cylinder		

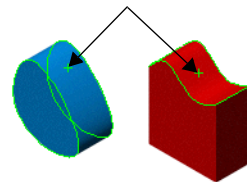
Here is a definition of the mate selections above:

- Cylinder** refers to the selection of a cylindrical face. For calculations, cylinder refers to the cylindrical face for Tangent mates and the cylinder's axis for Angle, Parallel, Perpendicular, and Distance mates.

- Extrusion** refers to the selection of a face that can be generated from an extruded profile. See the illustration at right for examples.

For calculations, extrusion refers to a single face of an extruded solid or surface feature for Coincident and Tangent mates. Extrusion refers to the direction of extrusion for Angle, Parallel, and Perpendicular mates.

Extrusions with draft are not allowed.





- Line** refers to the selection of a sketch line, an axis, or a linear edge.
- Plane** refers to the selection of a plane or a planar face.
- Point** refers to the selection of a sketch point, a vertex, or the origin.
- Sphere** refers to a spherical face.

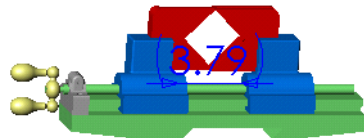
Minimum Distance in Dynamic Clearance


In addition to the Dynamic Clearance functionality from earlier versions, you can prevent two components from moving or rotating within a specified distance of one another. Dynamic Clearance previously only reported the minimum distance between components, but did not restrict movement.

To prevent components from moving within a specified distance:

- 1 Open `\wise\wise.sldasm`.
- 2 Click **Move Component** , select the **Dynamic Clearance** check box, and do the following:
 - Select the two supports for the **Check clearance between** box.
 - Click **Stop at Collision**  and type 0.5in.
 - Click **Resume Drag**.
- 3 Drag one of the supports.
- 4 Attempt to drag the components, eventually trying to move them closer than 0.5in from each other.

A dimension appears in the graphics area indicating the minimum distance between the selected components.



Because you clicked **Stop at Collision** , the components cannot move closer than the specified distance of .5”.

- 5 Save the assembly as you will use this for other examples.

Assembly Features


Hole Wizard Holes as Assembly Features

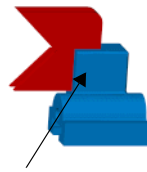
You can add hole wizard holes as assembly features which can go through more than one component. All of the hole wizard hole types are available such as countersink, counterbore, and so on.

Also new is the ability to display cosmetic threads on assembly feature holes. The cosmetic thread appears only on the last hole that the assembly feature creates.

For more information about hole wizard holes, see **Hole Wizard on a Plane** on page 3-16.

To add a hole wizard hole:

- 1 Open `\viselwise.sldasm`.
- 2 If you have not completed the **Minimum Distance in Dynamic Clearance** section, you must do so first before proceeding.
- 3 Zoom in on the support and bracket on the right-hand side of the vise.
- 4 Select the flat face of the support as shown.
- 5 Click **Hole Wizard**  or **Insert, Assembly Feature, Hole, Wizard**.



The **Hole Definition** dialog box appears.

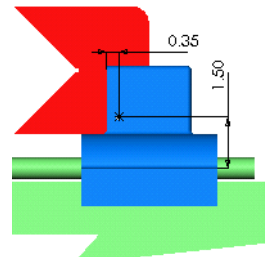
- 6 On the **Counterbore** tab, set the following properties:
 - **Standard** to **Ansi Inch**
 - **Size** to **#5**
 - **End Condition** to **Through All**.
- 7 Click **Next**.

The **Hole Placement** dialog box appears.

- 8 Dimension the point as shown and click **Finish**.

The counterbored hole goes through both the support and the jaw.


- 9 Save the assembly as you will use this for other examples.






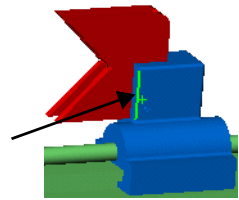
Component Pattern Using Assembly Feature Pattern


You can derive a component pattern from an assembly feature pattern. In earlier releases, you could derive a component pattern only from a feature pattern in the part.

To derive a component pattern from an assembly feature pattern:

- 1 Open `\viselwise.sldasm`.
- 2 If you have not completed the **Hole Wizard Holes as Assembly Features** section, you must do so first before proceeding.
- 3 Click **Insert, Assembly Feature, Linear Pattern**.
The **Linear Pattern** PropertyManager appears.
- 4 Select the Hole Wizard hole in the graphics area.
The feature automatically appears under **Features to Pattern** .

- 5 Under **Direction 1**, do the following:
 - Click the vertical edge shown as the **Pattern Direction**.
 - Set the **Spacing**  to 1in and the **Number of Instances**  to 2.
 - Click **Reverse Direction**  if necessary to create the second instance above the first.

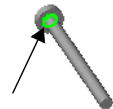


- 6 Click **OK**  to create the pattern.

- 7 Open `\viselbolt.sldprt`.

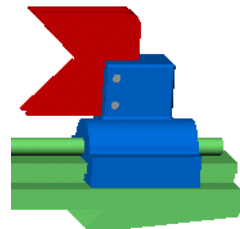
- 8 Create a SmartMate between the highlighted edge shown in the bolt and the original hole.

This creates a **Concentric** and **Coincident** mate between the bolt and the hole.



- 9 Click **Insert, Component Pattern** and do the following:
 - a) Leave the default as **Use an existing feature pattern (Derived)**, then click **Next**.
 - b) Select the bolt as the **Seed Component(s)**.
 - c) Select **LPattern1**, the assembly feature pattern, as the **Pattern Feature**.
 - d) Click **Finish**.

There is now a second instance of the bolt in the second hole created by the assembly feature pattern.



Drawings and Detailing

You can create 2D drawings from the 3D solid parts and assemblies you design. This chapter describes the enhancements to drawings in the following areas:

- General
- Visio for Schematics
- Drawing Views
- RapidDraft Drawings
- Detailing
- Dimensions
- Annotations

General

PropertyManager

The PropertyManager is now activated when you either create or select a drawing view.

Balloons and Stacked Balloons are also now created and edited in the PropertyManager. You can select a single balloon or a group of balloons for editing.

Document Properties

Three new document properties have been added: **SW-Folder Name**, **SW-Short Date**, and **SW-Long Date**. These properties can be used as standard notes in drawing sheets and sheet formats. See Linking Notes to Document Properties in *SolidWorks 2001 Online User's Guide* for details on using document properties in drawing documents.

- **SW-Folder Name** is the document's folder. For part `c:\docs\324-123\part1.sldprt`, the folder name is `c:\docs\324-123\`. The folder name has a backslash at the end of name, which is useful when adding the file name to the folder name. In case of a remotely loaded document, the folder may be in the form `\\some_machine\share\docs\324-123\`. For documents that have not yet been saved, the folder property name is empty.
- **SW-Short Date** and **SW-Long Date** are current date values. The dates are language and region dependent. For details see **Start/Settings/Control Panel/Regional Settings/Date** on your computer.

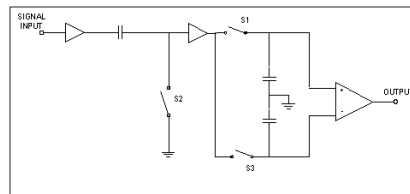
Selecting Silhouettes

You can now select silhouettes by box-select in drawings.

Visio for Schematics

Microsoft Visio® Technical Edition is now tightly tied into the SolidWorks software. Visio is best in its class for electrical circuit diagrams, piping and pneumatic diagrams, and schematics. Features of the Visio integration with SolidWorks include:

- You must have Visio on your system for the **Insert, Schematic** menu item to be available for SolidWorks drawings.
- The integration uses OLE technology.
- You can specify the size of the schematic before you choose a template.
- SolidWorks offers you a choice of common templates. You can also choose from the Visio list of templates.




Visio schematic in SolidWorks drawing


For further information about Visio, see the Microsoft Visio documentation.


Drawing Views


Create Views without Preselection

Section, Detail, Auxiliary, and Projected views can be created with fewer steps.

For a Section View, activate the view, click **Section View** , sketch a line (the **Line** tool is automatically active), and click in the graphics area to place the section view. To create a multi-line section view, or to use a centerline as the section line, sketch the section line before clicking the Section View tool.

For a Detail View, activate the view, click **Detail View** , sketch a circle (the **Circle** tool is automatically active), and click in the graphics area to place the detail view. If you want a profile other than a circle, sketch the profile before clicking the Detail View tool.


For an Auxiliary View, click **Auxiliary View** . You are prompted to select a reference edge. When you select an edge, you are prompted to select a location for the new view, aligned with the reference edge. Click in the graphics area to place the view.


For a Projected View, click **Projected View** . You are prompted to select a drawing view from which to project. Select a view, move the pointer away from the view, and click in the graphics area to place the projected view. You can create as many projected views as required without selecting the tool again. When you are finished, press **Esc**.

Section Views of Section Views

You can create Section Views of Section Views. A new section is calculated from the original solid model, and the view updates if the model changes.


To create section views of a section view:

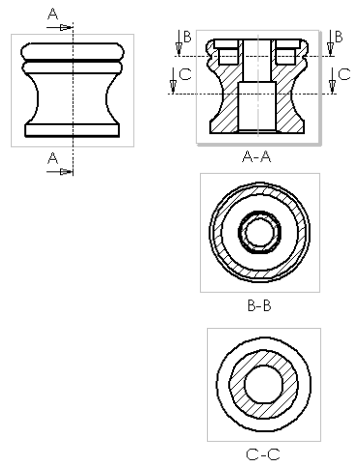
- 1 Open `\\handle\connector_section.sliddrw`.
- 2 Click **Section View**  or **Insert, Drawing View, Section**.

The **Line**  tool is activated automatically.

- 3 Sketch a line through the upper portion of the Section View as shown for line B-B and click in the graphics area to place the new section view.

A Section View aligns along an axis perpendicular to the section line in the parent view.

- 4 Click **Section View**  again.
- 5 Sketch a line through the middle of the first Section View as shown for line C-C and click in the graphics area to place the second new section view.



Section Views of a Section View

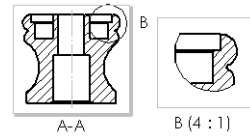
Detail Circle Styles

More styles have been added for detail circles. If you display the detail **Circle**, the new styles available are:

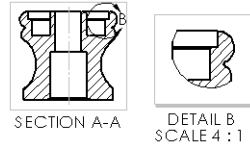
- **Per Standard**
- **Connected**

Per Standard replaces **Default**. **Per Standard** changes with the standard (ISO, ANSI, and so on) as **Default** did in previous releases.

The **Beside Profile** style has been removed. For this style, use **No Leader**.

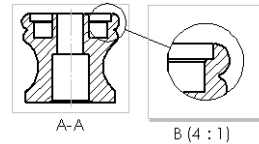


ISO standard detail circle



ANSI standard detail circle

The new **Connected** style displays the detail view with the profile. The **Connected** style can be selected only if the detail view is on the same sheet as its parent view.

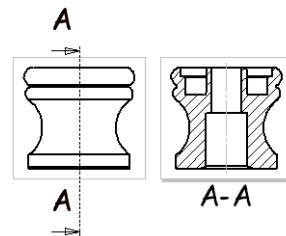


Connected style detail circle

Section and Detail View Label Font

In Section and Detail views, you can specify the font for the label on the section line or detail circle. Now, if you change the font for the section line or detail circle label, you can apply the new font to the **Section View** or the **Detail View** label as well. (Changes in label *text* are automatically applied to the Section or Detail view label.)

