SolidWorks 2001Plus

What's New

U.S. Patents 5,815,154, 6,219,049, 6,219,055

SolidWorks Corporation is a Dassault Systemes S.A. (Nasdaq:DASTY) company.

Document Number: SWXWNENG091501

Contents

Chapter 10 Import and Export

Chapter 11 Sheet Metal

Appendix A SolidWorks 2001 Service Pack Enhancements

Introduction

About This Book

There are many new features available in the SolidWorks® 2001Plus 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 2001Plus 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 2001Plus 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 unless instructed to do so.

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 2001Plus** in the *Online User's Guide*.

Moving to SolidWorks 2001Plus

Backup Copies

We recommend that you save backup copies of all SolidWorks documents (parts, assemblies, and drawings) before opening them in SolidWorks 2001Plus. These documents are automatically converted to SolidWorks 2001Plus 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 2001Plus

Because of changes to the SolidWorks files with the development of SolidWorks 2001Plus, 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 2001Plus 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 2001Plus**, then **SolidWorks Tools**. 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 2001Plus 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 2001Plus Getting Started

The *SolidWorks 2001Plus 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 2001Plus application.

1

SolidWorks Fundamentals

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

- ❑ Units of measure
- ❑ Planes and axes
- ❑ Shortcut menus
- ❑ Printing
- ❑ User interface
- ❑ Macros
- ❑ Display
- ❑ Library features
- ❑ Toolbars
- ❑ PropertyManager

Units of Measure

New units of measure are available in SolidWorks 2001Plus. These units include angstroms, nanometers, microinches, microns, and mils. The lower limit for dimension values has been changed from 0.001mm to 0.0001mm. The lower limit for the following units are:

- angstroms (A) = 1000
- nanometers (nm) = 100
- microinches (uin) = 3.937
- microns (um) = 0.1
- mils (mils) = 0.003937008

TIP: When using small units such as microns, you may want to create specific templates for these units. When creating a template, be sure to update the appropriate settings such as spin increment, grid spacing, and so on.

To set the units of measure for a document:

1. Click **Tools**, **Options**, **Document Properties**, **Units**.
2. Under **Linear units**, select the units to apply to the document.
3. Click **OK**.

TIP: If you type the unit of measure in a value box, you can use the unit abbreviations as noted in the parenthesis above.

Planes and Axes

SolidWorks 2001Plus includes new functionality for planes and axes, such as using shaded planes, creating multiple planes, and autosizing planes and axes. For more information about planes, see **Planes** on page 4-2.

Shaded Planes

In order to better visualize the plane on which you are working, you can now use shaded planes. Shaded planes are especially useful in complex assemblies, and they help you see how parts are associated with the planes. When you rotate the model, the planes change color to indicate the front and back sides of the plane. For more information about shaded planes, see **Transparent Shaded Planes** on page 4-5.

Autosize

Planes and axes you create now automatically size to either the geometry on which they are created, or to the bounding box of the model geometry. As the geometry sizes change, the planes and axes update accordingly. If a plane or axis is created with no geometry to determine the size of the plane or axis (default planes, offset planes, and so on), the plane or axis extends to the bounding box of the geometry.

You can override the autosize by manually changing the size of the plane or axis. If you override the autosize, you can re-activate it by right-clicking the item in the FeatureManager® design tree and selecting **Autosize**. For more information about autosizing, see **Automatic Sizing of Planes and Axes** on page 4-4.

Shortcut Menus

Open Drawing

You can now open drawings directly from part and assembly documents. See **Opening Drawings from Model Documents** on page 9-2.

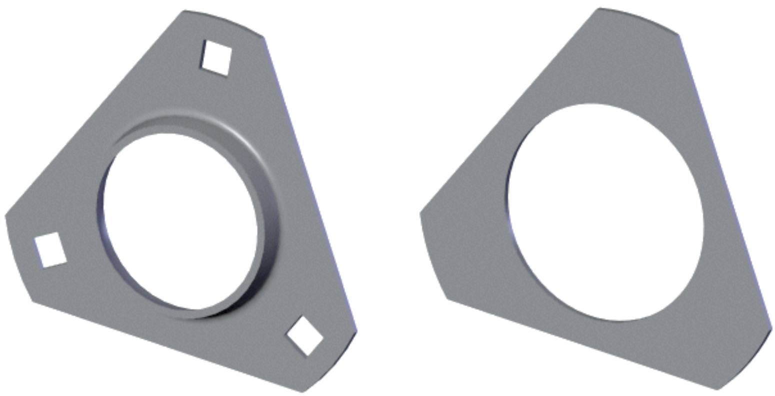
Rollback

The **Rollback** option is now available on the shortcut menu. When you roll a model back to an earlier state, you temporarily suppress the features below the rollback bar. Right-click any feature in the FeatureManager design tree, and select **Rollback** to roll your model back to the state before that feature.

To roll a model back using the shortcut menu:

1. Open **rollback.sldprt**.
2. Right-click **Cut-Extrude1** in the FeatureManager design tree and select **Rollback**.

   The part rolls back to its state before the extruded cut.



Before rollback　　After rollback

Select Chain

Select Chain has been added to the shortcut menu when selecting sketch entities. This command selects all sketch entities attached to the selected entity in both directions until a branch is encountered. For more information about select chain, see **Select Chain** on page 3-2.

Delete Face

If you want to delete a face in a surface body, right-click the face and select **Delete Face**, or click  on the Surfaces toolbar, or click **Insert**, **Face**, **Delete**. Previously, you had to select a face, then press **Delete**. For more information about the **Delete Face** functionality, see **Deleting Faces in Surface Bodies** on page 6-8.

Delete

In the PropertyManager, you can right-click items in a list to delete them from the selection. For example, to delete a face selection from the **Shell** PropertyManager, right-click the face and click **Delete**.

Printing

When you set printing parameters, some options are saved as document options, while others are saved as system options. Document options include headers and footers, scale, orientation, paper size, and paper source. System options include line weights, margins, and background. Additional printing enhancements are described below.

NOTE: You must have a SolidWorks document open to set printing parameters.

Document Options

Header/Footer

There is a new method to set predefined or custom headers and footers for individual documents.

To create a header/footer:

1 Click **File**, **Print**.

The **Print** dialog box appears.

2 Under **Document Options**, click **Header/Footer**.

The **Header/Footer** dialog box appears.

3 Select a predefined header or footer from the **Header** and **Footer** lists.

The predefined header or footer appears in the appropriate preview boxes.

- or -

Click **Custom Header** or **Custom Footer** to create a unique header or footer, select the items you want to appear on the page (**Page Numbers**, **Number of Pages**, **Date**, **Time**, or **Filename**), then click **OK**.

4 Click **OK** to close the **Header/Footer** dialog box.

5 Click **OK** to close the **Print** dialog box and print the document.

System Options

Line Weights

There is a new method to set the line weights that work best with your printer or plotter.

To set line weights:

1 Click **File**, **Print**.

The **Print** dialog box appears.

2 Under **System Options**, click **Line Weights**.

The **Line Weights** dialog box appears.

3 Set the line weights (**Thin**, **Normal**, and so on) to the desired values, then click **OK**.

4 Click **OK** to close the **Print** dialog box and print the document.

Margins

There is a new method to set values for the top, bottom, left, and right margins for the printed document.

To set print margins:

1 Click **File**, **Print**.

The **Print** dialog box appears.

2 Under **System Options**, click **Margins**.

The **Margins** dialog box appears.

3 Clear the **Use printer’s margins** check box.

4 Set the **Top**, **Bottom**, **Left**, and **Right** margins to the desired values, then click **OK**.

5 Click **OK** to close the **Print** dialog box and print the document.

Drawings

Settings for individual drawing sheets

When you print a drawing for the first time, SolidWorks sets the default printer and paper size to the settings used in the last drawing printed. If you select **Set each sheet individually**, SolidWorks sets a default printer for each sheet according to its page size.

If you want to change the default settings, you can set specific printing options for each sheet in a drawing. For example, you can choose different settings such as the printer, scale, orientation, and so on, for each sheet.

To set print settings for drawing sheets:

1 Click **File**, **Print**.

The **Print** dialog box appears.

2 Under **Document Options**, click **Page Setup**.

The **Page Setup** dialog box appears.

3 Under **Individual Drawing Sheet Control**, select **Set each sheet individually**, and do the following:

- **a)** Select the sheet for which you want to apply the print settings from the **Settings for** list.
- **b)** Choose print settings under **Scale**, **Paper**, **Drawing Color**, and **Orientation**.
- **c)** Repeat steps a and b for each sheet in the drawing.
- **d)** Click **OK**.

4 Click **OK** to close the **Print** dialog box and print the document.

Macros

If you click **Edit Macro** to edit macros that are password protected, the **Macro Password Validation** dialog box appears to verify your password.

Display

Transparency in Assemblies

There are new tools on the Assembly toolbar for setting transparency when you edit assembly components:

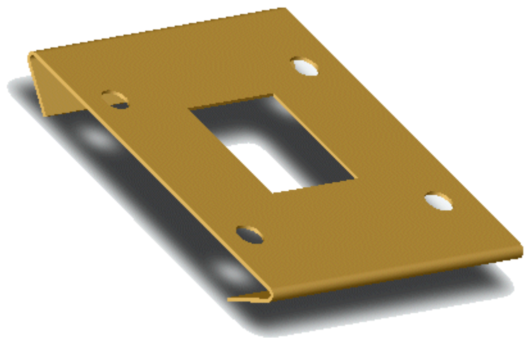
- **Opaque**. All components become opaque gray, except for the component you are editing, which becomes opaque pink.
- **Force Transparency**. All components become transparent except the one you are editing, which becomes opaque pink.
- **Maintain Transparency**. All components remain in their current state, except for the one you are editing, which becomes opaque pink.

NOTE: You can set the transparency level for **Force Transparency** by clicking **Tools**, **Options**, **System Options**, **Display/Selection**.

For more information on transparency in assemblies, see **Using Transparency in Assemblies** on page 7-9.

Shadows

In a part or assembly document, you can display a shadow under the model. When shadows are displayed, the light appears from the top-most part of the model in the current view. The shadow does not dynamically change positions when you rotate the model.



To turn on shadows:

Click **Shadows In Shaded Mode** on the View toolbar, or click **View**, **Display**, **Shadows In Shaded Mode**.

NOTE: To change the placement of the shadow, turn **Shadows In Shaded Mode** off, rotate the model, then turn the shadows on again.

Zebra Stripes

Zebra Stripes indicate how smooth the transition is between adjacent faces. For more information about zebra stripes, see **Curvature Continuous Fillets** on page 5-7, and **Checking a Surface Boundary with Zebra Stripes** on page 6-5.

To turn on zebra stripes:

1 In an open part, click **View**, **Display**, **Zebra Stripes**.

The **Zebra Stripes** PropertyManager appears.

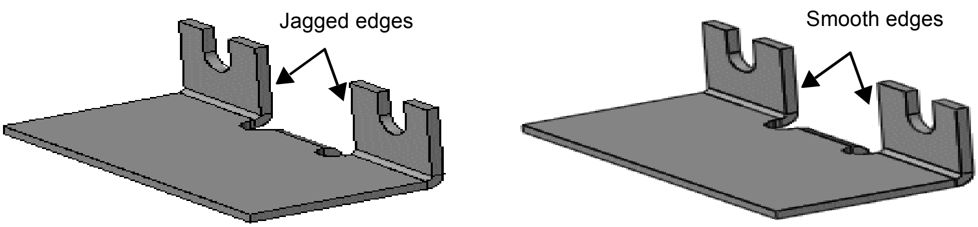
2 Under **Settings**, do the following:

- Set the **Number Of Stripes** .
- Set the **Width Of Stripes** .
- Set the **Stripe Accuracy** . A higher accuracy (when the slider is to the far right) produces a clearer image, but is slower to calculate.
- Click **Edit Color** to adjust the **Color Of Stripes** or **Color Of Background** if desired.

3 Click **OK** .

Anti-Alias Edge Display

The anti-alias edge display option smooths out jagged highlighted edges in **Display HLR Edges In Shaded Mode** .



To use the anti-alias edge display option in Display HLR Edges In Shaded Mode:

1 In an open SolidWorks document, click **Tools**, **Options**, **System Options**, **Display/Selection**.

2 Select the **Anti-alias HLR edges in shaded and fast HLR/HLG modes** check box.

3 Click **OK**.

Display HLR Edges in Shaded Mode Color

When your model is in **Display HLR Edges In Shaded Mode** , you can customize the color of the shaded edges. The default color is black.

To customize the color of shaded edges in Display HLR Edges in Shaded Mode:

1. In an open SolidWorks document, click **Tools**, **Options**, **System Options**, **Colors**.
2. Under **System colors**, click **HLR Edges in Shaded Mode**, then click **Edit**.
3. Select a color from the color palette or define a custom color, then click **OK**.
4. Select the **Use specified color for HLR edges in shaded mode** check box.
5. Click **OK**.

Show/Hide Curves, Sketches, and Annotations

You can globally show all curves, sketches, or annotations in a SolidWorks document by selecting the feature in the **View** menu. To hide these features, deselect the feature in the **View** menu.

To hide all items in the **View** menu, (planes, axes, curves, and so on) click **View**, **Hide All Types**.

NOTE: If you select **Hide All Types**, you cannot show any hidden items until you deselect **Hide All Types**.

Using the Middle Mouse Wheel to Zoom

If you use a mouse with a wheel in the middle, you can zoom in to the position of the pointer instead of the center of the graphics area in a SolidWorks document. This is the default. If the pointer is outside of the graphics area, the center of the model zooms to view.

NOTE: While rolling the mouse wheel, you must keep the pointer on the area where you want to zoom.

To turn this function off, click **View**, **Modify**, **Zoom About Screen Center**.

Remove Detail During Zoom/Pan/Rotate

The **Remove detail during zoom/pan/rotate** option is used in SolidWorks assemblies for faster graphics display. This option removes small and interior components and faces from the graphics area to improve system performance.

To turn on Remove detail during zoom/pan/rotate:

1 Click **Tools**, **Options**, **System Options**, **Performance**.

2 Under **Assemblies**, select the **Remove detail during zoom/pan/rotate** check box.

NOTE: This option is automatically disabled when you move or rotate a component, during mate animation, and during drag and drop animation.

---

3 Click **OK**.

NOTE: When **Remove detail during zoom/pan/rotate** is turned on, the **Optimize Zoom/Pan/Rotate** option is available if the model is changed. Click **View**, **Display** or right-click in the graphics area and select **Optimize Zoom/Pan/Rotate** to recalculate which components and faces should be hidden.

Drawings

When you create a drawing, you now have the option to display the model in **Shaded**  mode and to select **Display HLR Edges In Shaded Mode** . These views help you see the model easier in a drawing. For more information, see **Displaying Drawing Views in Shaded Mode** on page 9-2.

Library Features

Holes on non-planar surfaces that you create with the Hole Wizard can now be saved as library features.

Toolbars

There are two new toolbars in SolidWorks 2001Plus. These include:

- **Routing**. See **New Routing Toolbar** on page 2-6.
- **2D to 3D**. See **2D to 3D Conversion** on page 3-1.

PropertyManager

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:

- Add Relations
- Base-Thicken
- Bends
- Block
- Center Mark
- Composite Curve
- Curve Driven Pattern
- Cut (2D to 3D toolbar)
- Cut-Thicken
- Delete Face
- Display/Delete Relations
- Dowel Symbol
- Edit Bends
- Extrude (2D to 3D toolbar)
- Hole Callout
- Import Diagnosis
- Insert Bends
- Join
- Knit Surface
- Mirror
- Mirror All
- Mirror Components
- Move/Copy Surface
- Note
- Offset Surface
- Perspective View
- Planar Surface
- Plane
- Projected Curve
- Radiate Surface
- Replace Face
- Rip
- Route Line
- Scale
- Sketch Fillet
- Sketch Picture
- Sketch Plane
- Sketch Text
- Split Line
- SurfaceCut

Certain functions that use a wizard, such as **Mirror Components**, have been placed in the PropertyManager. Instead of using the **Next** and **Back** buttons in a dialog box, the PropertyManager uses next and back arrows for the new buttons.

You can also create your own next and back buttons. Name your bitmaps *<skin bitmap file name>*__**next.bmp** or *<skin bitmap file name>*__**back.bmp**, respectively.

Next and Back buttons using the wood skin

NOTE: The "__" is a double underscore.

2

Sketching

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

- ❑ User interface enhancements in sketching
- ❑ Override Dims on Drag with Move/Copy
- ❑ New Routing toolbar
- ❑ Pictures on sketch planes
- ❑ Sketch text on curves
- ❑ 3D sketching relations
- ❑ Enhancements to splines

User Interface Enhancements in Sketching

The Confirmation Corner for **Exit Sketch** now allows you to exit the sketch without saving changes by clicking **Cancel** ✘. This functionality is also available as a menu item, **Edit**, **Exit Sketch without Saving Changes**. A prompt asks if you are sure you want to discard the changes.

The following sketch menu items have been moved from the **Tools**, **Sketch Tools** menu to a new menu, **Tools**, **Sketch Settings**:

- **Automatic Relations**
- **Automatic Solve**
- **Automatic Inferencing Lines**
- **No Solve Move**
- **Detach Segment on Drag**
- **Override Dims on Drag** (see **Override Dims on Drag with Move/Copy** on page 2-5)

The **Align Grid** item on the **Tools, Sketch Tools** menu is now **Align**, with submenu items **Grid** and **Sketch**. The following items have been added to the **Tools, Sketch Tools** menu. For further information, see Chapter 3, “2D to 3D Conversion.”

- **Create new Sketch**
- **Repair Sketch**
- **2D to 3D**

The following sketch functions have been moved from dialog boxes to the PropertyManager and are discussed in detail in the next sections:

- **Edit Sketch Plane**
- **Sketch Fillet**
- **Add Relations**
- **Display/Delete Relations**

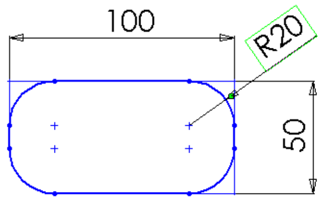
Edit Sketch Plane

In any part document, in the FeatureManager design tree, right-click a sketch and select **Edit Sketch Plane**. The **Sketch Plane** PropertyManager appears with the name of the plane in the **Sketch Plane/Face** box.

Sketch Fillet

You can pre-select the sketch entities to be filleted, or you can click the tool first, set the properties in the **Sketch Fillet** PropertyManager, and then select the sketch entities.

To fillet sketch entities:



1 Open a new part document, open a sketch, sketch a rectangle, and dimension the rectangle to 100 X 50.

2 Select the two dimensioned lines and click **Fillet** on the Sketch Tools toolbar.

The **Sketch Fillet** PropertyManager appears. Click **Keep Visible** if it is not already pushed in.

3 Set **Radius** to 20.00mm, select **Keep constrained corners**, and click **OK** .

The corner is filleted and an **Undo** button appears in the PropertyManager. The dimensions remain in place, now referencing the virtual sharp.

NOTE: If you clear **Keep constrained corners** and select a sketch entity with a dimension, the dimension is deleted.

---

4 With the **Sketch Fillet** PropertyManager still open, select two other lines of the rectangle, or a vertex.

The corner is filleted.

5 Continue to fillet the remaining corners.

6 Click **OK** to exit the **Sketch Fillet** PropertyManager, but keep the sketch open for the next procedure.

Add Relations

Add Relations is now handled in two ways: in the **Existing Relations** and **Add Relations** sections in sketch entity PropertyManagers, and in the **Add Relations** PropertyManager.

You can now use sketch relations suppression states in configurations. See **Sketch Relations** on page 8-4.

Only relations appropriate for the selected entities are available. For example, if you select two lines in a sketch, buttons are available for adding the following relations:

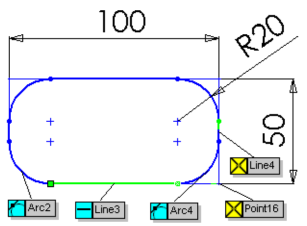
- **Horizontal**
- **Vertical**
- **Colinear**
- **Perpendicular**
- **Parallel**
- **Equal**
- **Fix**

Other combinations of sketch entities make additional relations available, including:

- **Coincident**
- **Concentric**
- **Coradial**
- **Intersection**
- **Merge**
- **Midpoint**
- **Tangent**

Relations are displayed in the **Existing Relations** box. Select the relation and a callout appears in the graphics area attached to the related sketch entity. The callout shows the name of the related entity and the type of relation.

To view the existing relations callouts:



1. In the example sketch from the previous procedure, select the bottom line in the rectangle.
2. In the **Existing Relations** box of the **Line** PropertyManager, select each of the relations.

   The relations callouts appear in the graphics area as you select them. The selected line is horizontal, tangent to the two arcs, and intersects Line4 and the point at the virtual sharp.
3. Click in the graphics area to close the **Line** PropertyManager.
4. Keep this example sketch open for the next procedure.

You can delete relations by deleting them from the **Existing Relations** box.

When you click **Add Relations** on the Sketch Relations toolbar, the **Add Relations** PropertyManager appears with **Keep Visible** active. The **Add Relations** PropertyManager contains the same sections for **Existing Relations** and **Add Relations** that appear when you select individual or multiple sketch entities.

Display/Delete Relations

Functionality for **Display/Delete Relations** is now in the PropertyManager.

For models with configurations, configuration control is available for sketch relations in the **Sketch Relations** PropertyManager. See **Sketch Relations** on page 8-4.

To view relations in a sketch:

1. In the example sketch from the previous procedure, click **Display/Delete Relations** on the Sketch Relations toolbar.

   The **Sketch Relations** PropertyManager appears with **All in this sketch** selected from the list of **Criteria** under **Relations**.

2 Select various relations from **Relations** and note the sketch entities listed under **Entities** and highlighted in green in the graphics area.

Note the following information:

- On the **Information** line under **Relations**, the status of the relation (**Satisfied**, for example) is displayed.
- Under **Entities**, the **Status** of the sketch entity (**Under Defined**, **Fully Defined**, and so on) is displayed, along with the location of the entity (**Current Sketch**, for example).

3 In the **Criteria** list under **Relations**, choose **Selected Entities** from the list, and select the bottom line of the rectangle.

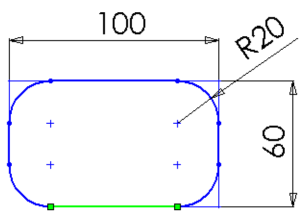
The name of the line appears in the **Selected Entities** box. Note the relations listed.

4 Click **OK** to close the **Sketch Relations** PropertyManager.

5 Keep this example sketch open for the next procedure.

Override Dims on Drag with Move/Copy

The option **Override Dims on Drag** in **Tools**, **Options**, **System Options**, **Sketch** is now called **Override Dims on Drag/Move**. **Override Dims on Drag** on the **Tools**, **Sketch Tools** menu is now **Override Dims on Drag/Move** on the **Tools**, **Sketch Settings** menu. The new name reflects that you can override the dimensions in **Move/Copy**.



1 With the example from the previous procedure open, click **Tools**, **Sketch Settings** and make sure that **Override Dims on Drag/Move** is checked.

2 Select the top line.

3 In the **Line** PropertyManager, click **Move/Copy**.

4 In the **Move/Copy** PropertyManager, under **Transformation**, **Translate**, set **Delta Y** ΔY to 10.00mm and click **Apply**.

The line moves up 10mm and the vertical dimension adjusts from 50mm to 60mm.

5 Close the sketch.

New Routing Toolbar

A new toolbar, Routing, contains two new tools for sketch lines: **Route Line** and **Jog Line**.

Route Line

The **Route Line** tool is used in explode line sketches. An Explode Line Sketch is a 3D sketch, and you can drag and jog the lines in the sketch as in any 3D sketch.

You insert an **Explode Line Sketch** into an assembly while the Configuration tab is selected, then you insert explode lines into the sketch with **Route Line**. Only one Explode Line Sketch can be inserted into a configuration.

For an example of creating explode lines in an assembly, see **Assembly Explode Lines** on page 7-6.

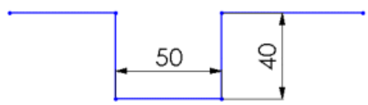
Jog Line

You can use the **Jog Line** tool to jog lines in 2D and 3D sketches in part, assembly, and drawing documents. The jog lines are automatically constrained to be perpendicular and parallel to the original line. You can drag and dimension jog lines.

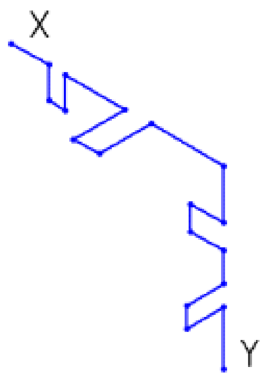
The **Jog Line** tool stays active so you can insert multiple jogs.

In 3D sketches, you can press **Tab** to toggle between two planes.

To jog a sketch line in a 2D sketch:



1. Open a new part and open a 2D sketch.
2. Sketch a line and click **Jog Line** on the Sketch Tools toolbar, or click **Tools**, **Sketch Tools**, **Jog Line**.
3. Click on the line where you want the jog to start, move the pointer to the width and height of the jog, and click to define the jog.
4. Dimension the width and height of the jog if necessary.
5. Exit the sketch.



To jog a sketch line in a 3D sketch:

1. Open a 3D sketch and click **Isometric** on the Standard Views toolbar.
2. Sketch a line on the X axis, then a second line on the Y axis.
3. Click **Jog Line**, or click **Tools**, **Sketch Tools**, **Jog Line**.

4 Click on the horizontal line where you want the jog to start, move the pointer to the width and height of the jog, and click to define the jog.

5 Click again on the same line and start the jog, but press **Tab** to change planes, move the pointer to the width and height of the jog, and click to define the jog.

6 Repeat steps 4 and 5 on the vertical line.

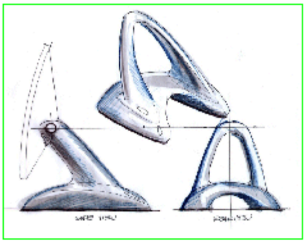
7 Exit the sketch.