If you change the label text or font for the Section View or Detail View, however, the change is not applied to the parent view label.



Section line label font change (left) also applied to section view label (right)

Section and Detail Views Expandable

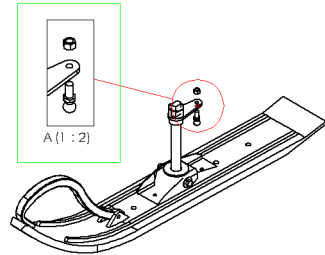
Section Views and Detail Views can now be expanded in the FeatureManager design tree so that all components and features are available.

Section View Hidden Edges

You can now show hidden edges in Section Views in Hidden Lines Removed mode. Right-click the component or feature in the graphics area or the FeatureManager design tree and select **Show/Hide, Show Hidden Edges** or **Show/Hide, Hide Hidden Edges**.

Views of Exploded Views

You can now create Detail, Auxiliary, and Projected views of exploded views in drawings.



Detail View of Exploded View

View Orientation Selection in Named Views

After a Named View has been placed in a drawing, you can change the view orientation by double-clicking an orientation in the **View Orientation** section of the **Named View** PropertyManager.

Alternate Position Views

Alternate Position Views allow you to superimpose one drawing view precisely on another. Alternate Position Views are often used to show the range of motion of an assembly.



The Alternate Position View is shown with phantom lines. You can dimension between the primary view and the Alternate Position View.

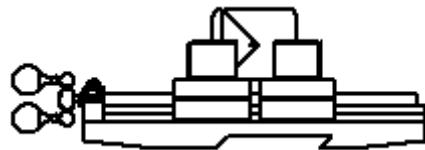
You can create more than one Alternate Position View in a drawing.

The Alternate Position View is not available in the following types of views:

- Broken View
- Detail View
- Cropped View

To add an alternate position view:

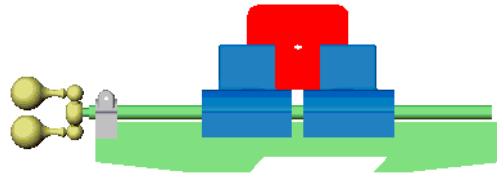
- 1 Open `\vise\alternate_position.sldasm`.
- 2 Open `\vise\vise.slddrw`.
The drawing document contains a Named View in View Orientation **view2**.
- 3 Select the Named View.
- 4 Click **Alternate Position View**  or **Insert, Drawing View, Alternate Position View**.
- 5 Click **OK**  to accept the new configuration name.



Drawing of vise in closed position

The **Alternate Position View** PropertyManager opens. Under **Configuration**, a proposed configuration name is shown in **New configuration**.

The assembly window is opened automatically with the **Move Component** command activated.




Vise in closed position

- 6 Drag the vise jaws apart. Because the supports and jaws are mated, moving one component moves both supports and jaws, in opposite directions. To show the range of motion, drag the components apart until the support on the left is close to the bracket.

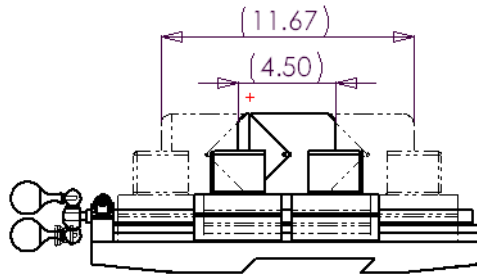


Vise in open position

- 7 Click **OK**  to close the **Move Component** PropertyManager and return to the drawing.

The components that have moved are shown in phantom lines in the drawing.

- 8 Dimension the jaws in both the closed and open position for comparison.



Alternate Position View in dashed lines

Component Line Font

The **Component Line Font** dialog box has been redesigned.

- **Use document defaults** check box has been added.
- **Reset** button has been added. This button allows you to reset the **Line style** and **Line weight** selections to the defaults you specified in **Tools, Options, Document Properties, Line Font**.

Break and Unbreak Commands with Multiple Views

The **Break** and **Unbreak** commands create and remove Broken Views. Now you can selected multiple views, and the commands will be applied to all the appropriate selected views. These commands previously applied to only the last selected view when multiple views were selected.

RapidDraft Drawings

When you open a RapidDraft drawing, a **Load Model** check box is now included in the **Open** dialog box. When the check box is selected, models are loaded automatically.

When the model is not loaded, new functionality in RapidDraft drawings includes:

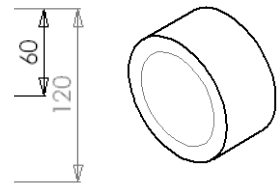
- Selecting faces
- Viewing temporary axes
- 3D highlighting
- Referencing and dimensioning cosmetic threads
- Significant improvement in update performance
- True reference dimensions

Detailing

Hide/Show Annotations

Hide/Show Dimensions is now **Hide/Show Annotations** on the **View** menu and in the tooltip on the Drawing toolbar. Hidden annotations are shown in gray when the tool is active. Otherwise, they are not visible.

In addition to dimensions, you can hide and show Cosmetic Threads with **Hide/Show Annotations**. You cannot hide other types of annotations (Notes, Geometric Tolerance Symbols, and so on).



Hide/Show Annotations displaying hidden dimension and cosmetic thread in gray

Bill of Materials

Changes in bills of materials include:


- SolidWorks remembers settings in the **Bill of Material Properties** dialog box from session to session.
- **Contents** and **Control** tabs are now available in the dialog box when you create a bill of materials. Previously they were available only when editing.
- On the **Configuration** tab, the wording for the anchor corner has been changed. **Make corner X coincident to anchor point** is now **Anchor point coincident to X**, where **X** is the list of corners (**Upper left**, **Upper right**, and so on).
- You can now embed dimensions and mass property parameters into values of Custom Properties. See **Dimensions and Mass Property Parameters** on page 3-17 for a description of the embedding procedure. These Custom Properties can be used in notes (see **Custom Properties in Notes** on page 5-10) and bills of materials, as described in the *SolidWorks 2001 Online User's Guide*. The entry in the bill of materials updates when the part dimension or mass property changes.

- A new set of templates for bills of materials has been added. The default template, **Bomtemp.xls**, contains columns for **ITEM NO.**, **QTY.**, **PART NO.**, and **DESCRIPTION**. The new templates add the following columns to the default format:

BOM template file name	Added columns
Bomtemp-Material.xls	MATERIAL
Bomtemp-StockSize.xls	STOCK SIZE
Bomtemp-Vendor.xls	VENDOR NO
Bomtemp-Weight.xls	WEIGHT
Bomtemp-All.xls	VENDOR NO, WEIGHT, STOCK SIZE, MATERIAL

You have to add the custom properties into your part in order to take advantage of these new templates. See the *SolidWorks 2001 Online User's Guide* for information on adding custom properties to a bill of materials.

To add a custom property in one of the new bill of materials templates:

- 1 Open `\handle\connector_custom.sldprt`.
- 2 Click **File, Properties**. In the **Summary Information** dialog box, on the **Custom** tab, note that **material** and **description** have been defined in the **Properties** box. Click **OK**.
- 3 Open `\handle\connector_custom.slddrw`.
- 4 Select the view of the connector and click **Insert, Bill of Materials**.
- 5 In the **Select BOM Template** dialog box, select **bomtemp-material.xls** and click **Open**.
- 6 In the **Bill of Materials Properties** dialog box, on the **Configuration** tab, under **Anchor point**, **Use table anchor point** should be selected, with **Anchor point coincident to: Top Left**.
- 7 Click **OK**  .

The bill of materials appears at the top left corner of the drawing sheet with the custom properties automatically inserted into the appropriate columns as shown.

ITEM NO.	QTY.	PART NO.	MATERIAL	DESCRIPTION
1	1	Connector_custom	brass	connector

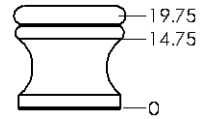
Dimensions

Dimensions can now be modified, moved, and deleted while the **Dimension** tool is active. Previously you had to click the **Select** tool to access these functions.

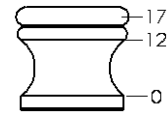
Ordinate Dimensions

Ordinate dimensions now have the following features:

- You can attach ordinate dimensions to the centers and minimum and maximum points of arcs.
- You can drag the zero ordinate to a new position, and all the ordinate dimensions update to match the new zero position.
- Ordinate dimensions are now available in part sketches. See **Ordinate Dimensions** on page 2-7.



Ordinate dimension to arc



Ordinate zero moved

Dragging Dimension Witness Lines

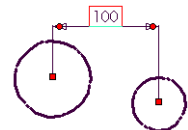
You can drag linear and ordinate circle and arc dimensions to the new positions using their witness line attachment points.

Dragging the end point perpendicular to the witness line, you can move the line to the minimum, the center, or the maximum of the arc or circle.

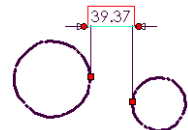
To change dimension positions by witness lines:

- 1 Open a new drawing.
- 2 Draw two circles and create a linear dimension between the centers as shown.
- 3 Drag one of the witness lines to the edge of the circle and release it. The witness line attaches to the edge of the circle.
- 4 Drag the other witness line to the edge of its circle.

You can modify the dimension as required.



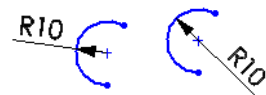
Linear dimension between circles



Witness lines dragged to new positions

Radial Dimensions

Radial dimensions change automatically from outside to inside an arc as you drag the dimension. Previously you had to select or clear the **Dimension to inside of arc** check box in the **Dimension Properties** dialog box.



Automatic radial dimension orientation

Annotations

Multiple Annotations



New features in handling multiple annotations include:

- You can select any combination of annotations, dimensions, drawings views, groups, and sketched entities. You can select items individually or by dragging a selection box around multiple items.
- You can cut and paste or drag multiple items in drawings. This action does not create a group, so any individual item can be moved, deleted, and so on at the new location.
- When multiple items are pasted, the paste point is at the center of all the selected items. All items retain their relative positions. You can cut or copy items on one drawing sheet and paste them on another sheet.
- You can cut and paste annotations that belong to different views. When the annotations are pasted, they are all attached to the active view.

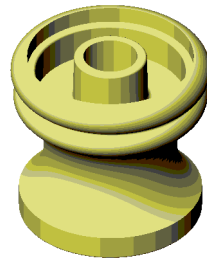
Custom Properties in Notes

You can add dimensions and mass properties to notes. The note can be in a part, assembly, or drawing document. The numeric values in the notes are updated if the properties change in the model. See **Dimensions and Mass Property Parameters** on page 3-17 for details on defining the custom properties.

To add custom properties to a note:

- 1 Open `\handle\connector_custom.sldprt`.
- 2 Click **Note**  or **Insert, Annotations, Note**.
- 3 Place the cursor in the **Note text** box and click **Link to Property** .
- 4 In the **Link to Property** dialog box, under **Property Name**, select **description** from the list, and click **OK**.
The variable name of the property, **\$PRP:"description"**, appears in the **Note text** box.
The actual value of the property, **connector**, will appear in the note.
- 5 Place the cursor at the end of the text in the **Note text** box and press **Enter**. Type **Height:**, then repeat step 4, choosing the property **height**.
- 6 Place the cursor at the end of the second line of text and press **Enter**. Type **Volume:**, then repeat step 4, choosing the property **volume**.
- 7 With the dialog box still open, click in the graphics area to place the note.
- 8 Click **OK** to close the **Note Properties** dialog box.

connector
Height: 22 mm
Volume: 4686.11 mm³



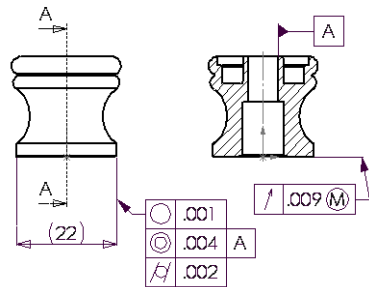
Note with custom properties

Geometric Tolerance Symbols

You can now create Geometric Tolerance symbols with more than two feature frames.

Automatic witness lines are created when you drag the leader of a Geometric Tolerance symbol off a model edge.

If you hold the **Alt** key down while dragging the Geometric Tolerance symbol, the symbol and leader move as a single item, maintaining their orientation. The automatic witness line adjusts to the position of the leader.




Geometric Tolerance Symbols with multiple feature frame and witness line

Cosmetic Threads

New features of cosmetic threads include:

- When a model is inserted into a drawing, cosmetic threads are inserted automatically. Only one copy of a cosmetic thread callout is imported to a sheet.

NOTE: The **Cosmetic Thread** check box has been removed from the **Insert Model Items** dialog box.

- When you move the pointer over a cosmetic thread, the pointer changes to .
- Notes now inference to cosmetic threads, both when placing leaders while creating notes and when dragging leaders to the cosmetic thread. The new cosmetic thread pointer is displayed when the leader infers the thread.
- You can now attach notes to the silhouette edges of cosmetic threads in drawings. Previously notes could be attached only to the circular faces of the threads.
- A cosmetic thread that is created in a part cannot be deleted from a drawing, but it can be hidden and shown. The shortcut menu for cosmetic threads now includes **Hide/Show**. Cosmetic threads can also be hidden and shown using the **Hide/Show Annotations** tool.
- The visibility of cosmetic threads follows the visibility of the owner feature. When you change display mode, add features to the **Show Hidden Edges** list, or hide components, the visibility of cosmetic threads changes automatically.
- You can dimension both the circular cosmetic thread and the linear dimension of the sides in drawings. You can dimension the silhouette edge of a cosmetic thread to a sketch entity.
- You can now reference patterned cosmetic threads.

- You can now display cosmetic threads on assembly feature holes. The cosmetic thread appears only on the last component cut by the hole.
- In drawings, **Insert Callout** has been added to the shortcut menu. If a cosmetic thread callout is defined in the part but is not displayed in the drawing, selecting this menu item displays the callout.

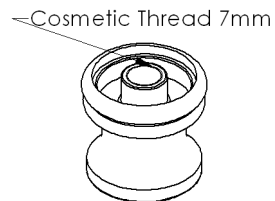
To display a cosmetic thread callout:

- 1 Open `\handle\connector_cosmetic_thread.sldprt`.
- 2 In the FeatureManager design tree, expand **Cut-Extrude3**.
- 3 Right-click **Cosmetic Thread1** and select **Edit Definition**.
- 4 In the **Cosmetic Thread** dialog box, type **Cosmetic Thread 7mm** in the **Thread callout** box and click **OK**.

NOTE: If the cosmetic thread is created in the Hole Wizard and you specify **Cosmetic thread with thread callout**, text is inserted into the box automatically.

- 5 Open `\handle\connection_cosmetic_thread.slddrw`.
- 6 Right-click the cosmetic thread in the center of the connector in the graphics area and select **Insert Callout**.

A leader attaches to the thread by default. The cosmetic thread callout is a note. You can edit the callout as you would edit any note.

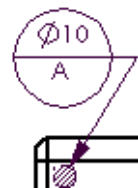


Cosmetic thread with cosmetic thread callout

Datum Target Symbols

Datum Target Symbol functionality has been improved as follows:

- Leader length is now correct over zoom changes.
- Initial leader length has returned to the shorter length found in SolidWorks 99.
- The symbols dynamically snap to faces when tracking in parts and assemblies, and to edges when tracking in drawings.



Datum Target Symbol


Balloons

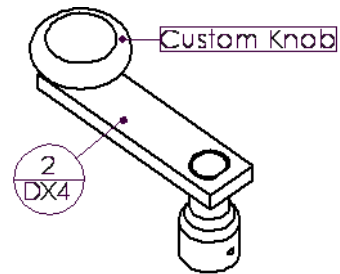
You can now set options for stacked balloons, and you can add custom text to balloons. Options have been added for both these features. In addition, options for note borders have been moved to **Tools, Options, Document Properties, Detailing, Notes**.

To set the new options for balloons, click **Tools, Options**. On the **Document Properties** tab, under **Detailing**, click **Balloons**.

- *Stacked Balloons*. You can select a **Style** and **Size** under **Stacked balloons**.
- *Custom text*. You can choose **Custom** for the text in any type of balloon and in either or both sections of a **Circular Split** style of balloon.

To add custom text to a balloon:


- 1 Open a new drawing.
- 2 Click **Balloon**  or **Insert, Annotations, Balloon**.
The **Balloon** PropertyManager appears.
- 3 Under **Balloon text**, select **Custom**.
- 4 Type text into the box that opens below.
- 5 Click in the graphics area to place the balloon.
- 6 In the PropertyManager:
 - Under **Style**, select **Circular Split Line**.
 - Under **Lower text**, choose **Custom**.
 - Type text into the text boxes.
- 7 Click in the graphics area to place the balloon.
- 8 To release the tool, press **Esc** or click the balloon tool.



Custom balloon text

Blocks

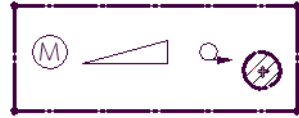
New features for blocks include:

- You can create a block in place, explode it, and edit it on screen. This capability allows you to make a block from imported DXF/DWG data.
- A block can include sketch entities, notes, other annotations, and area hatch. Stacked balloons are not allowed in a block.
- You can include annotations from different views.
- When the pointer is over a block, the pointer changes to .
- **Explode** and **Save to file** have been added to both the **Tools, Block** menu and the shortcut menu. On the **Tools, Block** menu, **New** has been replaced by **Make**.
- Saving a block to file for later use is now optional.
- You now have two methods for editing a block, as described below. Previously, editing was available only in the block editor.

- You can edit the text of a note in place by double-clicking the note, even when the note is part of a block.

To create a block:

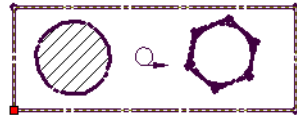
- 1 Open a new drawing.
- 2 Draw a circle. With the circle still selected, click **Insert, Area Hatch**.
- 3 Add a note. In the Note Properties dialog box, click **Add Symbol** and select the **Flattened Length** symbol. Place the note in the graphics area.
- 4 Add a balloon with a **Style** of **Triangle**, and with text **4T** in a **Custom** text box.
- 5 Draw a rectangle around the entities already created
- 6 Box-select all the entities and click **Tools, Block, Make**.



Block with sketch entities, note symbol, balloon, and area hatch

To edit a block on screen:

- 1 Select the block.
A red point at the lower left corner of the block shows that the block is selected.
- 2 Right-click anywhere in the block and select **Explode Block**.
- 3 Delete the balloon.
- 4 Sketch a polygon.
- 5 Box-select all the entities and click **Tools, Block, Make**.



Block after editing on screen

To edit a block in the block editor:

- 1 Select the block.
- 2 Right-click and select **Save to file**.
- 3 In the **Save As** dialog box, accept **block1.sldsym** as the file name and click **Save**.
Extension **.sldsym** is added automatically to block file names.
- 4 Click **Tools, Block, Edit**.
- 5 In the **Open** dialog box, select **block1.sldsym** and click **Open**.
- 6 In the block editor, right-click the block and select **Explode Block**.
- 7 Delete the circle, which also deletes the area hatch.
- 8 Right-click and select **Make Block**.
- 9 Click **Save** and close the editor.



Block after editing in block editor

Import and Export

This chapter describes the enhancements to Import and Export items, including:

- General Information
- ACIS[®] Files
- DXF/DWG Files
- IGES[®] Files
- Pro/ENGINEER[®] Files
- STEP Files

General Information

Import Diagnosis

There is a new **Improve Geometry** function added to the **Import Diagnosis** tool. This functionality performs geometry simplification, model tolerance reset, and redundant topology removal on the imported body.

For example, if you are unable to create a 2D sketch on an imported surface that appears to be planar, **Improve Geometry** simplifies the surface, allowing you to create the sketch.

Improve Geometry works if the imported spline surface geometry is within an acceptable tolerance.

NOTE: You *might* not be able to sufficiently rebuild all imported bodies with the **Improve Geometry** function.


To see an example of improving the geometry of an imported surface:

- 1 Click **File, Open**.

The **Open** dialog box appears.

- 2 Open the SolidWorks document, **assem1.sldasm**.

The assembly document contains features that were originally imported into SolidWorks.

- 3 Select the two edges shown in the example. Click **Mate** .

Notice that the selected edges do not appear under **Selections** in the **Mate** PropertyManager.

- 4 Click one of the edges.

A pop-up tooltip appears informing you that non-linear edges cannot be used for mating.


- 5 Close the **Mate** PropertyManager.

- 6 In the FeatureManager design tree, right-click **Part3<1>** and select **Open part3.sldprt**.



If a message box appears asking if you want to recognize features, click **No**.

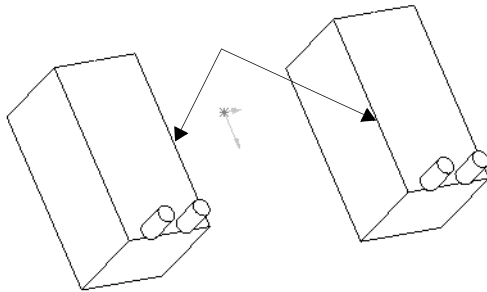
- 7 In the **Part3** FeatureManager design tree, right-click **Imported1**, and select **Import Diagnosis, Improve Geometry**.

The geometry of the feature is rebuilt, turning the non-linear edge into a linear edge.

- 8 Open the Assembly document again, click **Mate** , and select the two edges.

The two edges appear under **Selections** in the **Mate** PropertyManager.

- 9 Click **Coincident** , then click **OK**  to mate the two edges.



Reloading Imported Features

The reloading of imported features has been improved. You can now edit the definition of an imported feature and reload the file that was used to create the imported body. SolidWorks attempts to rebuild the subsequent features whenever possible.

In previous versions of SolidWorks, after performing the **Edit Definition** command, you had to recreate any subsequent features you had added to the imported body.

To see an example of this functionality:

- 1 Click **File, Open**.

The **Open** dialog box appears.

- 2 Set **Files of type** to **STEP AP203/214 (*.step,*.stp)**, and open **Edit_definition.step**.

Do not recognize the features if prompted.



- 3 Select the bottom face of the body.

- 4 Click **Chamfer** .

The **Chamfer** PropertyManager appears.

Face<1> appears in the **Edges and Faces or Vertex** box.

- 5 Under **Chamfer Parameters**, do the following:

- Make sure **Angle Distance** is selected.
- Set **Distance**  to 0.001m.
- Set **Angle**  to 10.00deg.

- 6 Click **OK** .

The part should look similar to the example.

- 7 Right-click **Imported1** in the FeatureManager design tree, and select **Edit Definition**.