Pictures on Sketch Planes

You can insert pictures (**.bmp**, **.gif**, **.jpg**, **.tif**, or **.wmf** format) onto either side of sketch planes. Pictures can be used as an underlay for creating 2D sketches. The **Sketch Picture** PropertyManager controls a graphic's position, size, angle, orientation, and whether to maintain its aspect ratio. A few items to note about pictures on sketch planes are:

- Sketch pictures are shown in the FeatureManager design tree, under their sketches
- The shortcut menu contains **Edit Sketch**, **Suppress**, and **Delete**
- If you sketch on top of the picture, there is no snap to picture or autotracing capability.
- If you hide the sketch, the image is also hidden.
- The image is not linked. If you change the image, the sketch does not update.

To insert a bitmap into a sketch:



1 Open a new part and open a sketch.

2 Click **Tools**, **Sketch Tools**, **Insert Picture** and open **bitmap_on_sketch_plane.bmp**.

The picture is inserted with the zero, zero point of the picture at the sketch origin, and the **Sketch Picture** PropertyManager appears. The initial size is 1 pixel per 1 mm. **Lock Aspect Ratio** is selected by default.

3 Under **Properties**, try the following changes to see their effect on the picture.

- Set both **Origin X Position** and **Origin Y Position** to -100.00mm.
- Set **X Width** to 100.00mm. Since **Lock Aspect Ratio** is selected, **Y Height** automatically adjusts.
- Increase the **Angle** to 10.00deg. The picture rotates in the counterclockwise direction. Decrease the angle to return to rotate clockwise.
- Click **Flip Horizontally** and **Flip Vertically**. Click each again to return to the original orientation.

4 Double-click the picture and resize it by dragging the handles. You can also drag the picture without changing its size when the pointer changes to .

5 Click **OK**.

Sketch Text on Curves

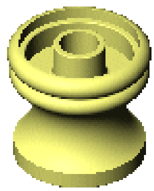
You can sketch text on the face of a part and extrude or cut the text. Now this capability has been extended so that the text can be sketched on any set of continuous curves or edges, including circles or profiles made up of lines, arcs, curves, or splines. You can insert multiple texts in a sketch.

You can control the appearance of the text in the following ways:

- Align the text left, right, or center, or justify it
- Flip the text above or below the curve, and flip it horizontally
- Scale the letters horizontally, scale the spaces, or scale letter size by percentage
- Select a font face, style, size, and effects
- Add bold or italic style
- Rotate text

To edit text in a sketch, open the sketch, and double-click the text (the pointer changes to when it is over the text).

To add to text to a part:

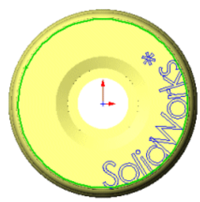


1 Open **sketch_text.sldprt**.

2 Click **Bottom** on the Standard Views toolbar and open a sketch on the bottom face.

3 Select the outside edge of the face and click **Text** on the Sketch Tools toolbar.

The **Sketch Text** PropertyManager appears with the name of the edge under **Curves** in the **Select Edges, Curves, Sketches, Sketch segments** box.



4 Under **Text**, type **SolidWorks *** (with a space after the asterisk).

The text appears inside the edge on the part in the document's font.

5 Click **Full Justify** so the text fills the curve.

6 Under **Text**, select the text "Solid" and click **Bold**.

7 Clear **Use document's font** and click **Font**.

8 In the **Choose Font** dialog box, do the following:

- Under **Font**, select **Arial** from the list.
- Under **Height**, choose **Points** and select **16** from the list.

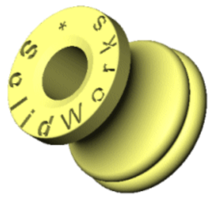
9 Click **OK** to close the dialog box, and click **OK** to close the PropertyManager.

To extrude the text:

1 With the sketch still open, click **Extruded Cut** on the Features toolbar.

2 In the **Cut-Extrude** PropertyManager, under **Direction 1**, leave **End Condition** as **Blind** and set **Depth** to **1.00mm**.

3 Click **OK**.



To edit the text or its properties, edit the sketch, move the pointer over the text (the pointer changes to), then right-click and select **Properties**. The **Sketch Text** PropertyManager reappears.

Inspect Curvature

Inspect Curvature has been available on the shortcut menu for splines. Now the functionality has been expanded as follows:

- Most sketch entities support **Inspect Curvature**.
- The curvature combs remain visible when a sketch is closed.
- You can turn off the display with a shortcut menu item **Remove Curvature Information**.

To view curvature information:

1 Open a new sketch and sketch a circle, an ellipse, a parabola, an arc, or a spline.

2 Right-click a sketch entity and select **Inspect Curvature**.

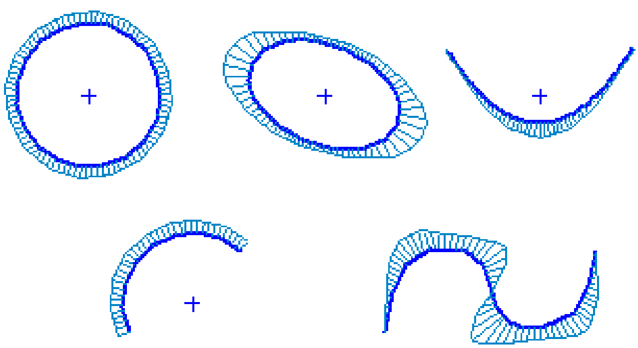
A curvature comb appears.

3 Close the sketch.

The curvature comb remains visible.

4 Right-click the sketch entity and select **Remove Curvature Information**.

The curvature comb disappears.



3D Sketching Relations

You can constrain 3D sketch lines to be tangent to a surface.

The tangency may not be obvious visually as the lines are projected.

To constrain 3D sketch lines to be tangent:

1 Open **sketch_cylinder.sldprt** and open a 3D sketch.

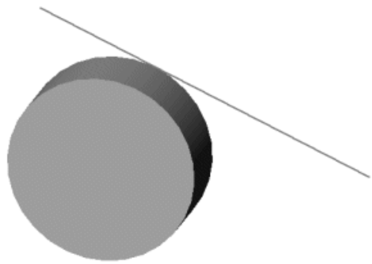
2 Click **Line** and sketch a line.

3 Hold **Ctrl** and select the line and the cylindrical face.

4 In the PropertyManager, under **Add Relations**, select **Tangent**, and click **OK**.

5 Exit the sketch.

The line is tangent to the cylindrical face.



Enhancements to Splines

Simplify Spline

You can now simplify splines that have been created through importing and with tools such as **Convert Entities**, **Offset Entities**, **Intersection Curve**, and **Face Curves**. Such splines could not be modified previously.

Simplifying splines has been made easier with new **Smooth** and **Previous** buttons in the **Simplify Spline** dialog box. When you click **Smooth**, the software adjusts the tolerance and calculates a new curve with fewer points, saving you from choosing a tolerance. A preview of the smoothed curve appears in the graphics area. You can click **Smooth** until you reach the minimum of two points. You can click **Previous** to return through the sequence as far as the original curve.

The tolerance of the current spline is shown in the **Tolerance** box. You can specify the tolerance of the curve as in previous releases of the software, if you prefer.

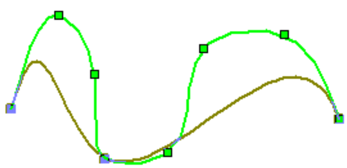
To simplify a spline:

1 Open a sketch and sketch a spline with several points.

2 Right-click the spline and select **Simplify Spline**.

3 In the **Simplify Spline** dialog box, click **Smooth**.

Under **Number of spline points**, the number of points is reported the **In original curve** and **In simplified curve** boxes. The original spline is displayed in the graphics area along with a preview of the smoothed curve.



4 Continue to click **Smooth** until only two points remain.

5 Click **Previous** several times through the sequence to the original curve.

6 Click **OK**.

Two-Point Spline with Tangency

You can now create two-point splines, either with the **Spline** tool or by simplifying a spline with more than two points. You can add end tangency to a two-point spline. When you simplify a spline to two points, the tangency of the end points is retained.

To create a two-point spline and add tangency:

1 In an open sketch, click **Spline** and sketch a two-point spline.

2 Click **Line** and sketch a line that joins an endpoint of the spline.

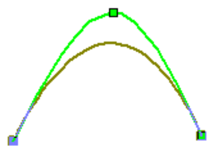
3 Select both the line and the spline.

4 In the **Properties** PropertyManager, under **Add Relations,** click **Tangent** and click **OK**.

To simplify a spline to two points with tangency:

1 In an open sketch, click **Spline** and sketch a three-point spline.

2 Right-click the spline and select **Simplify Spline**.



3 In the **Simplify Spline** dialog box, click **Smooth**.

The spline is smoothed to two points, and the preview shows that tangency is retained at both points.

4 Click **OK**, and exit the sketch.

3

2D to 3D Conversion

A new set of tools for 2D to 3D conversion is introduced in SolidWorks 2001Plus.

- ❑ Introduction
- ❑ Sketch tools
- ❑ Converting a 2D drawing to a 3D part
- ❑ Extracting sketches
- ❑ Aligning sketches
- ❑ Extruding a base feature
- ❑ Cutting features

Introduction

You can now convert imported 2D drawings into 3D models using new conversion tools on the 2D to 3D toolbar and the Sketch Tools menu. Converting a drawing to a part by this method in most cases is faster than constructing the same part by creating a sketch.

NOTE: Even though the sketch used for conversion can be an imported drawing, it must be imported into a sketch in a *part* document. You can copy and paste the sketch from a drawing document, or you can import the drawing directly into a 2D part sketch.

The 2D sketch can be an imported drawing or it can be a sketch constructed in SolidWorks. In either case, you are working from a single sketch in a part document.

When importing a drawing into a part document, open the **.dwg** or **.dxf** file in SolidWorks. In the **DXF/DWG Import** dialog box, select **Import to a new part**, and then select **Import to a 2D sketch**. The drawing appears as **Sketch1**.

When you import DXF/DWG drawings for 2D to 3D conversion, it is recommended that you eliminate crosshatching. See **Crosshatches** on page 10-5.

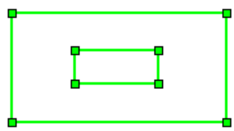
Sketch Tools

The tools described in this section are used in the 2D to 3D conversion process, and they can also be used for sketching in general.

Select Chain

Select Chain has been added to the shortcut menu when selecting sketch entities. This command selects all sketch entities attached to the selected entity in both directions until a branch is encountered. **Select Chain** is especially useful for selecting geometry for conversion from an imported drawing, but it can also be used with sketches created in SolidWorks.

To select a chain of sketch entities:



1 Open a new part and open a sketch.
2 Click **Rectangle** and sketch a rectangle, then sketch another rectangle inside the first.
3 Exit the sketch.
4 Right-click one side of one rectangle and choose **Select Chain**.

   All sides of the rectangle are selected.
5 Hold **Ctrl**, right-click one side of the other rectangle, and choose **Select Chain**.

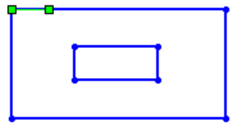
   All sides of both rectangles are selected.
6 Keep the part open for the next procedure.

Repair Sketch

Repair Sketch can often fix errors in a sketch so that the sketch can be used to extrude or cut a feature. Some of the ways it can repair sketches are:

- Resolve overlapping geometry
- Delete small gaps
- Collect small segments into a single entity

To repair a sketch:



1. In the sketch of the rectangles in the previous procedure, click **Line** and sketch a line over one segment of the rectangle.
2. Exit the sketch, then select the sketch in the FeatureManager design tree.
3. Click **Extruded Boss/Base** on the Features toolbar.

   An error message states that the sketch cannot be used for a feature because the endpoint is wrongly shared by multiple entities.
4. Click **OK**.

5. Select the sketch in the FeatureManager design tree again, click **Repair Sketch** on the 2D to 3D toolbar, or click **Tools**, **Sketch Tools**, **Repair Sketch**, and click **Extruded Boss/Base** again.

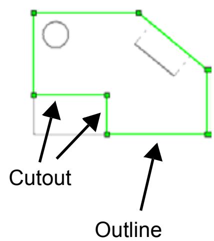
   The **Extrude** PropertyManager appears.
6. Choose a depth for the extrusion and click **OK**.

   Now the rectangle is extruded.

Create New Sketch

You can use **Create New Sketch** to extract only those elements of a sketch (usually in an imported drawing) that you require to create a feature. You can extract a sketch, for example, and then modify it before creating a feature.

To create a new sketch:



1. Open **2Dto3D.sldprt**, which is a drawing imported to a part.
2. Select the edges in the lower left view, as shown.

TIP:	Right-click a line in the outline and choose **Select Chain**. The outline except for the cutout is selected. Then hold **Ctrl** and select the two lines in the cutout.

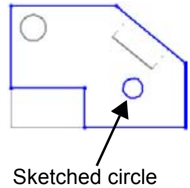
3. Click **Create New Sketch** on the 2D to 3D toolbar, or click **Tools**, **Sketch Tools**, **Create New Sketch**.

   A new sketch appears in the FeatureManager design tree.

4 Select **Sketch2** in the FeatureManager design tree and click **Sketch** on the Sketch toolbar to open the new sketch.