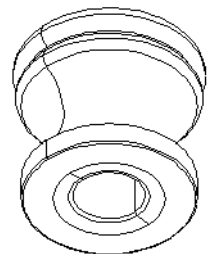
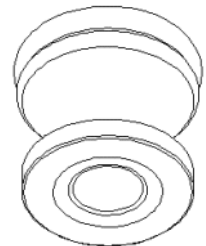
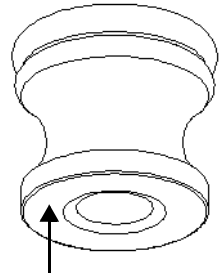
A message box appears warning that this feature has a parent/child relation or is being referenced.

- 8 Click **OK**.

The **Open** dialog box appears.

- 9 Select **Edit_definition.step** and click **Open**.

Do not recognize the features if prompted. The feature reloads and there are no feature errors.



ACIS Files

Body and Face Colors

SolidWorks now supports export and import of body and face colors to and from ACIS (.sat) files.

Importing Attributes

SolidWorks now supports import of generic named attributes (integer, position, real, string, and vector) associated with bodies and faces from ACIS (.sat) files. These attributes are displayed as features in the FeatureManager design tree. Their values are accessible only through the SolidWorks Application Programming Interface (API).

DXF/DWG Files

Import Wizard

There is a new DXF/DWG import wizard to assist you with importing DXF and DWG files into SolidWorks. You can do the following:

- Import these files to a new drawing or to a new part
- Set various input and output file properties
- Make scaling and positional changes to paper space drawings

To import a .dwg file into a new SolidWorks drawing document:

- 1 Click **File, Open**.

The **Open** dialog box appears.

- 2 Set **Files of type** to **Dwg Files (*.dwg)**, and open **DWG_Import_Sample.dwg**.

The **DXF/DWG Import - Document Type** dialog box appears.

- 3 Click **Import to a new drawing**, then click **Next**.

NOTE: You can also click **Finish** to accept all default settings and import the file immediately.

The **DXF/DWG Import - Document Settings** dialog box appears.

- 4 Select the **Show preview** check box.

A preview of the imported drawing file appears. The red frame indicates the drawing sheet size. The white information is the imported file geometry.

- 5 In the **Geometry positioning** box, click **Center in sheet**.

The drawing entities are centered on the sheet.

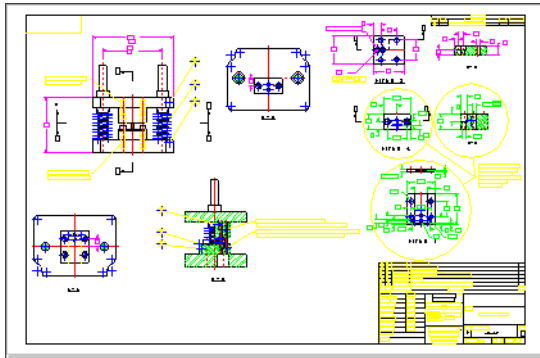
- 6 Set the **Data units**, **Paper size**, **Document template**, and **Geometry scaling** options as desired, then click **Next**.

The **DXF/DWG Import - Drawing Layer Mapping** dialog box appears.

- 7 Click **Import all data to the sheet**, then click **Finish**.

If a message appears about dimension values and actual sizes, click **OK**.

The SolidWorks drawing document appears with the imported geometry. All original colors from the **.dwg** file are maintained in the SolidWorks drawing document.



Converting Imported Dimensions from DXF/DWG Files

When you import DXF/DWG files, dimensions are no longer automatically converted to the SolidWorks defined dimension style. The dimensions retain their original style and properties.

However, if you drag a dimension, it is now changed to the SolidWorks style. You can then modify its properties as well.

To see how dimension conversion works:

- 1 Click **File**, **Open**.

The **Open** dialog box appears.

- 2 Set **Files of type** to **Dxf files (*.dxf)** and open **DXF_dimensions.dxf**.

The **DXF/DWG Import - Document Type** dialog box appears.

- 3 Click **Import to a new drawing**, then click **Next**.

The **DXF/DWG Import - Document Settings** dialog box appears.

- 4 Do the following:

- Select the **Show preview** check box.
- Set **Paper size** to **C - Landscape**.
- Click **Center in sheet**.

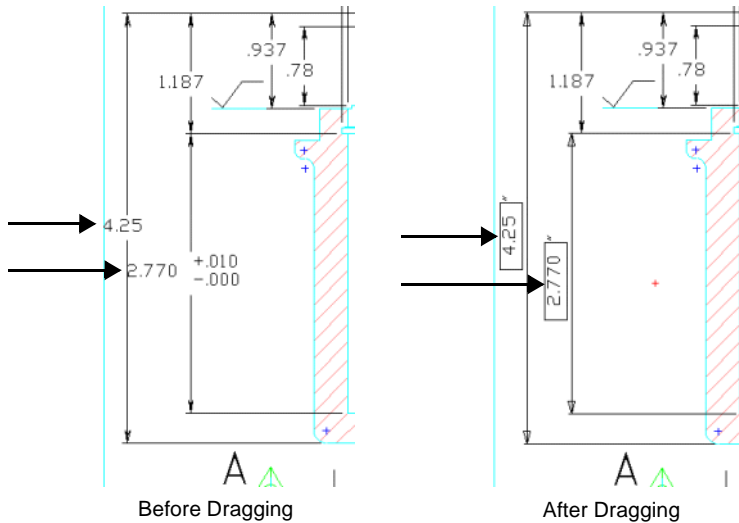
Chapter 6 Import and Export

5 Click **Finish** to open the DXF file.

If a message appears about dimension values and actual sizes, click **OK**.

6 Zoom to the lower left corner of the drawing.

7 Drag the dimensions **4.25** and **2.770**.



The dimensions are converted from the imported dimension style, to the SolidWorks dimension style. Note the change of arrowhead style.

Importing and Exporting of OLE Objects

SolidWorks now supports importing and exporting of OLE objects through the DXF/DWG interface. When you import or export a DXF/DWG file that has an OLE object, the OLE object is now visible in the application. This includes, for example, design tables in drawings, bitmaps of your company logo, or Microsoft Word documents.

IGES Files

Conversion Options

You can now import 3D curves in an IGES file as 3D sketch entities.

To see this new option in the dialog box:

- 1 Click **File, Open**.

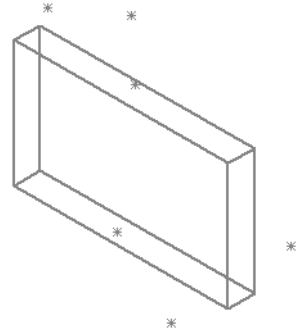
Set **Files of type** to **IGES Files (*.igs,*.iges)** and open **IGES_3d_new.IGS**.

- 2 The **IGES - In Surface, Curves, and Levels** dialog box appears, containing two options:

- **Import IGES curves as 3D curves**
- **Import IGES curves as 3D sketches**

The **Import IGES curves as 3D sketches** is a new option.

- 3 Under **Curves**, select **Import IGES curves as 3D sketches**, then click **OK**.
- 4 If a message box appears with a report on gaps or bad faces, click **OK**.
The part opens.



Pro/ENGINEER Files

You can now import Pro/ENGINEER surface-trim and surface-extend features into SolidWorks. These features are read in from the Pro/ENGINEER file and mapped to SolidWorks.

To import a surface trim feature:

- 1 Click **Tools, Add-Ins**.
The **Add-Ins** dialog box appears.
- 2 Select **SldTrans 1.0**, then click **OK**.
- 3 Click **File, Open**.

- 4 Set **Files of type** to **Pro/ENGINEER Part File (*.prt.*)** and open **ProE_surfacetrim.prt.2**.

The **Pro/ENGINEER to SolidWorks Converter** dialog box appears.

- 5 Select **Prompt me later after analysis**, and click **Import**.

The part opens. Note the feature, **Surface-Trim1**, in the Feature Manager design tree.



STEP Files

Boundary Representation (BREP) Information

You can now import STEP files using the boundary representation (BREP) information for import. This speeds up the import process.

To import a STEP file using BREP information:

- 1 Click **File, Open**, and set **Files of type** to **STEP AP203/214 (*.step,*.stp)**.
- 2 Click **Options**.

The **Import Options** dialog box appears.

- 3 Select **BREP** as the **Import Mode**, then click **OK**.
- 4 Browse to any STEP file and click **Open**.

The file opens using BREP information, if available in the file.

Colors

Body and face colors are now supported for import and export of STEP AP214 files. The STEP AP203 standard does not have any color implementation.

Units

SolidWorks now supports exporting the unit of length of a part or assembly document to a STEP AP203 or AP214 file. In earlier versions of the SolidWorks software, only meters was supported as a unit of length during export to a STEP file.

Sheet Metal

This chapter describes the enhancements to the SolidWorks sheet metal functionality including:

- ❑ New sheet metal features such as:
 - Base Flange
 - Edge Flange
 - Miter Flange
 - Closed Corner
 - Tab
 - Sketched Bend
 - Unfold/Fold
 - Flat Pattern
- ❑ Improved sheet metal features such as:
 - Mirror all
 - Bend reliefs
 - Flat pattern drawings
 - Bend tables, including bend deduction values
- ❑ A comparison of the SolidWorks 2000 and SolidWorks 2001 sheet metal features
- ❑ A step-by-step example combining many of the new and improved sheet metal features




New Sheet Metal Features

In SolidWorks 2001, many new sheet metal-specific features have been added. The advantage to these features is that they allow you to design sheet metal parts directly. This is contrary to the SolidWorks 2000 method of designing a solid body and later converting it to a sheet metal part. For more information, see **Comparison of Sheet Metal in SolidWorks 2000 and SolidWorks 2001** on page 7-17.

Base-Flange

A Base-Flange is the first feature in a new sheet metal part. When a Base-Flange is added to a SolidWorks part, the part is marked as a sheet metal part. Bends are added wherever appropriate, and sheet metal-specific features are added to the FeatureManager design tree.

A Base-Flange feature creates three new features in the FeatureManager design tree. The following are the three new features:

-  **Sheet-Metal1.** The Sheet-Metal feature contains the default bend parameters. To edit the default bend radius, bend allowance or bend deduction, or default relief type, right-click the Sheet-Metal feature and select **Edit Definition**.
-  **Base-Flange.** The Base-Flange feature is the first solid feature of the sheet metal part. Expand the Base-Flange feature to see any bends created by the feature.
-  **Flat-Pattern1.** The Flat-Pattern feature flattens the sheet metal part. Notice it is suppressed by default, as the part is in its folded state. Unsuppress the feature to flatten the sheet metal part. For more information, see **Flat-Pattern** on page 7-11.

Here are some additional items to note about a Base-Flange:

- There can only be one Base-Flange feature in a SolidWorks part.
- The thickness and bend radius of the Base-Flange feature become the default values for the other sheet metal features.
- The base flange is created from a sketch. The sketch can be a single open, a single closed, or multiple-enclosed profiles. Illustrations of each are shown as follows:



Single open




Single closed







Multiple-enclosed

To create a Base-Flange feature:

- 1 Create a sketch.
- 2 Click **Base-Flange/Tab**  on the Sheet Metal toolbar or click **Insert, Features, Sheet Metal, Base Flange**.

The **Base-Flange** PropertyManager appears.

NOTE: The controls on the **Base-Flange** PropertyManager update according to your sketch. For example, the **Direction 1** and **Direction 2** boxes do not appear for a sketch with a single closed profile.

- 3 Under **Direction 1** and (optionally) **Direction 2**, set the parameters for the extrusion **End Condition** and **Depth** .
- 4 Under **Sheet Metal Parameters**, do the following:
 - Set **Thickness**  to the desired sheet metal thickness.
 - Select the **Reverse direction** check box, if necessary, to thicken the sketch in the opposite direction.
 - Set **Bend Radius**  to the desired bend radius.
- 5 Click **OK** .


Edge-Flange

An Edge-Flange feature adds a flange to your sheet metal part at a selected edge.








Here are some additional items to note about an Edge-Flange feature:

- The selected edge must be linear.
- The profile does not have to be rectangular, though one sketch line of the profile must lie on the selected edge.
- The thickness is automatically linked to the thickness of the sheet metal part.
- Reliefs are automatically added as needed. You can set the type and size of relief.

To create an Edge-Flange feature in a part with a Base-Flange feature:

- 1 Click **Edge Flange**  on the Sheet Metal toolbar or click **Insert, Features, Sheet Metal, Edge Flange**.

The **Edge-Flange** PropertyManager appears.

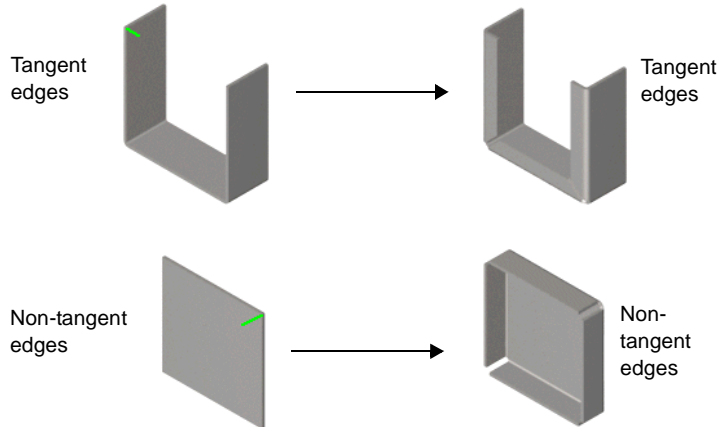
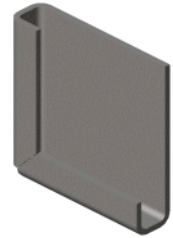
- 2 Under **Flange Parameters**, do the following:
 - Select the **Edge**  where you want to place the feature.
 - Click **Edit flange profile** to edit the sketch of the flange. Use this to change from a rectangular profile to a different profile, or to add interior holes to the profile. Also, you must use this if your edge flange is shorter than the selected edge. Click **Back** to continue editing the parameters of your Edge-Flange feature.
 - To use something other than the default bend radius, click to clear the **Use default radius** check box and set **Bend Radius**  to the desired bend radius.
- 3 Under **Angle**, set the **Flange Angle** .
- 4 Under **Flange Length**, set the end condition and its corresponding parameters. For example, if you select **Offset from Surface** you must select a surface and type an offset value.
- 5 To determine the position of the flange and its bend, set the following under **Flange Position**:
 - Set the bend position to **Material inside** , **Material outside** , or **Bend outside** . For more information, see **Bend Positions** on page 7-13.
 - To remove extra material in neighboring bends, select the **Trim side bends** check box. Extra material appears when the bend of an Edge-Flange touches another bend.
 - To offset the flange from the sheet metal body, select the **Offset** check box and set the end condition and its corresponding parameters.
- 6 Under **Relief Type**, select a relief type and set its corresponding parameters.
- 7 Click **OK** .

Miter-Flange

A Miter-Flange feature adds a series of flanges to one or more edges of a sheet metal part.

Here are some additional items to note about a Miter-Flange feature:

- ❑ The sketch can contain only lines.
- ❑ The miter flange profile can contain more than one continuous line as shown at right.
- ❑ The sketch plane must be normal to the first edge where the Miter-Flange is created.
- ❑ The thickness is automatically linked to the thickness of the sheet metal part.
- ❑ You can create a Miter-Flange feature on a series of tangent or non-tangent edges. Illustrations of each are shown as follows:



To create a Miter-Flange feature in a part with a Base-Flange feature:

- 1 Create a sketch that is perpendicular to an edge of the sheet metal part. The sketch plane must be normal to the first edge where the Miter-Flange feature is created.


TIP: For an example of creating a sketch for a Miter-Flange feature, see Step 1 under **Adding a Miter Flange** on page 7-21.

- 2 Click **Miter Flange**  on the Sheet Metal toolbar or click **Insert, Features, Sheet Metal, Miter Flange**.




The **Miter Flange** PropertyManager appears.

The first edge for the Miter-Flange feature is selected and a preview of the miter flange appears in the graphics area.

3 Under **Miter Parameters**, do the following:

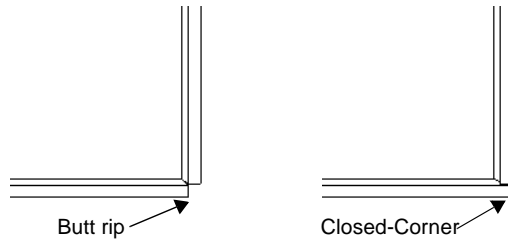
- Select the edges to miter in the **Along Edges**  box.

TIP: To select all of the edges tangent to a selected edge, click the handle that appears at the midpoint of the selected edge.

- To use something other than the default bend radius, click to clear the **Use default radius** check box and set **Bend Radius**  to the desired bend radius.
 - Set the **Flange position** to **Material inside**, **Material outside**, or **Bend outside**. For more information, see **Bend Positions** on page 7-13.
 - To remove extra material in neighboring bends, select the **Trim side bends** check box. Extra material appears when one of the bends of an Miter-Flange touches another bend.
 - To use something other than the default gap, set **Gap distance**  to the desired distance.
- 4 Click **OK** .

Closed-Corner




You can create a Closed-Corner feature to extend one face of a Butt rip so it overlaps the other face of the Butt rip. This is shown as follows:

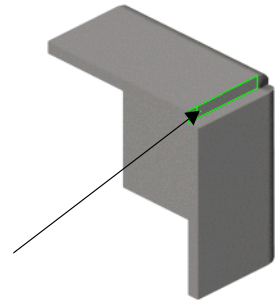


Here are some additional items to note about a Closed-Corner feature:

- You can close more than one corner at a time. Select the faces for all of the rips you want to close.
- You can select only planar faces to close.

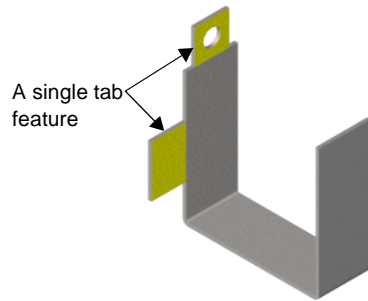
To create a Closed-Corner feature:

- 1 Create a sheet metal part with a Base-Flange and a Miter-Flange, so that one corner looks like the illustration at right.
 - 2 Click **Closed Corner**  on the Sheet Metal toolbar or click **Insert, Features, Sheet Metal, Closed Corner**.
The **Closed Corner** PropertyManager appears.
 - 3 Select one of the faces on the corner as shown as the **Faces to Extend** , then click **OK** .
- The face is extended to close the corner.



Tab


A Tab feature adds a tab to the sheet metal part. The depth of a Tab feature is automatically set to the thickness of the sheet metal part. The direction of the depth is automatically set to coincide with the sheet metal part to prevent a disjoint body.



Here are some additional items to note about a Tab:

- The sketch must be on a plane or planar face that is perpendicular to the direction of thickness of the sheet metal part.
- The sketch can be a single closed, multiple closed, or multiple-enclosed profile.
- You can edit the sketch, but you cannot edit the definition. This is because the depth, direction, and other parameters are set to match the parameters of the sheet metal part.

To create a Tab feature in a part with a Base-Flange feature:

- 1 Create a sketch on a plane or planar face that meets the above requirements.
- 2 Click **Base-Flange/Tab**  on the Sheet Metal toolbar or click **Insert, Features, Sheet Metal, Tab**.

The tab is added to the sheet metal part. The tab's depth and direction are automatically set to match the parameters of the Base-Flange feature.

Sketched-Bend

You can add bend lines to the sheet metal part while the part is in its folded state using a Sketched-Bend feature. This allows you to dimension the bend line to other folded-up geometry.


Here are some additional items to note about a Sketched-Bend feature:

- Only lines are allowed in the sketch. You can add more than one line per sketch.
- The bend radius must be greater than zero.
- The bend line does not have to be the exact length of the faces you are bending.


TIP: A Sketched-Bend feature is commonly used with a Tab feature to bend the tab.

To create a Sketched-Bend feature:


1 Create a sketch on a planar face of your sheet metal part that contains the bend lines.


2 Click **Sketched Bend**  on the Sheet Metal toolbar or click **Insert, Features, Sheet Metal, Sketched Bend**.

The **Sketched Bend** PropertyManager appears.

3 Select a face that does not move as a result of the bend as the **Fixed face** .

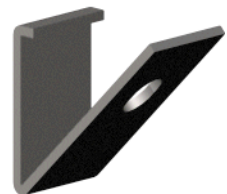
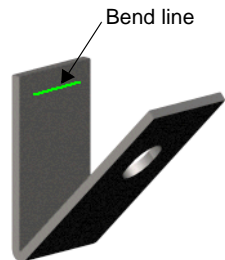
4 Select a **Bend position** of **Bend centerline**, **Material inside**, **Material outside**, or **Bend outside**. For more information, see **Bend Positions** on page 7-13.

5 Set the **Bend Angle**, and click **Flip Bend Direction**  if necessary.

6 To use something other than the default bend radius, click to clear the **Use default radius** check box and set **Bend Radius**  to the desired bend radius.

7 Click **OK** .

The new bend is created.




Unfold/Fold

Using the Unfold and Fold features, you can flatten and bend one, more than one, or all of the bends in your sheet metal part.



This combination is useful when adding a cut across a bend. First, add an Unfold feature to flatten the bend. Next, add your cut. Lastly, add a Fold feature to return the bend to its folded state.

TIP: For faster system performance, only unfold and fold the bends that you need for the task you are performing.


To add an Unfold feature, then add a Fold feature:

- 1 In a sheet metal part, click **Unfold**  on the Sheet Metal toolbar or click **Insert, Features, Sheet Metal, Unfold**.

The **Unfold** PropertyManager appears.

- 2 Select a face that does not move as a result of the Unfold feature as the **Fixed face** .
- 3 Select one or more bends as the **Bends to unfold** , or click **Collect All Bends** to select all the bends in the part.

Note that you can select only bends in the graphics area; you cannot select faces, edges, and so on.


- 4 Click **OK** .

The selected bends unfold. Notice the bend region appears in its flattened state as shown.

You can now add any cuts that you want across the flattened bend.

- 5 Click **Fold**  on the Sheet Metal toolbar, or click **Insert, Features, Sheet Metal, Fold**.

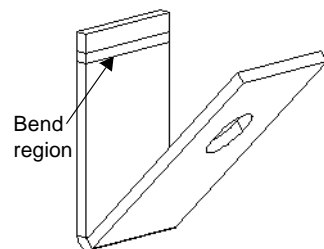
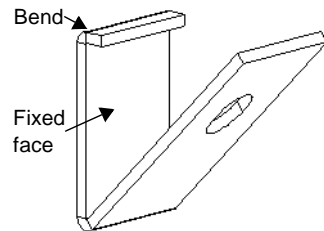
- 6 The **Fold** PropertyManager appears.