The **Sketch2** entities appear in green.

5 Sketch a circle approximately as shown.

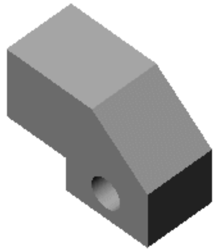


6 Click **Extruded Boss/Base** on the Features toolbar.

7 Set **Depth** to 50mm and click **OK**.

A base part with a hole is extruded.

8 Close the part without saving the changes.



Extrude and Cut on the 2D to 3D Toolbar

The **Extrude** and **Cut** tools on the 2D to 3D toolbar create features by extracting only portions of sketches, creating new sketches before generating the features. The **Extruded Boss/Base** and **Extruded Cut** tools on the Features toolbar, by contrast, use complete sketches.

To extrude a feature in an imported drawing:

1 Open **2Dto3D.sldprt** again.

2 Select the edges as in the previous procedure.

Note that **Extruded Boss/Base** on the Features toolbar is not available.

3 Click **Extrude** on the 2D to 3D toolbar, or click **Tools**, **Sketch Tools**, **2D to 3D**, **Extrude**.

4 In the **Base-Extrude** PropertyManager, set **Depth** to 50mm and click **OK**.

A part is extruded using the selected entities.

5 Close the part without saving the changes.

Converting a 2D Drawing to a 3D Part

The 2D to 3D tools facilitate converting a single sketch into a set of separate sketches, each positioned and oriented as required to create a 3D model. The tools fold up the 2D sketch as if it were a piece of paper.

When you import a drawing for conversion, you specify which portions of the drawing are the sketches for the front view, right view, and so on. You can also create auxiliary sketches that are not parallel to the principal view planes. Hidden lines, which are dashed lines in an imported drawing, are construction lines.

You use the 2D to 3D tools to align sketches with each other, with edges and vertices of a part, or with the origin. You can align sketches before you use them to create a part, or you can align sketches during the part construction process.

The following tools are used to convert a single flat sketch into separate sketches in their correct position and orientation. For example, you select all the sketch segments that belong to the top view and click the **Top** tool to create a new sketch in the top view orientation.

Front
Top
Right
Left
Bottom
Back
Auxiliary

NOTE: You must define a front view before defining any of the other views. When creating an auxiliary sketch, you must select a line in another sketch to specify the angle of the auxiliary view.

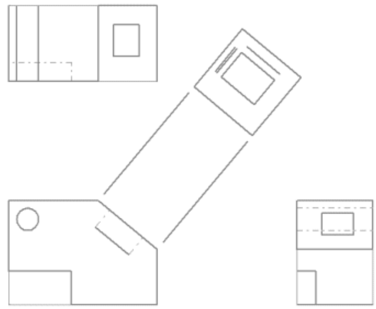
The **Align Sketch** tool is also used during the conversion procedure.

Extracting Sketches

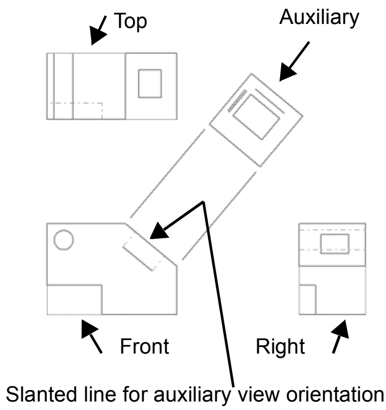
To prepare sketches for constructing a 3D model:

1 Open **2Dto3D.sldprt** again.

NOTE: The drawing has been imported into a part document and is in **Sketch1**.



2 Right-click **Sketch1** in the FeatureManager design tree and select **Edit Sketch**.



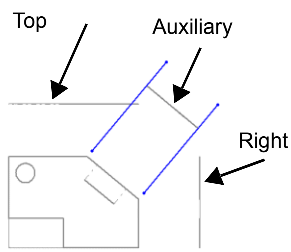
3 Box select the view at the lower left and click **Front** on the 2D to 3D toolbar, or click **Tools**, **Sketch Tools**, **2D to 3D**, **Front**.

The **Front** tool is the only view tool available at first. The front view must be defined before the other views can be selected.

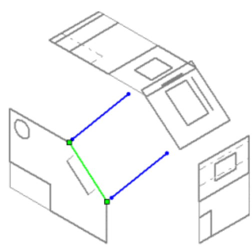
4 Box select the view at the top left and click **Top** on the 2D to 3D toolbar, or click **Tools**, **Sketch Tools**, **2D to 3D**, **Top**.

The view folds down so that from the front, the top view appears as a line.

5 Box select the view at the bottom right and click **Right** on the 2D to 3D toolbar, or click **Tools**, **Sketch Tools**, **2D to 3D**, **Right**.



6 Box select the remaining view at the upper right, hold **Ctrl** and select the slanted line in the front view, and click **Auxiliary**, or click **Tools**, **Sketch Tools**, **2D to 3D**, **Auxiliary**.

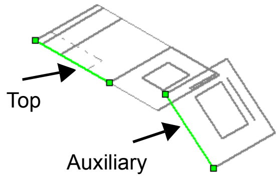


7 Click **Isometric** on the Standard Views toolbar to see the orientation of the sketches.

Aligning Sketches

You align the sketches so that sketch entities are in the correct orientation for creating features.

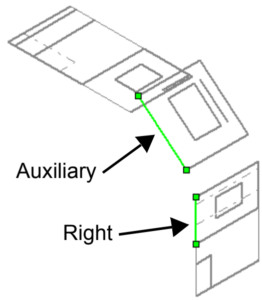
To align the new sketches:



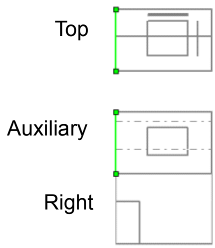
1 Select the left edge in the *auxiliary* view, hold **Ctrl** and select the left edge in the *top* view, and click **Align Sketch** on the 2D to 3D toolbar, or click **Tools**, **Sketch Tools**, **Align**, **Sketch**.

The order of selection is important. The view selected first aligns with the view selected second.

2 Click in the graphics area to clear the selection.



3 Select the left edge in the *right* view, hold **Ctrl** and select the left edge in the *auxiliary* view, and click **Align Sketch** .



4 Click **Right** on the Standard Views toolbar to see that the top, auxiliary, and right views are aligned.

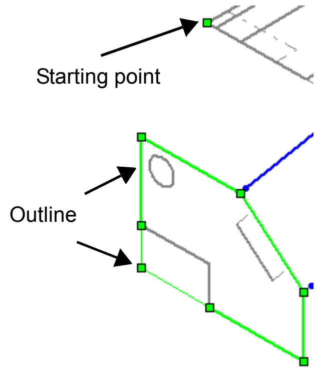
5 Click **Isometric** again.

Extruding a Base Feature

Now the sketches are created and positioned so that you can extrude a base feature where the sketches project.

To extrude the base feature:

1 Select the outline of the front view, as shown.



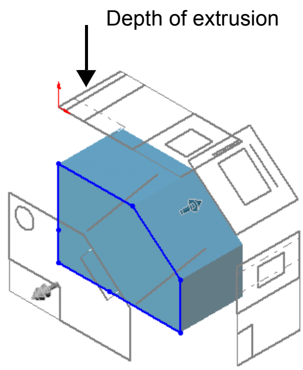
NOTE: The notch will be cut later. Right-click an edge, choose **Select Chain**, then hold **Ctrl** and select the *outside* edges of the notch.

2 Hold **Ctrl** and select a point (in this case, a vertex) on the top view, as shown, as the starting point for the extrusion.

3 Click **Extrude** on the 2D to 3D toolbar, or click **Tools**, **Sketch Tools**, **2D to 3D**, **Extrude**, and click in the graphics area to establish the direction of the extrusion.

4 Click **Depth** in the **Extrude** PropertyManager, select the top left edge in the top view as shown to specify the depth of the extrusion, and click **OK**.

The base feature is created at the center of the sketches.



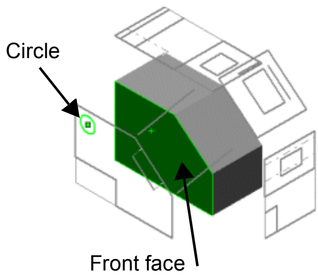
Cutting Features

Now the holes and notch can be cut into the base feature.

To cut features:

1 Select the circle on the front view sketch.

2 Hold **Ctrl** and select the front face of the base to establish where the cut is to start.

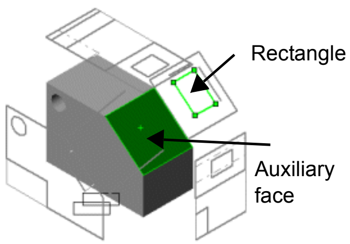


3 Click **Cut** on the 3D to 3D toolbar, or click **Tools**, **Sketch Tools**, **2D to 3D**, **Cut**, and click in the graphics area to establish the direction of the cut.

4 In the **Cut-Extrude** PropertyManager, select **Through All** from the **End Condition** list and click **OK**.

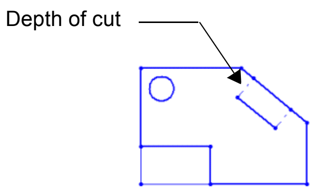
A hole is cut through the base.

5 Select the sides of the rectangle on the auxiliary sketch as shown, hold **Ctrl** and select the angled surface on the base to establish the beginning of the cut. Click **Cut** on the 2D to 3D toolbar, and click in the graphics area to establish the direction of the cut.

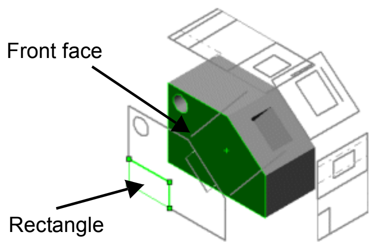


6 In the **Cut-Extrude** PropertyManager, click **Depth**, select the line in the front view sketch as shown, and click **OK**.

A rectangular hole is cut on the angled face of the base to the depth of the selected line.

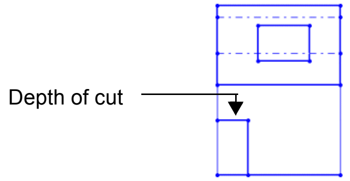


7 Select the sides of the rectangle at the lower left of the front sketch as shown, hold **Ctrl** and select the front face of the base, click **Cut**, and click in the graphics area.

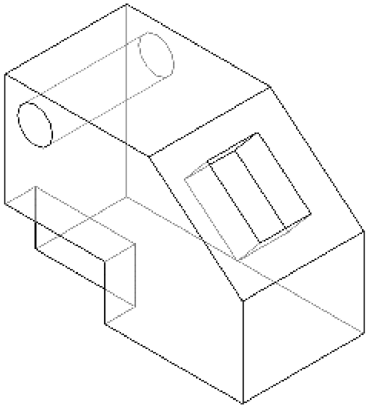


8 In the **Cut-Extrude** PropertyManager, click **Depth**, select the vertical line at the lower left of the right view as shown, and click **OK**.

A notch is cut in the lower left corner of the base to the depth of the selected line.



9 Hide the sketches and click **Hidden in Gray** on the View toolbar to see the cut features.



4

Reference Geometry

This chapter describes the enhancements to Reference Geometry items, including:

- Planes
- Projected curves
- Composite curves
- Split lines

Planes

Plane Creation

All plane creation dialog boxes have moved to the **Plane** PropertyManager. All previous dialog box functionality is retained in the PropertyManager.

New functionality includes the following:

- The **3 Points** and **Line&Point** plane options are combined into one **Through Lines/Points** option in the PropertyManager.
- You can create multiple offset planes by setting the **Number of Planes to Create**. Each plane is created separately using the offset distance from the selected plane.
- When you have selected enough entities to create a plane, the **OK** pointer appears. Right-click to create the plane.
- If you select items before creating the plane, SolidWorks attempts to select the appropriate type of plane. You can always select a different type of plane.
- Some selection capabilities are now also included in the shortcut menu you access by right-clicking in the graphics area.
- You can keep the **Plane** PropertyManager open by clicking **Keep Visible**, which is useful when creating multiple planes of various types.

To create a plane using the new Plane PropertyManager:

1 Open a new part, sketch a circle, and extrude it as a thin feature.

2 Click **Plane** on the Reference Geometry toolbar, or **Insert**, **Reference Geometry**, **Plane**.

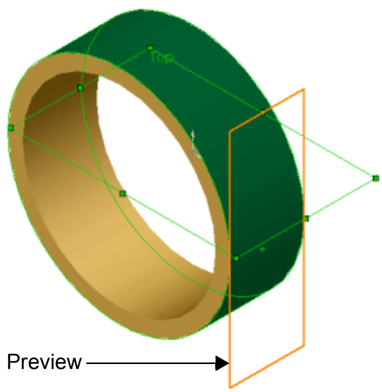
The **Plane** PropertyManager appears.