- 7 Select a face that does not move as a result of the Fold feature as the **Fixed face** .

- 8 Select one or more bends as the **Bends to fold** , or click **Collect All Bends** to select all the bends in the part.

Note that you can select only bends in the graphics area; you cannot select faces, edges, and so on.

- 9 Click **OK** .



Flat-Pattern

The Flat-Pattern feature is intended to be the last feature in the folded sheet metal part. All features before Flat-Pattern in the FeatureManager design tree appear in both the folded and flattened sheet metal part. All features after Flat-Pattern appear only in the flattened sheet metal part.

Here are some additional items to note about a Flat-Pattern feature:

- ❑ **New features in folded part.** When the Flat-Pattern feature is suppressed, all features that you add to the part automatically appear before this feature in the FeatureManager design tree. You no longer have to rollback to add features to the folded sheet metal part.
- ❑ **New features in flattened part.** You flatten the sheet metal part by unsuppressing the Flat-Pattern feature. To add features to the flattened sheet metal part, you must first unsuppress the Flat-Pattern feature.

TIP: Sometimes you need to add a feature in the flattened state, but you want it to appear in the folded sheet metal part. To do this, use the **Unfold** and **Fold** features so your new feature appears before Flat-Pattern in the FeatureManager design tree.

- ❑ **Bend lines sketch.** The sketch containing the bend lines is shown by default when you unsuppress the Flat-Pattern feature. To hide this sketch, expand the Flat-Pattern feature, right-click the Bend-Lines sketch, and select **Hide**.
- ❑ **Flat-pattern properties.** You can edit the definition of a Flat-Pattern feature. In the **Flat-Pattern** PropertyManager, you can set the **Fixed Face** and toggle the **Merge faces** check box. When you select the **Merge faces** check box, faces that are planar and coincident in the flat pattern are merged and no edges are shown in the bend regions.
- ❑ **Reorder features.** You cannot reorder sheet metal features to go below the Flat-Pattern feature in the FeatureManager design tree. So, you cannot reorder a cut with the **Normal cut** option underneath the Flat-Pattern feature.
- ❑ **Reorder bends.** Right-click Flat-Pattern and select **Reorder Bends**. In the **Reorder Bends** dialog box, select a bend and click **Move Up** or **Move Down** to order the bends as desired.
- ❑ **Progressive bend simulation.** To show how to fold or unfold the part, you can change the suppression state of one bend at a time under the Flat-Pattern feature. Additionally, to quickly toggle between different folded and flattened states, you can create configurations with different bends suppressed.

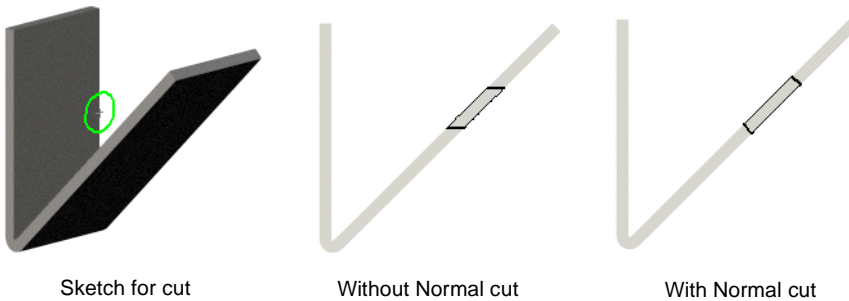
Improved Sheet Metal Features

Mirror All

You can mirror a sheet metal part and many of the bends. The only bends that are not mirrored are those that are normal to and coincident to the mirror plane; those bends are extended.

Normal Cut

When adding a cut to a folded sheet metal part, you can select the **Normal cut** check box to ensure that the cut is created normal to the sheet metal thickness. The **Normal cut** check box is on by default for sheet metal features, and is located in the **Cut-Extrude** PropertyManager. The effect of this check box is illustrated as shown below:







Bend Reliefs

A new relief type called Obround is available. You can set the size of an Obround bend relief. For comparison, the following illustrations show an Edge-Flange feature with a Rectangular relief, and a new Obround relief.



Bend Positions

In a Miter-Flange, Edge-Flange, or Sketched-Bend feature, you have to select a bend position. The following table explains your choices.

Bend Position	Preview
Material Inside  . The top of the shaded preview of the flange coincides with the top of the fixed sheet metal entity.	
Material Outside  . The bottom of the shaded preview of the flange coincides with the top of the fixed sheet metal entity.	
Bend Outside  . The bottom of the shaded preview of the flange is offset by the bend radius.	
Bend Centerline  . The bend line is placed such that it equally splits the bend region in the flattened part. This is available only for Sketched-Bend features.	No preview available.


Flat Pattern Drawings

You can create a drawing of the flattened sheet metal part. The SolidWorks software automatically creates the flattened view for you.


Here are some additional items to note about sheet metal drawings:

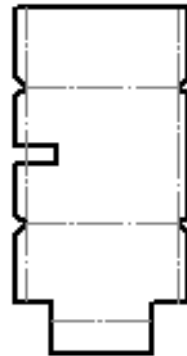
- You can select the configuration of the model to display in a flattened drawing view.
- The bend lines automatically appear in a flattened drawing view.
- By default, faces that are planar and coincident in the flat pattern are merged. For more information, see **Flat-Pattern** on page 7-11.

To create a drawing view of a flattened sheet metal part:

- 1 Create a new drawing.
- 2 Click **Named View** , and select a sheet metal part.
- 3 Click where you want to place the view in the drawing.
- 4 In the **Named View** PropertyManager, do the following:
 - a) Double-click **Flat pattern** in the **View Orientation** list.
 - b) Select a different configuration from the **Reference Configuration** list, if desired.
 - c) Set the **Custom Scale**, if necessary.

The flattened sheet metal part appears in the drawing view.
Notice that the bend lines are shown automatically.

TIP: To rotate the drawing view, click **Rotate View**  on the Standard toolbar and drag the view.



To toggle the visibility of the bend lines:

- 1 Expand the new drawing view in the FeatureManager design tree.
- 2 Locate the **Bend-Lines** sketch under the **Flat-Pattern** feature.
- 3 Right-click the sketch and select **Hide** or **Show**.

Bend Tables

In SolidWorks 2001, the user interface for bend tables has been enhanced to include the following:

- ❑ **Embedded Excel spreadsheet.** The bend table is now an embedded Excel spreadsheet. When you share your part with a colleague, your bend table is automatically included. Additionally, you add your bend table values into the cells of the spreadsheet, insuring that the table is read properly by the SolidWorks software.

NOTE: You should not leave an empty cell between values in a row in the table. An empty cell marks the end of the row.

- ❑ **Bend deduction.** You can now create a bend table with bend deduction values. This is in addition to bend allowance values (available in SolidWorks 2000). For more information, see **Bend Deduction** on page 7-16.
- ❑ **New interface for creation.** There is a dialog box to assist you in the creation of a valid bend table spreadsheet. Default radius and thickness values are automatically added to the spreadsheet.
- ❑ **Angle values.** Bend tables in SolidWorks 2001 include only values for 90° bends. Bend allowance or bend deduction values for bends of other angular values are calculated for you by SolidWorks.
- ❑ **Sample tables.** Sample tables are included with the software. The sample tables include values extracted from the Machinery Handbook - 26th Edition with permission from Industrial Press, Inc.

To create a bend table:

- 1 In a sheet metal part, click **Insert, Bend Table, New.**
A message appears to inform you that the developed length of the flat pattern may change.
- 2 In the **Bend Table** dialog box, do the following:
 - Set **Units** to the desired system of measurement.
 - Set **Type** to either **Bend Allowance** or **Bend Deduction**.
 - Type a **File name** along with the correct path.
- 3 Click **OK.**
An embedded Excel window, containing the bend table spreadsheet, appears in the SolidWorks window. The bend table spreadsheet contains default radius and thickness values.
- 4 Add the bend allowance or bend deduction values to the spreadsheet, depending on your choice of **Type** in Step 2.
- 5 Click outside the table, but within the SolidWorks graphics area, to close the spreadsheet.

To edit a bend table:

- 1 Click **Edit, Bend Table**.

An embedded Excel window, containing the bend table spreadsheet, appears in the SolidWorks window.

- 2 Edit the bend allowance or bend deduction values in the spreadsheet.
- 3 To toggle between different units of measure, type a valid name such as **Millimeters**, **Meters**, or **Inches**, in cell **B1**.
- 4 To toggle between bend allowance and bend deduction values, type **Bend Allowance** or **Bend Deduction** in cell **B2**.
- 5 Click outside the table, but within the SolidWorks graphics area, to close the spreadsheet.

To delete a bend table:

- 1 Click **Edit, Delete Bend Table**.

In a sheet metal part, a message appears to inform you that if you delete the bend table, the Sheet-Metal feature will use a k-factor of 0.5.

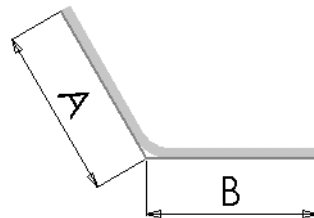
- 2 Click **Yes** to continue.

Bend Deduction

In SolidWorks 2001, bend deduction values are available for calculating the flat length of sheet stock to give the desired dimension of the bent part. The following equation is used to determine the total flat length:

$$L_t = A + B - \mathbf{BD}$$
 where


- L_t is the total flat length
- **A** and **B** are shown
- **BD** is the bend deduction value

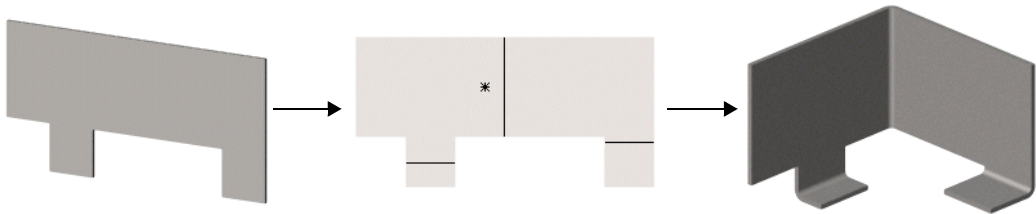


Comparison of Sheet Metal in SolidWorks 2000 and SolidWorks 2001



You can better understand the new features available in SolidWorks 2001 by comparing the old and new methods of creating sheet metal parts. The following pages include sample parts displaying how to build the part using both the old and new methods.

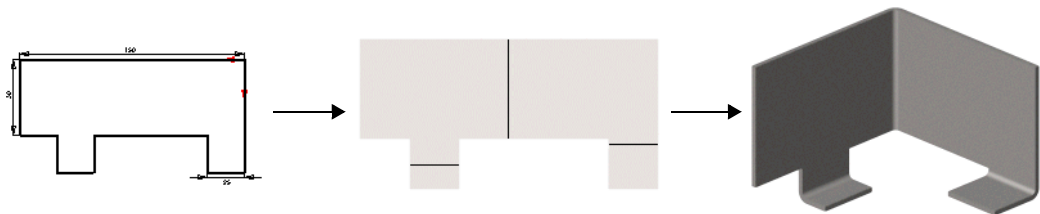
A Design from the Flattened State - SolidWorks 2000

- 1 Create a Base-Extrude feature representing the flattened sheet metal part.
- 2 Click **Insert Bends** , or **Insert, Features, Sheet Metal, Bends**, to convert the part to a sheet metal part.
- 3 Edit the **Flat-Sketch** under the **Process-Bends** feature, add the bend lines, and close the sketch.
The part appears in its folded state.
- 4 Rollback before the Sheet Metal features in the FeatureManager design tree.
- 5 Add any additional tabs, cuts, or other features to the folded model, then roll forward in the FeatureManager design tree.