3 Under **Selections**, click **On Surface**.

4 In the graphics area, select the outer face of the model.

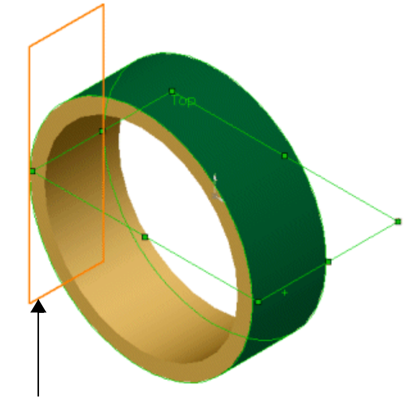
Face<1> and **Top** appear in the **Reference Entities** box. SolidWorks selects the **Top** plane automatically.

A preview of the new plane appears in the graphics area.



5 In the **Plane** PropertyManager, click **Other Solutions**.

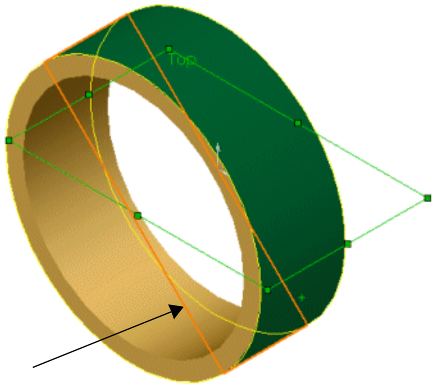
A preview of a different solution for placement of the new plane appears in the graphics area.



Preview of a different solution

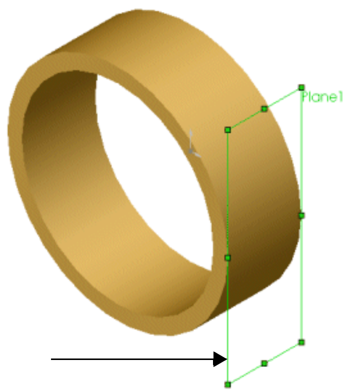
6 Select the **Normal plane** check box.

A preview of a new plane appears that is normal to the **Top** plane, rotated 45 degrees relative to that plane.



7 Click to clear the **Normal plane** check box, click **Other Solutions**, then click **OK** .

The new plane, **Plane1**, appears in the graphics area and in the FeatureManager design tree.



Automatic Sizing of Planes and Axes

Planes and axes you create now automatically size to either the geometry on which they are created, or to the bounding box of the model geometry. As the geometry changes size, the planes and axes update accordingly. You can override the automatic sizing by manually changing the plane or axis size, which disables future autosizing for that entity. You can re-enable autosizing by using the new **Autosize** shortcut menu item.

Previously, plane and axes displayed at a default size that was often too large or too small for the geometry on which they were created.

To see an example of automatic sizing of planes:

1 Open a new part, then create a block that is 10mm x 10mm x 8mm.

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

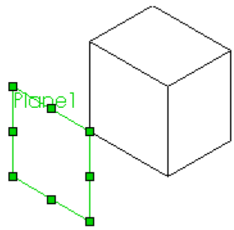
The **Plane** PropertyManager appears.

3 Select the front face of the model.

A preview of the new plane appears.

4 Click **OK**.

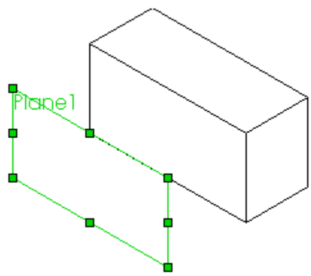
Offset **Plane1** appears and is automatically sized to the geometry of the face from which it was created.



5 Right-click **Base-Extrude** in the FeatureManager design tree and edit **Sketch1** to modify the dimension of the bottom edge of the model from 10mm to 20mm.

6 Exit the sketch.

The **Plane1** size updates automatically when you change the geometry of the face on which it was created.



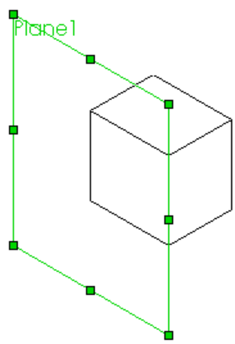
7 Drag upwards to manually resize the top edge of **Plane1**.

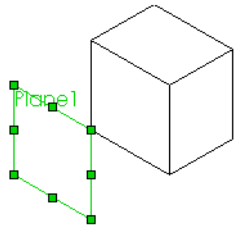
Autosizing becomes disabled.

8 Edit **Sketch1** again to change the bottom edge dimension back to 10mm.

9 Exit the sketch and click **Plane1** in the FeatureManager design tree.

Plane1 does not resize automatically because autosizing is disabled.





10 Right-click **Plane1** in the graphics area and select **Autosize**.

Autosizing is re-enabled. **Plane1** automatically resizes to the updated model geometry.

11 Save the file with the name **Planes.sldprt**. You will use this part in the next section about transparent shaded planes.

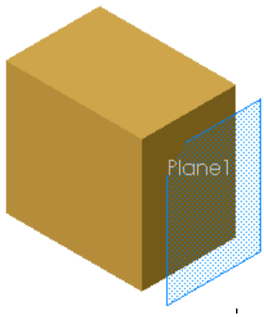
Transparent Shaded Planes

You can now create transparent shaded planes with a wireframe edge that have different front and back colors. You can disable transparency to show just the wireframe edge. You can also change the plane colors, adjust the transparency, and change other plane display options.

Previously, when working with complex layouts for parts and assemblies using multiple planes, there was no visual distinction between different planes, making orientation difficult.

To create transparent planes:

1 Open the part, **Planes.sldprt**, that you created in the previous section.



2 Make sure **Shaded** on the View toolbar is enabled, and select **View**, **Planes**.

Plane1 appears as a colored transparent plane with wireframe edges. Transparent planes are enabled by default.

3 Rotate the model to look at the other side of **Plane1**.

The back face color of **Plane1** is a different color than the front face.

NOTE: You may see only the wireframe plane, depending on the transparency setting. You adjust this setting in **To set the transparent plane colors:** on page 4-6.

To disable and enable transparent planes:

1 Click **Tools**, **Options**. On the **System Options** tab, click **Display/Selection**.

2 Click to clear the **Display shaded planes** check box, then click **OK**.

Plane1 transparency is disabled, leaving visible only the wireframe edges.

3 Click **Tools**, **Options**. On the **System Options** tab, click **Display/Selection**.

4 Select the **Display shaded planes** check box to re-enable transparent planes. Do not close the **Options** dialog box.

To set the transparent plane colors:

1. Select the **Document Properties** tab, then click **Plane Display**.

   You can set the plane colors for faces, edges, and intersecting plane lines.

2. Click **Front Face Color**.

   The **Color** dialog box appears.

3. Change the color, then click **OK**.
4. Click **Back Face Color**, change the color, then click **OK**.
5. Move the **Transparency** slider in either direction to change the transparency.
6. Make sure the **Show intersections** check box is selected (default).

   The edges take the same color as the front and back faces, are not transparent, and are always displayed. You can show or hide plane intersections, and set the intersection line color by clicking **Line Color**.

7. Click **OK**, then rotate the model to see the face color and transparency changes in **Plane1**.
8. Close the part document without saving changes.

Projected Curves

The **Project sketch onto faces** and **Project sketch onto sketch** dialog boxes have been moved to the **Projected Curve** PropertyManager. The previous dialog box functionality is retained in the PropertyManager.

New functionality includes the following:

- You no longer have to pre-select items to enable the **Projection** tool.
- When you have selected enough entities to complete a projected curve, the **OK** pointer appears. Right-click to create the projected curve.
- Some selection capabilities are now also included in the shortcut menu you access by right-clicking in the graphics area.

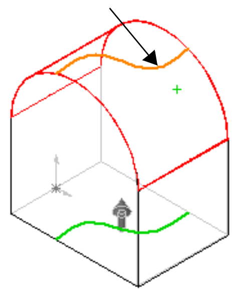
To see an example of the new projected curves functionality:

1. Open **ProjectedCurve.sldprt**.
2. Click **Projection** on the Curves toolbar, or **Insert**, **Curve**, **Projected**.

   The **Projected Curve** PropertyManager appears.
3. Under **Selections**, do the following:
   - Set **Projection Type** to **Sketch onto Face(s)**
   - Select the **Reverse Projection** check box
   - Select the curve in the graphics area to place **Sketch 2** in the **Sketch to Project** box
   - Select the **Projection Faces** box, then select the cylindrical face on the model to set **Projection Faces** to **Face<1>**

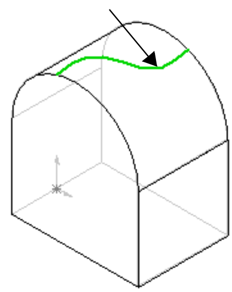
NOTE: If you select the cylindrical face before selecting the **Reverse Projection** check box, a message box appears. Click **OK**, then select the **Reverse Projection** check box.

A preview of the projected curve appears.



4. Click **OK**.

   The curve is projected onto the selected face.



Composite Curves

The **Composite Curve** dialog box has been moved to the **Composite Curve** PropertyManager. The previous dialog box functionality is retained in the PropertyManager.

New functionality includes the following:

- When you have selected enough entities to complete a composite curve, the **OK** pointer  appears. If you have selected all the desired entities, right-click to create the composite curve.
- Some selection capabilities are now also included in the shortcut menu you access by right-clicking in the graphics area.

Split Lines

The **Split Lines** dialog box has been moved to the **Split Lines** PropertyManager. The previous dialog box functionality is retained in the PropertyManager.

New functionality includes the following:

- When you have selected enough entities to create a split line, the **OK** pointer  appears. Right-click to create the split line.
- Some selection capabilities are now also included in the shortcut menu you access by right-clicking in the graphics area.

5

Features

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. Changes include the following:

- Migrations to the PropertyManager
- Shaded previews for lofts and sweeps
- Faces to pattern for mirror functions
- Pattern along a planar curve
- Curvature continuous fillets
- Control points for variable radius fillets
- Fillet by feature
- Loft add section
- Loft side tangency
- Translate surface option with offset from surface extrude
- Hole Wizard on non-planar faces
- Delete faces on a solid
- Draft analysis tool

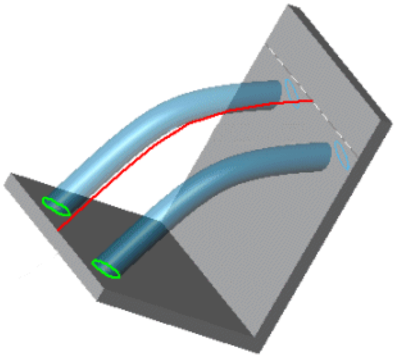
PropertyManager Interface

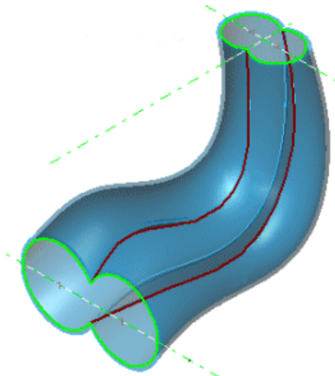
The following functions have moved from dialog boxes to the PropertyManager. All previous dialog box functionality is retained in the PropertyManager.

- Mirror Features
- Mirror All
- Scale

Shaded Previews for Lofts and Sweeps

Shaded previews were added for sweeps and lofts.





Pattern Features

Two new methods of implementing patterns were added for SolidWorks 2001Plus. The methods include:

- **Faces to Pattern**. Allows you to select the faces as opposed to the feature when specifying a mirror pattern.
- **Pattern along a planar curve**. Allows you to use any sketch, curve, or edge which lies in a plane to define the pattern.

Faces to Pattern

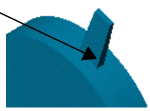
You can now select **Faces to Pattern**, as well as **Features to Pattern**, with the following:

- **Curve Pattern**
- **Linear Pattern**
- **Circular Pattern**
- **Mirror Feature/Face**
- **Sketch Driven Pattern**
- **Table Driven Pattern**

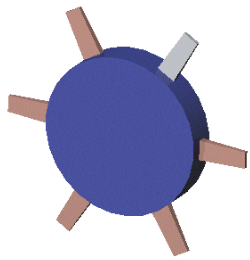
You can use **Faces to Pattern** with imported geometry when, as a result of the import process, an entity you want to mirror has not created a feature that is recognized by SolidWorks. If the imported entity displays the faces, you can select the faces instead of the feature. Using **Faces to Pattern**, it patterns the feature in SolidWorks.

You can also apply **Faces to Pattern** to parts created with SolidWorks.

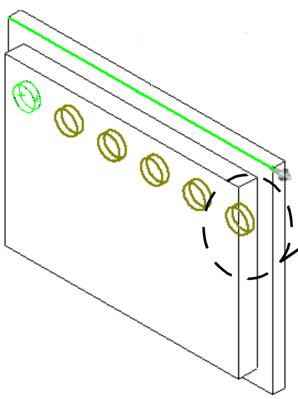
Select all the faces in the **Faces to Pattern** box



Create a circular pattern.

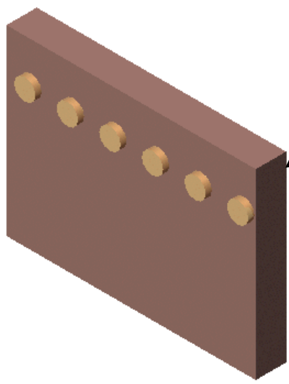


NOTE: The pattern must remain within the same face or boundary. It cannot cross boundaries. For example, a cut across the entire face or different levels (such as a raised edge) would create a boundary and separate faces.



Pattern Fails

Pattern is not within the same boundary because the pattern instance does not rest on a single boundary.

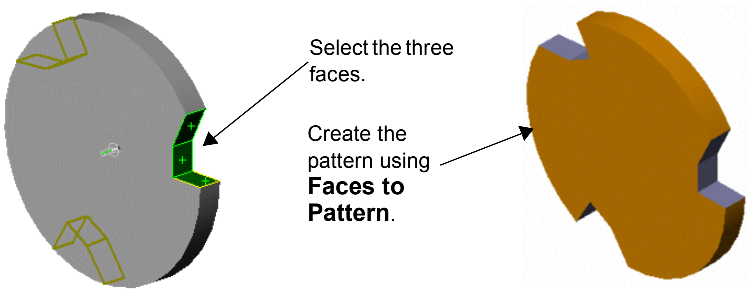


Pattern Succeeds

Pattern is within the same boundary

To create a pattern using Faces to Pattern:

1. Open **mirror_circular_face_pattern.sldprt**.
2. Click **Circular Pattern** , or **Insert, Pattern/Mirror, Circular Pattern**.
3. Under **Parameters**, do the following:
   - **a)** Select the axis in the graphics area as the **Pattern Axis**.
   - **b)** Select the **Equal spacing** check box.
   - **c)** Set the **Number of Instances** to 3.
4. Under **Faces to Pattern** , select the three faces of the cut, as shown.



5. Click **OK** .

Curve Driven Pattern

The **Curve Driven Pattern** feature allows you to create patterns along a planar curve in much the same way as you create linear and sketch driven patterns. You can use any sketch segment, edge, solid or surface that lies along the plane to define the pattern.

Like other pattern types, you can select to skip pattern instances, and pattern in one or two directions. Elements specific to curve driven patterns include the following:

- Type of curve
- Curve method
- Alignment method

Type of Curve

The type of curve you use to create your pattern determines how you specify the number of instances to pattern for both directions. You can base your pattern on an open curve or a closed curve (such as a circle).

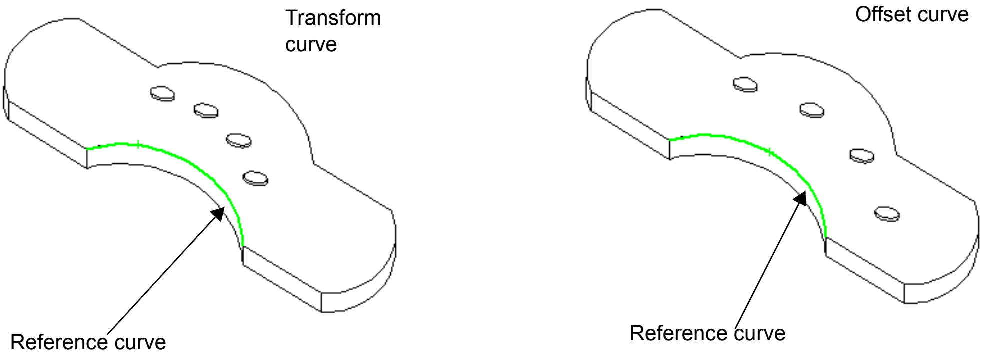
Direction 1. You can select to equally space the pattern instances by applying the **Equal spacing** option. Alternatively, you can specify a step distance along the curve between each pattern instance. The distance between the curve or sketch segment and the **Feature to Pattern** is measured normal to the curve.

Direction 2. For **Direction 2**, you can also apply the **Equal Spacing** option or specify step distances. If you use a closed curve, it allows you to create patterns of concentric rings or rows of identically patterned features.

Implicit Direction 2. If you select the **Direction 2** check box without specifying a sketch element or edge, an implicit pattern is created. The implicit **Direction 2** is based on what you specified for **Direction 1**.

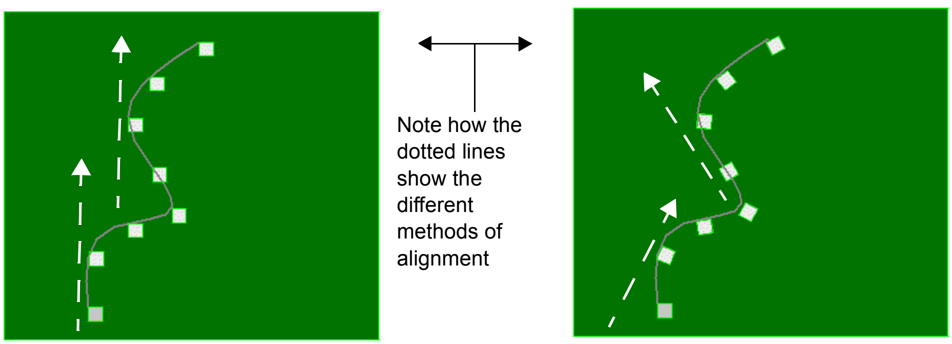
Curve Method

You can define the direction of the pattern you create by transforming how you use the reference curve (an edge, sketch, or curve) on which your pattern is based. The default method is **Transform curve**, in which the feature you pattern follows the shape of the reference curve. Or you can use **Offset curve**. This causes the feature you pattern to be offset from the reference curve.



Alignment Method

You can select to align the features to the original alignment of the seed feature. Alternatively, you can select to align the features tangent to the direction reference.



Alignment method: Align to seed

Alignment method: Tangent to curve

To create a curve pattern:

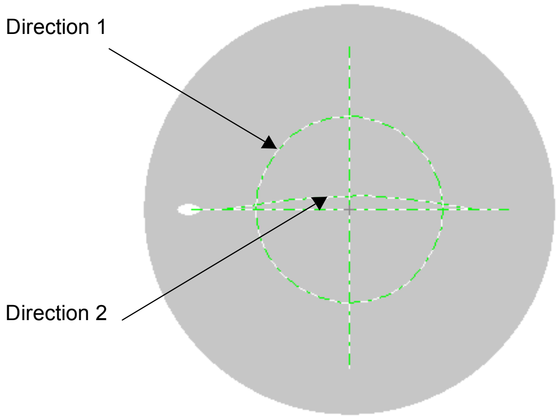
1 Open **curve_pattern.sldprt**.

2 Click **Curve Driven Pattern** on the Features toolbar, or **Insert, Pattern/Mirror, Curve Driven Pattern**.

The **Curve Driven Pattern** PropertyManager appears.

3 Under **Direction 1**, do the following:

a) In the graphics area select the construction circle, as shown.



b) Select **Number of Instances** and enter 8.

c) Select the **Equal spacing** check box.

d) For the **Curve method**, select **Offset curve**.

e) For the **Alignment method**, select **Align to seed**.

4 Under **Direction 2**, do the following:

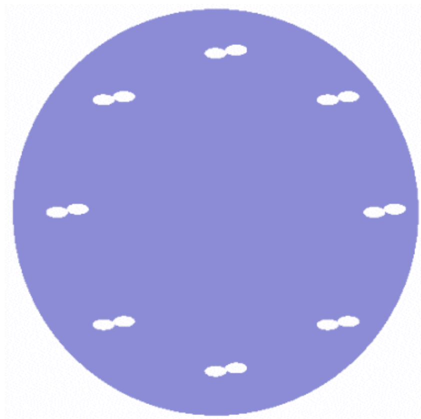
a) In the graphics area, select the construction spline, as shown.

b) Select **Number of Instances** and enter 2.

c) Set the **Spacing** to 12mm.

5 Select **Features to Pattern**, and using the flyout FeatureManager design tree, select **Cut-Extrude1**.

6 Click **OK**.



Fillet Features

Fillet features include the following enhancements:

- **Curvature Continuous fillets**. You can use the **Curvature Continuous** option with the **Face fillet** feature to achieve a smooth curvature between adjoining faces.
- **Control points**. With **Variable radius fillets**, you have the option to apply radius values to user-defined points along the selected edge, as well as to the vertices.
- **Fillet by feature**. You can create edge fillets by selecting the feature that the edge belongs to.

Curvature Continuous Fillets

The **Curvature Continuous** option for fillets resolves discontinuity problems when you apply a face fillet between adjacent surfaces. The **Curvature Continuous** option creates a smoother curvature between the adjacent faces.

You can use **Curvature Continuous** with or without the **Hold line** option. When you use **Hold lines** and the **Curvature continuous** options together, you can select a hold line for each face.

Zebra Stripes is an option that displays imperfections between faces. Used in conjunction with the **Curvature Continuous** fillet option, you can verify whether two adjacent faces are in contact, are tangent, or have curvature continuity.

NOTE: See **Checking a Surface Boundary with Zebra Stripes** on page 6-5 for analyzing curvature on a surface.

To use the Curvature Continuous option:

1 Open **fillet_curvature_continuous.sldprt**.

2 Click **Fillet**, or **Insert, Features, Fillet/Round**.

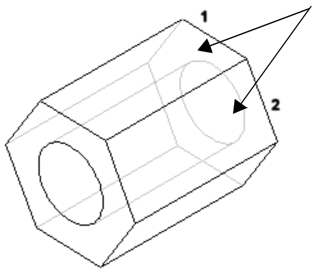
3 Under **Fillet Type**, click **Face fillet**.

4 Under **Items to Fillet**, do the following:

   a) Set the **Radius** to 40mm.

   b) Click **Face Set 1**, and select the first face in the graphics area, as shown.

   c) Click **Face Set 2**, and select the second, adjacent face, as shown.



5 Under **Fillet Options**, select the **Curvature Continuous** check box.

6 Click **OK**.

To see the difference that **Curvature Continuous** option makes to the surface, you need to toggle **Zebra Stripes** on.

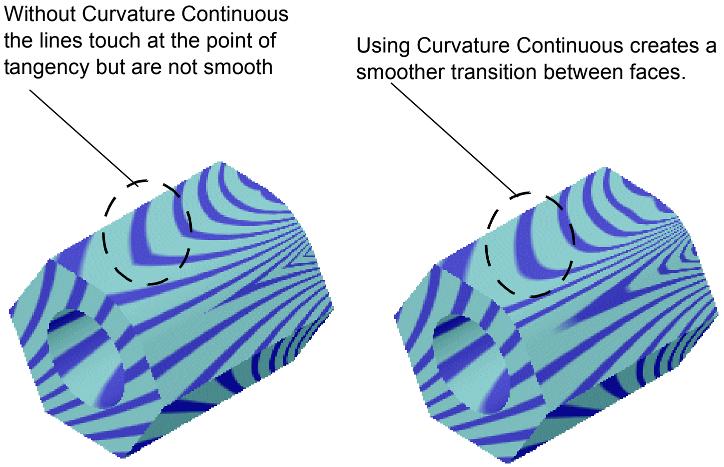
NOTE: If you are in the **Wireframe**, **Hidden in Gray**, or **Hidden Lines Removed** views, and you toggle **Zebra Stripes** on, the view automatically changes to **Shaded**.

To use Zebra Stripes to view curvature continuous fillets:

1 Click **View, Display**, **Zebra Stripes**.

The **Zebra Stripes** PropertyManager appears, where you can control the number of stripes, their width, and the colors.

2 Click **OK** ✓.



Variable Radius Control Points

With variable radius fillets, you can assign different radii to control points along the edge you select to fillet. Variable radius control points operate as follows:

- The system defaults to three control points, located at equidistant increments of 25%, 50%, and 75% along the edge between the two variable radii.
- You can change the relative position of any control point by changing the percentage of that control point.
- You can add or subtract control points along the edge you select to fillet. Adding or subtracting control points along the edge positions the control points in equidistant increments along the edge you select.

To use variable radius control points:

1 Sketch a box and extrude so that it resembles the part below (60 x 80 x 140).

2 Click **Fillet**, or **Insert, Features, Fillet/Round**.

3 Under **Fillet Type**, select **Variable radius**.

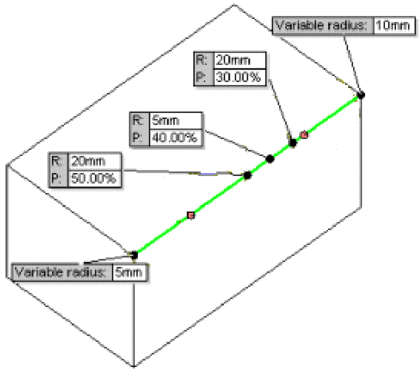
4 Under **Items to Fillet**, select one of the long edges in the graphics area, as shown.

5 Under **Variable Radius Parameters**, do the following:

a) Click **V1** in the **Attached Radii** box, and change the value in **Radius** box to 10mm.

b) Click **V2**, and change the value to 5mm

NOTE: You must set the radius values for **V1** and **V2** before you set any control point values.



6 In the graphics area, select each control point to assign a radius (**R**), and to change the default position (**P**) of the control points (pre-set at 25%, 50%, and 75%). Assign values as follows:

a) Control point 1 (closest to **V1**), **R** equals 20mm, and **P** equals 30%.