A Design from the Flattened State - SolidWorks 2001


- 1 Create a sketch of the outline of the part, and create a **Base-Flange**  to convert the part to a sheet metal part.
- 2 Add a new sketch for the bend lines, and create a **Sketched Bend** .
The part appears in its folded state.
- 3 Add any additional tabs, cuts, or other features to the folded model without having to rollback.



A Design from a Solid - SolidWorks 2000

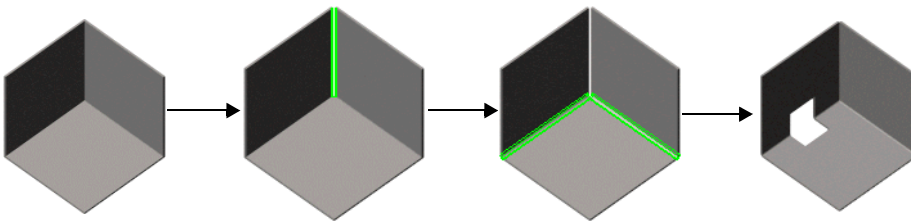
- 1 Create a part using bosses, cuts, fillets, and so on. Make sure the part is uniform in thickness (using the shell feature is recommended.)

The part shown is done with a Base-Extrude and a Shell.





- 2 Add a **Rip** feature to any edges that require a gap between adjacent faces.
- 3 Click **Insert Bends** , or **Insert, Features, Sheet Metal, Bends**, to convert the part to a sheet metal part.

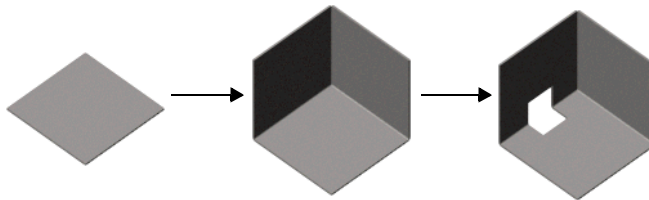
The bends and other sheet metal features are added to the part.

- 4 To add a cut across the bends, roll back above the Process-Bends feature. Add a Cut-Extrude feature, then roll forward.




A Design from a Solid - SolidWorks 2001

- 1 Create a **Base-Flange**  from a rectangular sketch.
- 2 Add a **Miter Flange**  to create the walls.
- 3 Add the cut across the bends using **Unfold** , a Cut-Extrude feature, and **Fold** .











Reasons to Use the SolidWorks 2000 Features

The SolidWorks 2000 features are still available in SolidWorks 2001. There are a few instances where it is to your advantage to use the SolidWorks 2000 features, listed as follows:

- ❑ **Imported Solid Bodies.** Use the SolidWorks 2000 features if you import a sheet metal file with bends from a legacy CAD system. Because the bends are already in the model, using **Insert Bends**  is your best option for converting the imported file to a SolidWorks sheet metal part.
- ❑ **Conical Bends.** Conical bends are not supported by the SolidWorks 2001 sheet metal features. Therefore, you must use the SolidWorks 2000 features to add bends to a conical sheet metal part.
- ❑ **Consecutive Bends.** Consecutive bends that form an “S-like” shape are not supported by the SolidWorks 2001 sheet metal features.

Combining SolidWorks 2000 Features with SolidWorks 2001 Features

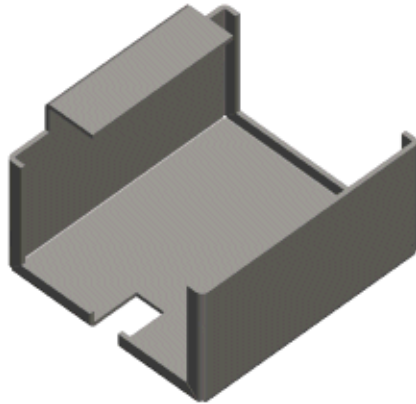
You can add SolidWorks 2001 sheet metal features to a part created with the SolidWorks 2000 features. As soon as you add a SolidWorks 2001 sheet metal feature (a **Miter Flange** , an **Edge Flange** , and so on), the following occurs:

- ❑ **Flat-Pattern1**  is added to the FeatureManager design tree, and suppressed.
- ❑ To flatten the part, unsuppress **Flat-Pattern1**  rather than suppressing **Process-Bends1** .
- ❑ Bends for new sheet metal features are stored under each individual feature instead of under **Flatten-Bends1**  or **Process-Bends1** . Additionally, all the pre-existing and new bends are listed under the **Flat-Pattern1**  feature.

Step-By-Step Example


To showcase the new sheet metal features, you create the part shown at right. This part includes the following features:

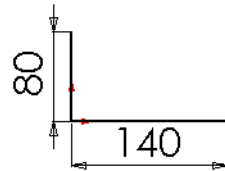
- Base Flange
- Miter Flange
- Mirror All (to create new bends)
- Tab
- Sketched Bend
- Fold and Unfold






Creating the Base Flange

To create the sheet metal part with a uniform thickness, sketch an open profile and use the *base flange* feature to create the thin feature and the bends.

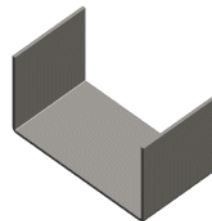
- 1 Open a new part from the **Tutorial** tab and open a sketch on the **Front** plane.
- 2 Placing the lower left corner of the sketch on the origin, sketch and dimension the profile as shown. Add an **Equal** relation between the two vertical lines.
- 3 Click **Base-Flange/Tab**  on the Sheet Metal toolbar, or click **Insert, Features, Sheet Metal, Base Flange**.



The **Base Flange** PropertyManager appears.


- 4 Under **Direction1**, leave **End Condition** as **Blind** and set the **Depth**  to 75mm.
- 5 Under **Sheet Metal Parameters**, do the following:
 - a) Set **Thickness**  to 3mm.
 - b) Set **Bend Radius** to 1mm. This is the default bend radius.
- 6 Click **OK** .

The sketch is extruded and the bends are automatically added.




Adding a Miter Flange

To add flanges to your sheet metal part with corners that are automatically mitered, use the *miter flange* feature.

- 1 Select the inside vertical edge approximately as shown and click **Sketch**  to open a new sketch.


A sketch plane is created normal to the selected edge with its origin at the closest endpoint of the edge.

- 2 Sketch a horizontal line starting from the inside vertex shown extending towards the middle of the part. Dimension the line to 8mm.


- 3 Click **Miter Flange**  on the Sheet Metal toolbar, or click **Insert, Features, Sheet Metal, Miter Flange**.

The **Miter Flange** PropertyManager appears.

- 4 Click the handle that appears at the midpoint of the selected edge.

The five tangent edges appear in the **Along Edges**  box under **Miter Parameters**.

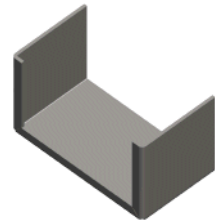
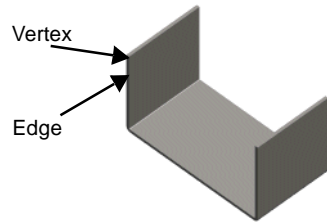
Also, a preview of the miter flange appears on the model.

- 5 Leave the **Flange position** as the default, and click **OK** .

The flange is added to the selected edges. Notice the bend reliefs are automatically added to allow the miter flanges to fold and unfold.

Notice the Miter-Flange1 feature appears above the Flat-Pattern1 feature in the FeatureManager design tree. SolidWorks automatically inserts new features above the suppressed Flat-Pattern1 feature.

- 6 Save the part as **Cover.sldprt**.

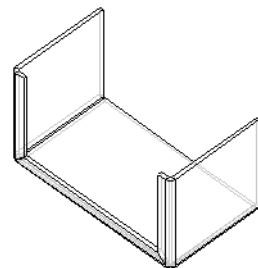



Mirroring the Sheet Metal Bends

- 1 Click **Hidden in Gray** .
- 2 Click **Insert, Pattern/Mirror, Mirror All**.

The **Mirror All** dialog box appears.

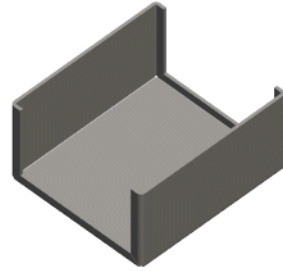
- 3 Select one of the back planar faces as the **Mirror Plane**. Use **Select Other** from the shortcut menu if necessary.



- 4 Click **OK** and click **Shaded** .

The entire part is mirrored as well as the sheet metal bends from the miter flange.

- 5 Expand the **Mirror1** feature in the FeatureManager design tree and notice that there are new bends to include the mirrored geometry.



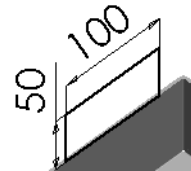
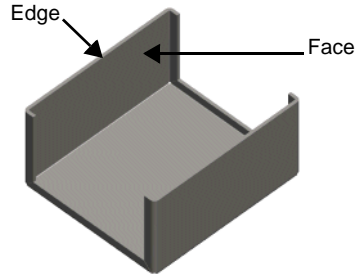
Adding a Tab

To create a tab of equal thickness to the rest of the sheet metal part, use the *tab* feature.

- 1 Open a sketch on the vertical face shown.
- 2 Sketch a rectangle above the sheet metal body with its lower edge coincident to the edge shown. Dimension the rectangle to 50mm high and 100mm wide.
- 3 Add a coincident relation between the midpoint of one of the horizontal lines of the rectangle and the **Front** plane.

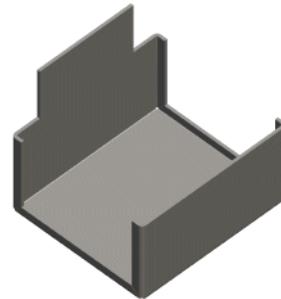
TIP: To select the midpoint of a line or edge, right-click the line or edge and click **Select Midpoint**.

The sketch should look like this when you are done.




- 4 Click **Base-Flange/Tab** .

The tab is added to the part. You do not need to specify a depth because the SolidWorks software links the thickness of the tab to the thickness of the base flange.




Bending the Tab


Once you add a tab, you must specify how to bend it using the *sketched bend* feature.

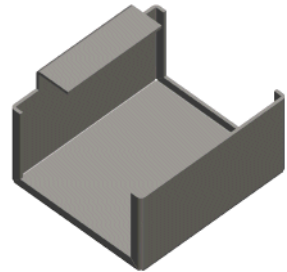
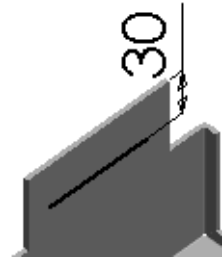
- 1 Select the same vertical face from the previous section and open a sketch.
- 2 Sketch a horizontal line of any length. Dimension it to 30mm below the top of the tab, as shown.
- 3 Click **Sketched Bend**  on the Sheet Metal toolbar, or click **Insert, Features, Sheet Metal, Sketched Bend**.

The **Sketched Bend** PropertyManager appears.