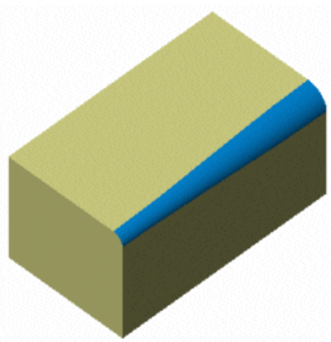
b) Control point 2 (middle), **R** equals 5mm, and **P** equals 40%.

c) Control point 3 (closest to **V2**), **R** equals 20mm, and **P** equals 50%.

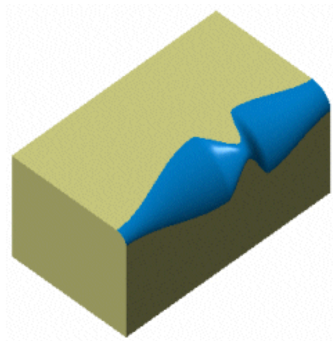
NOTE: To change the number of control points between the radii, use the arrows or enter a new value in the box.

---

7 Click **OK**.



Variable radius applied without specifying control points.



Variable radius applied after specifying radius (**R**) and position (**P**) of control points.

Fillet by Feature

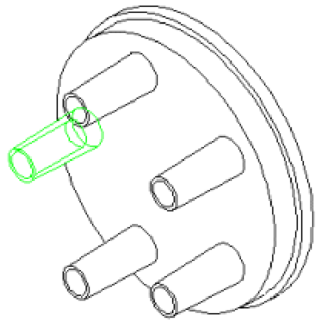
You can create edge fillets by selecting features that edges belong to. The **Fillet** PropertyManager displays the features in the **Edges, Faces, Features, and Loops** box.

Select the features using the flyout FeatureManager design tree, or move the pointer over the face and use **Select Feature** from the shortcut menu.

To fillet by feature:

1 Open **fillet_by_feature.sldprt**.

2 In the FeatureManager design tree, hold down **Ctrl** and select **Boss-Extrude2** and **Cut-Extrude1** to identify the seed features for the circular pattern, as shown.



3 Click **Fillet**, or **Insert, Features, Fillet/Round**.

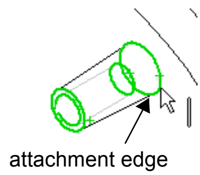
Note that both **Boss-Extrude2** and **Cut-Extrude1** are listed in the **Edges, Faces, Features, and Loops** box.

4 Under **Items to Fillet**, set the **Radius** to 0.50mm.

5 In the graphics area, move the pointer over any of the circular pattern instances, and right-click to open the shortcut menu.

6 Click **Select Feature**.

7 Move the pointer over the edge of the **Boss-Extrude1** feature as shown, and right-click to open the shortcut menu.

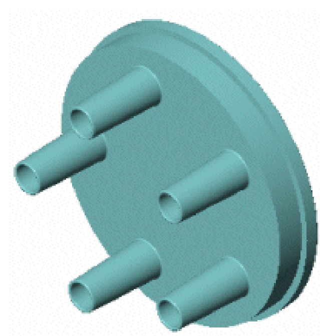


8 Click **Select Loop**.

The fillet extends to the edge of the boss feature, and **Loop1** is listed in the **Edges, Faces, Features, and Loops** box.

NOTE: If the feature you want to fillet is an attachment, you can omit the fillet from the attachment edges. Under **Fillet Options, Feature attachment**, click the **Omit attach edges** check box.

9 Click **OK**.



Loft Add Section

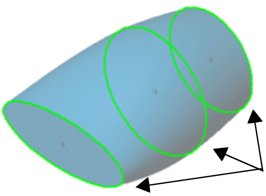
Using the shortcut menu, you can add new loft sections and position them in an existing lofted body. When you use **Add Loft Section**, it creates a sketch (the loft section) along with a temporary plane. You can position the plane in the following ways:

- Drag the plane and the loft section along the axis of the loft. The pointer changes to .
- Rotate the plane around the loft to adjust the angle. The pointer changes to .

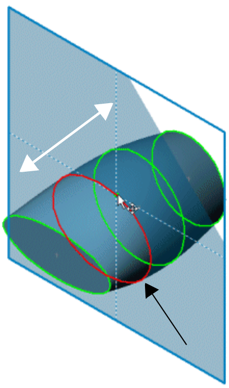
Dragging and rotating the plane along the loft can provide an analysis tool to inspect the curvature of the loft in specific regions of the surface.

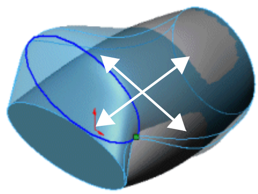
Using the shortcut menu, you can also select **Edit Loft Section** while you position the plane to edit the sketch using dimensions, add relations, and so on. Or you can create your own plane before adding the loft section, and select to use the plane for the new loft section. Selecting your own plane provides the full capabilities of **Edit Loft Section** to the loft section.

NOTE: You should not use **Add Loft Section** with a loft that includes guide curves.



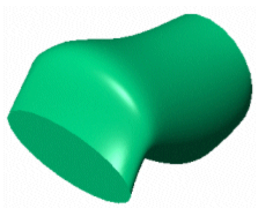
Start with a three section loft.





Use **Edit Loft Section** to position the new section by dragging the sketch, as shown. Depending on the type of sketch created, you can also position and modify the sketch using relations, dimensions, and so on.

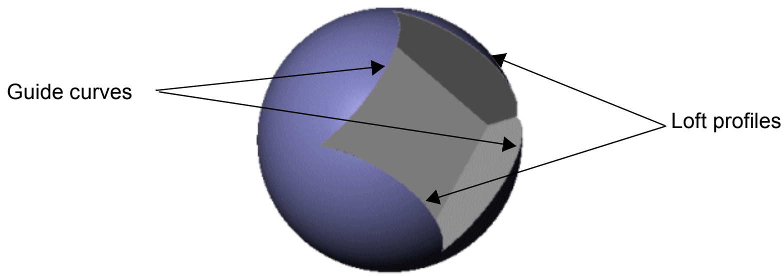
Use **Add Loft Section** to create a new sketch (loft section). You can position the plane with the new loft section by dragging the center point, as shown. You can also rotate the plane by selecting its edges.

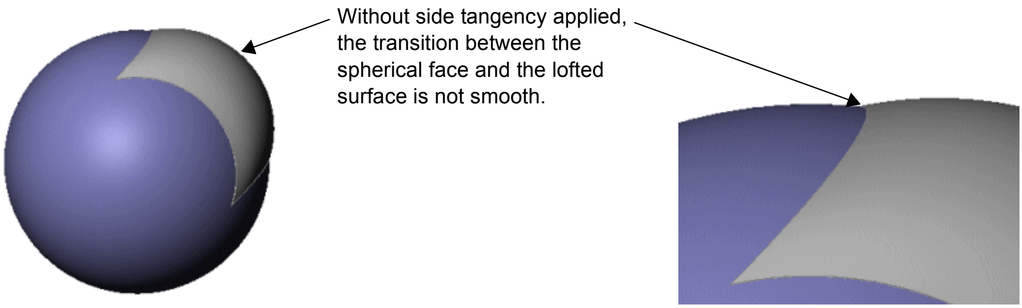


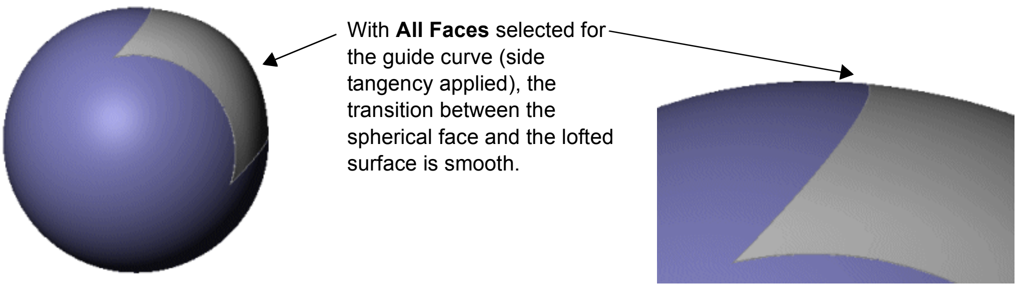
Four section loft with new loft section repositioned.

Loft Side Tangency

When creating lofts with guide curves, you can select **All Faces** (as opposed to **None**) as the **Guide tangency type.** When you select **All Faces**, it adds side tangency between adjacent faces that lie along the path of the guide curve. This creates a smoother transition between the adjacent faces. This option is available with solids or surfaces.The following is a surface loft between two edges to patch a square hole in a sphere.







NOTE: For best results when using side tangency, the profiles should also be tangent to the tangency faces at the point where each profile meets the guide curve. Ideal tolerance is 2 degrees or less. You can use profiles with connection points up to 30 degrees off tangency before the loft fails.

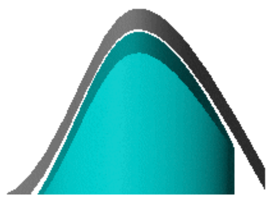
Offset from Surface

A new check box option, **Translate surface** in the **Extrude** PropertyManager, is available when you select **Offset from Surface** as an end condition during extrusions.

- **Translate surface** *Cleared*. The end of the extrusion is a true offset of the reference surface.
- **Translate surface** *Selected*. The end of the extrusion is a simple translation of the reference surface.

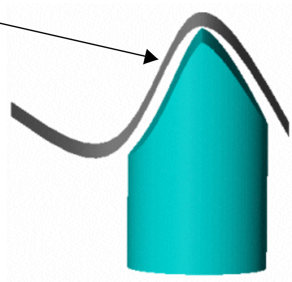
NOTE: Any features created prior to the SolidWorks 2001Plus release show the option selected. You can toggle the option.

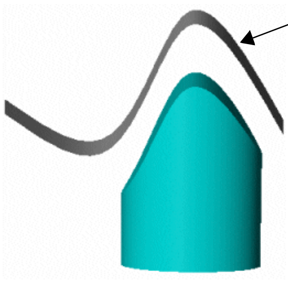
With **Translate surface** *cleared*, the surface of the extrusion is a true offset.



With **Translate surface** *checked*, the surface is translated by the offset distance from the reference surface.

Translate surface *cleared*, with an offset distance of 7mm





Translate surface *checked* with an offset distance of 27mm

TIP: Checking **Translate surface** sometimes allows you to create offsets at a greater distance than you could when using a true offset. Using a true offset produces errors when the offset is large enough to create self-intersecting geometry.

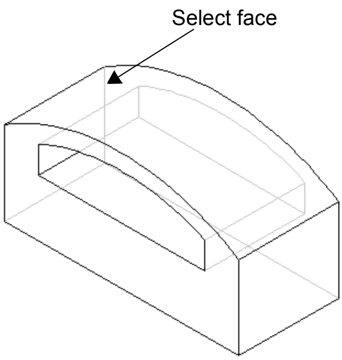
Hole Wizard Holes on Non-Planar Surfaces

Prior to SolidWorks 2001Plus, placing holes on non-planar surfaces using the **Hole Wizard** required creating a separate plane. In SolidWorks 2001Plus, you can create **Hole Wizard** holes directly on a non-planar surface. If necessary, you can also pattern the feature on the non-planar surface, or create multiple holes on different faces as part of the same feature.

To create Hole Wizard holes on a non-planar surface:

1 Open **hole_wiz_non_planar.sldprt.**

2 In the graphics area, select the face, as shown.



3 Click **Hole Wizard**, or **Insert, Features, Hole, Wizard**.

4 In the **Hole Definition** dialog box, click the **Countersink** tab.

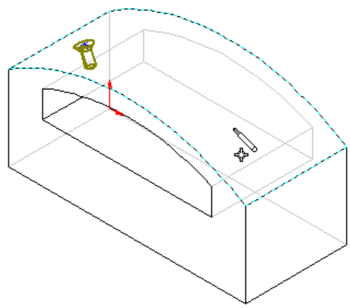
5 Select the following:

- **a)** **Standard**. Select **Ansi Inch**.
- **b)** **Screw Type**. Select **Flat Head (82)**.
- **c)** **Size**. Select **#8**.

6 Set **End Condition & Depth** to **Up to Next**.

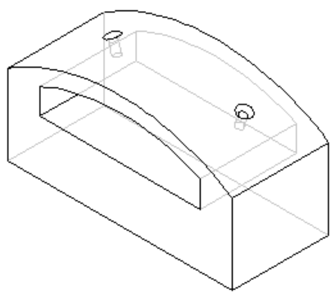
7 Click **Next**.

8 Use the pointer to create a second **Hole Wizard** instance, adjacent to the first instance, as shown.



9 Position the two **Hole Wizard** instances by dragging the points or by using **Dimension** and **Add Relations**.

10 Click **Finish**.

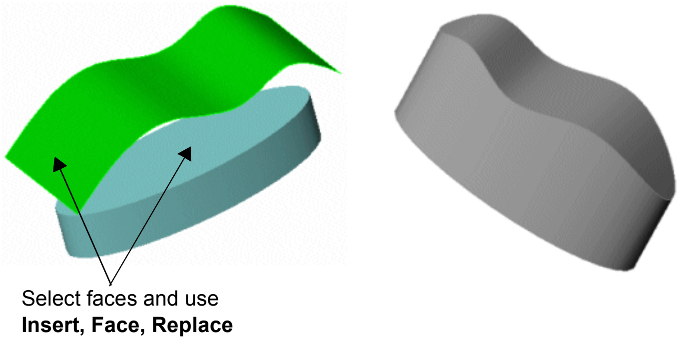


Delete Faces on Solids

You can select faces on a solid and delete the faces (if the faces are replaced). **Delete Faces** fills the gap by extending the faces around the hole until the hole is covered. You can apply **Delete Faces** to patch faces created by various features including:

- Fillets
- Cuts
- Chamfers

You can also use replace faces on a surface body to repair surfaces, or replace a face on a solid body. For details on the procedure, see **Replacing a Face in a Surface Body** on page 6-8.