- 4 Under **Bend Parameters**, do the following:


- Select the vertical face that you are sketching on as the **Fixed face** .
- Set **Bend position** to **Bend outside**.
- Make sure **Bend Angle** is set to 90° and the **Use default radius** check box is selected.

- 5 Click **OK** .
- 6 Save the part.





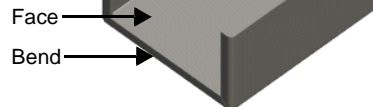
Adding a Cut Across a Bend

To cut across a bend, you start by unfolding only the bend that you want to cut across.

- 1 Click **Unfold**  on the Sheet Metal toolbar, or click **Insert, Features, Sheet Metal, Unfold**.

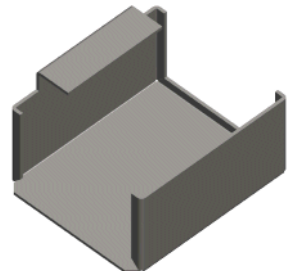
The **Unfold** PropertyManager appears.

- 2 Select the face shown as the **Fixed face**  and the bend shown as the **Bends to unfold** .





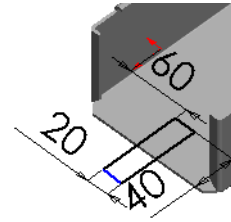
- 3 Click **OK** .

The selected bend only is unfolded.

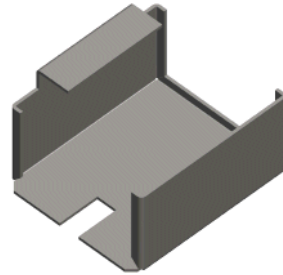


Next, you create a sketch and cut through the bend.


- 1 Open a sketch on the face shown, and sketch and dimension a rectangle, as shown. The 60mm dimension goes from the left edge of the rectangle to the origin.
- 2 Click **Extruded Cut**  or **Insert, Cut, Extrude**. Select **Through All** as **End Condition**, then click **OK** .




The cut goes through the bend region.





Now that you made your cut, return the model to its folded state.

- 1 Click **Fold**  on the Sheet Metal toolbar, or click **Insert, Features, Sheet Metal, Fold**.

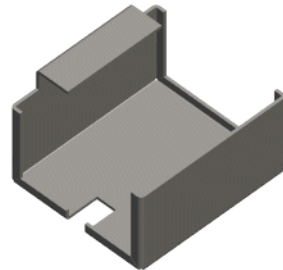
The **Fold** PropertyManager appears.

- 2 Select the bottom face of the sheet metal part as the **Fixed face** .
- 3 Click **Collect All Bends** to collect all the unfolded bends.

The unfolded bend appears in the **Bends to fold**  box.


- 4 Click **OK** .

The part returns to its fully folded state with the cut across the bends.




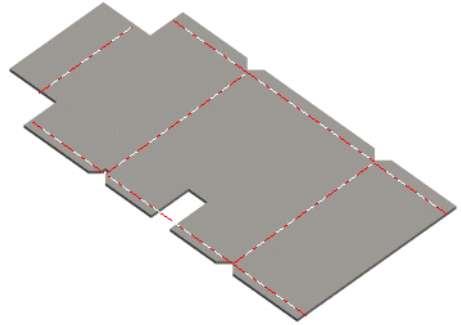
Folding and Unfolding the Entire Part

You can flatten all the bends of your sheet metal part at once.

- 1 Select the **Flat-Pattern1** feature in the FeatureManager design tree, and click **Flattened**  on the Sheet Metal toolbar.

The flattened sheet metal part appears with all of the bend lines shown.

- 2 To fold the part back up, click **Flattened**  again.
- 3 Save the part.



SolidWorks 2000 Functionality

This appendix contains information about new and changed functionality that has been introduced in the Service Packs between release of SolidWorks 2000 and release of SolidWorks 2001.

General Information

Lists general data about applications and where to locate information.

SolidWorks 3D Meeting

SolidWorks 3D Meeting is an online collaboration tool that allows you to share your SolidWorks session with anyone who has Microsoft® NetMeeting®. Click **Tools, 3D Meeting** to access the tool. For more information, click **Help, 3D Meeting Help**.

SolidWorks 3D Instant Website

SolidWorks 3D Instant Website allows you to create a web page from your SolidWorks application. You can use default templates (with embedded Viewpoint, RealityWave, CATweb, and eDrawing viewer) or customized templates. Your web page can be on a password-protected secure site hosted by SolidWorks, or on a local or network drive.


Documents on Help Menu

The following documents now appear on the Help menu: Getting Started Manual, What's New Manual, and SolidWorks Release Notes.

Assemblies

You can display assembly components in the HLR/HLG color of the component or of the assembly. Click **View, Display, Use Component Color in HLR/HLG**. When selected, assembly components appear in the color of the component.

Configurations

You can suppress and unsuppress light sources in the **Lighting** folder  of the FeatureManager design tree in configurations in the same way other features are suppressed and unsuppressed.

Mass Properties

The sign convention for products of inertia terms (L_{xy} , L_{xz} , L_{yx} , L_{yz} , L_{zx} , L_{zy}), reported as: "Taken at center of mass and aligned with the output coordinate system," has been changed from tensor sign convention to moment of inertia sign convention. Products of inertia terms previously reported as positive values are now reported as negative values, and vice versa. The actual numerical values are unchanged.

Drawings and Detailing

Drawings and detailing includes the following topics:

- Print selection
- Sketch entities in drawings
- Section views
- View alignment
- RapidDraft drawings
- Inserting duplicate dimensions
- Default tolerance options

Print Selection

The **Selection** option is now available in **File, Print** when a drawing is open in View-Only mode.

Sketch Entities in Drawings

Over defined dimensions attached to a single sketch entity in drawings update to reflect the correct value of the sketch entity. The over defined values cannot be changed. If you have existing drawings with more than one driving dimension per sketch entity, you may experience the following symptoms:

When you open an old drawing with over defined dimensions in this version of the SolidWorks software, all of the over defined dimensions with incorrect values attached to a sketch entity are updated to reflect the correct dimension value of the sketch entity.

If the dimension is over defined, you cannot change the dimension value by double-clicking on it.

Section Views

The **Hide/Show Components** tab has been added to the **Properties** dialog box for Section Views.

Hide Behind Plane has been implemented for Section Views. Right-click a non-sectioned face and select **Show/Hide, Hide Behind Plane**.

View Alignment

In the Drawing View shortcut menu, the following items under **Alignment** have changed:

- **Align Horizontal** is now **Align Horizontal by Origin**
- **Align Vertical** is now **Align Vertical by Origin**
- **Align by origin** is the same functionality as previously, where views align according to the origins of the models. The following items have been added:
 - **Align Horizontal by Center**
 - **Align Vertical by Center**

With align by center, the center points of the view geometry determine the alignment. This option is useful for aligning views of different models where the origins may not match up.

RapidDraft Drawings

Virtual Sharps can now be added in RapidDraft drawings without the model being loaded into memory.

Internal changes have been made to reduce file size in RapidDraft drawings.

An option has been added for controlling the out-of-sync watermark in RapidDraft drawings. Click **Tools, Options**. On the **System Options** tab, select **Drawings**. To turn off printing of the watermark, clear the **Print out-of-sync watermark in RapidDraft** check box.

Inserting Duplicate Dimensions

When you insert model items into drawings, you now have the option of inserting duplicate dimensions. Two new check boxes control insertion of duplicate dimensions.

- 1 Click **Tools, Options, System Options, Drawings**. **Eliminate duplicate model dimensions on insert** is selected by default. Clear this check box for duplicate dimensions to be inserted into drawings.
- 2 Click **Insert, Model Items**. The **Eliminate duplicate model dimensions** check box in the **Insert Model Items** dialog box follows the option setting. You can select or clear this check box for each insertion of dimensions.

Default Tolerance Options

You can now set default tolerance options under **Tools, Options, Document Properties, Dimensions, Tolerance** so that your settings are available when you access the **Dimension Properties** dialog box in a drawing.

Features

In the Hole Wizard, **Add Cosmetic Thread** is now a three-item list instead of a check box. The choices are:

- **No Cosmetic thread**
- **Add Cosmetic thread with thread callout**
- **Add Cosmetic thread without thread callout**

Fundamentals

The SolidWorks Corporation License and Subscription Service Agreement that is displayed during installation is now available for review after installation. In an active SolidWorks session, click **Help** and select **About SolidWorks 2001**. In the dialog box, click **License Agreement**.

Import/Export

The following translators are now available as add-ins:

- **CGR** - exports SolidWorks documents to CGR (CATIA® graphics) file.
- **DXF 3D** - extracts ACIS (.sat) information from DXF files
- **HCG** - exports SolidWorks documents to HCG (highly compressed graphics) files
- **IPT** - imports Autodesk Inventor™ (.ipt) files
- **MDT** - imports (MDT) Mechanical Desktop® files
- **MTS** - exports SolidWorks documents to MTS (Metastream, now called Viewpoint) and MTX files
- **PAR** - imports the Parasolid information of a Solid Edge™ (.par) file
- **ZGL** - exports SolidWorks documents to RealityWave ZGL files

DXF/DWG

Insert, Sketch from Drawing is available for imported DXF/DWG entities.

IGES and Parasolid files

You can now export hidden or suppressed components in a SolidWorks assembly document to IGES or Parasolid files.

You can now export a model to IGES format as both trimmed surfaces and 3D curves.

HTML Format

The SldHTML Add-In has been removed. Files cannot be saved in HTML format.

Mechanical Desktop

The Mechanical Desktop (MDT) translator supports the following:

- MDT4.0 or higher (MDT4.0 with Windows 2000 is not supported)
- MDT5.0 only with Windows 98, Windows 2000, or Windows NT (Windows 98 must have MDT5.0 SP1 or later)

MetaStream

MetaStream is now called Viewpoint in the Tools, Add-Ins list, and in the Save as type list.

Pro/ENGINEER

You can now choose to import Pro/ENGINEER files as surface models.

Unigraphics II

There have been modifications to how SolidWorks handles the import of Unigraphics® II files. See the SolidWorks 2001 Online User's Guide for more information.

When you import a Unigraphics II part or assembly, a dialog box now gives you the option to form surfaces or solids.

Options

The following new options were included in SolidWorks 2000.

Transparency

The Transparency choices in **System Options** have been changed to allow the selection of high quality for normal (static) view mode and also for dynamic view mode. Previously, when you rotated or moved a model with high-quality transparency, the application switched to low-quality transparency. Now, you can retain high-quality transparency while rotating or moving a model using the rotation or pan tools. Depending on your graphics card, this option may result in slower performance.

To set the options, click **Tools, Options**. On the **System Options** tab, select **Performance**. In the **Transparency** section, select **High quality** for normal view mode or **High quality for dynamic view mode**, or both.

Dynamic Motion Drawing View

A new check box, **Smooth dynamic motion of drawing view**, has been added to **Tools, Options, System Options, Drawings**. When selected (the default), dynamic operations to drawings, such as panning and zooming, display smoothly. In most cases, keep the default. For some combinations of drawing size (small drawings in particular), graphics card, and system performance, operations such as panning and zooming may cause slower display performance than in previous versions. In such cases only, clear the **Smooth dynamic motion of drawing view** check box.

Display Names In Reference Geometry

A new option has been added so that you can choose whether to display the names of reference geometry entities such as planes, axes, and coordinate systems that appear in drawings. In **Tools, Options, System Options, Drawings**, select or clear the **Show Reference Geometry names in drawing** check box. The names do not appear in RapidDraft drawings.

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