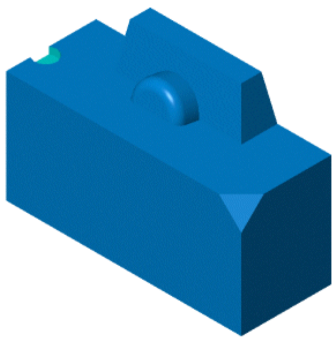


To delete faces on solids:

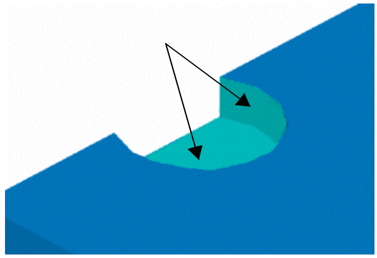
1 Open **delete_faces_solid.sldprt**.

2 Click **Insert, Face, Delete**.

The **Delete Face** PropertyManager appears.



3 In the graphics area, select the two faces that make up the circular cut on the left side, as shown.



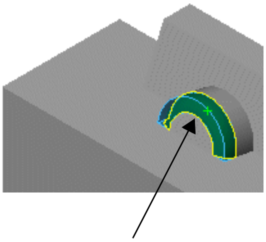
Under **Selections**, **Face1** and **Face2** appear in the **Faces to Delete** box.

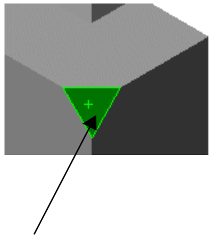
4 Under **Options**, select **Delete and Patch**.

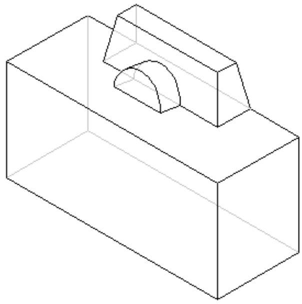
5 Click **OK**.

The cut is filled.

6 Repeat the process to fill the fillet and the chamfer features, as shown (you must do each feature individually).







Draft Analysis

Useful to mold designers and plastic part designers, the **Draft Analysis** tool is formulated to implement mold analysis functions. You can use this analysis tool to help correct draft angles, as well as locate parting lines, injection, and ejection surfaces in parts.

A draft analysis is based on the following criteria that you specify:

- **Direction of Pull**. Select the plane used to conduct the draft analysis.
- **Reverse Direction** . Change the direction of the draw normal to the **Direction of Pull** (plane) you selected.
- **Draft Angle** . Enter a reference draft angle, and then compare that reference angle to those currently existing on the model.

Types of Analysis

When you calculate the draft angles on the model, you can display two different types of data.

- **Face-based**. the analysis generates face colors based on the category of draft (positive draft, negative draft, draft required, or a combination of positive and negative draft). To conduct a face-based analysis, select the **Face classification** check box.
- **Contour map**. The analysis generates a contour map of the face angles, with either uniform or graduated colors.

Categories for Draft Analysis

When you specify a face-based draft analysis, the results are based on four color-coded categories which analyze the type of draft on each face. The four categories that identify the type of draft are as follows:

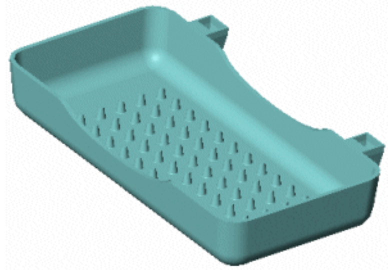
- **Positive Draft**. Display faces with a positive draft direction whose draft angle is equal to or greater than the reference angle. A face with a positive draft direction has a normal towards the direction of pull.
- **Negative Draft**. A face with a negative draft direction has a normal opposite to the direction of pull.
- **Draft Required**. Displays faces with less than the reference draft angle.
- **Straddle Faces**. Displays faces that contain both positive and negative types of drafts. **Straddle Faces** applies only to a face-based analysis.

Each category also displays the number of faces based on the reference and the direction of pull. You can suppress each category using **Show** or **Hide** .

You can also change the default display color for each category.The next time you perform a draft analysis, the colors you specified are the default.

After you calculate the draft angles using the face-based draft analysis, you can save the model with the color classifications.

To apply draft analysis:



1 Open **draft_analysis_soapdish.sldprt**.

2 Click **Tools, Draft Analysis**.

The **Draft Analysis** PropertyManager appears.

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

a) Select the **Front** plane as the **Direction of Pull**.

b) Note the draw direction. To change the draw direction, click **Reverse Direction** .

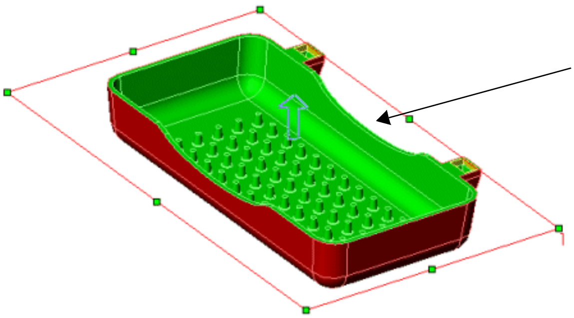
c) Enter a **Draft Angle** of 3 degrees.

d) Select the **Face classification** check box.

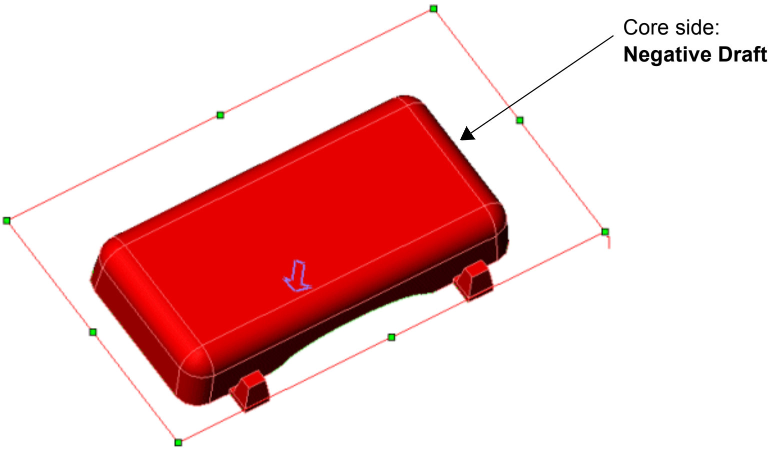
e) Click **Calculate**.

4 Examine the results.

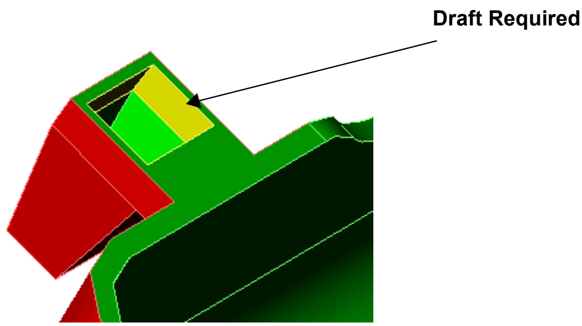
a) The cavity side displays the **Positive Draft,** meaning the angle of the face, with respect to the direction of pull, is more than the reference angle of 3 degrees.



b) Rotate the model, and note the other side of the tooling is a different color. This indicates a **Negative Draft**.



c) Rotate the model and zoom into the soap dish holders, as shown. Note the different color, indicating **Draft Required**. This means no draft was applied, or the draft applied was less than the 3 degrees you specified.



5 Click **OK** ✔ in the PropertyManager and click **OK** if you want to save the model with the colors generated by the draft analysis.

6

Parts and Surfaces

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:

- Split part
- Section view selection
- Zebra stripes
- New functions in PropertyManager
- Mirror surfaces
- Delete face
- Replace face

Parts

Split Part

Using **Split Part**, you can create multiple parts from an existing part. For example, you can split a weldment into its constituent parts, or divide a molded part into smaller pieces. You can automatically create an assembly of the new parts, or create separate part files.

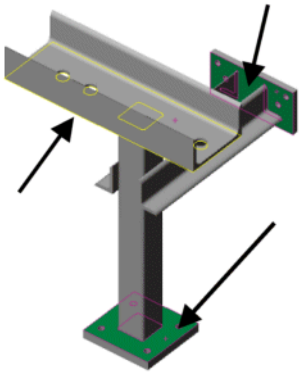
The new parts are similar to derived parts. They contain references to the original part which you can keep or break as needed. If you keep the references, the new parts update as the original part changes.

To split a part:

1. Open **weldment.sldprt.**
2. Click **Split**  on the Features toolbar, or click **Insert**, **Features**, **Split**.

   The **Split** PropertyManager appears.
3. Select the underside of the channel and the two other faces shown to define the planes on which the part is split.

   The selections appear in the **Trim tool** list.

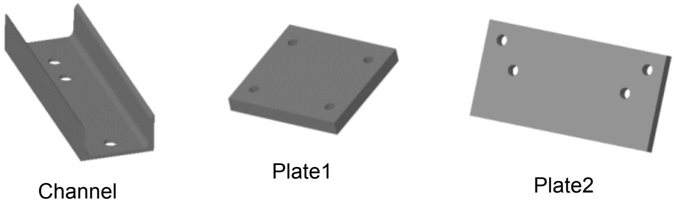
   
4. Click **Cut Part**.

   Split lines appear on the part, showing the four different bodies formed by the split. Callouts appear for the different bodies. As you move the pointer over the part in the graphics area, the bodies are highlighted.
5. Select the channel.
6. The **Save As** dialog box appears.
7. Type **Channel** for the new part and click **Save**.

   The new part name appears in the **Resulting Bodies** list and in the callout box in the graphics area. The body appears yellow in the graphics area.
8. Select each of the two small plates and name them **Plate1** and **Plate2**.

9 Click **OK** .

New part documents appear under the current document for each of the new parts. You do not need to save these parts again since they were saved during splitting. You can examine each part as needed, then close the windows.



10 Examine **Weldment.sldprt**.

The FeatureManager design tree contains all of its original features plus a new feature called **Split1**.

The solid body displayed in the graphics area is the original solid body minus the new parts.

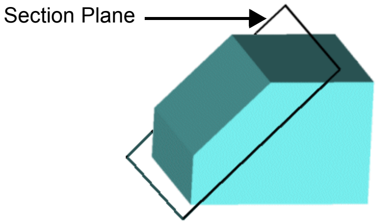
NOTE: It is possible to save all the split bodies during a split, in which case no solid body would be displayed in the original part.

If you delete the split feature in the original part, the new parts still exist, but the status of the external reference is dangling.

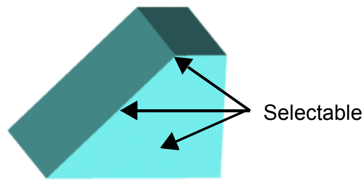
TIP: To see the original solid body, move the rollback bar in the FeatureManager design tree above the split feature, or suppress the **Split** feature.

Section View Selection

You can now select the faces, edges and vertices that are created by a section view of a part or assembly. Previously, the section view was for visualization only, and you could not select faces or edges that were cut by the section plane.

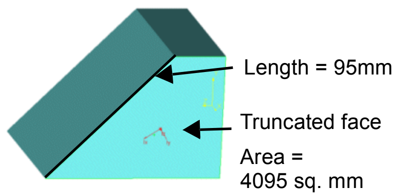


Section View Inactive

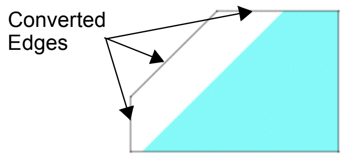


Section View Active

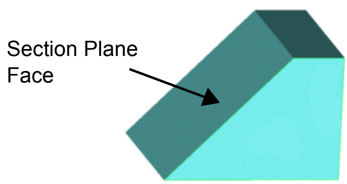
For example, you can select a truncated face and measure the length of its edges, or check its section properties (for example, area).



For functions other than **Measure** and **Section Properties**, the truncated faces act as if they are not truncated. For example, if you open a sketch on a truncated face and convert the edges, the edges of the full face are converted.



You can select faces on the section plane, but you cannot open a sketch on those faces.



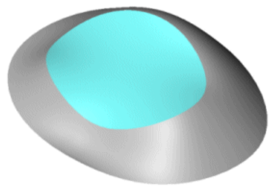
Checking a Surface Boundary with Zebra Stripes

Zebra Stripes is a type of display that allows you to see small changes in a surface that may be hard to see with a standard display. It simulates the reflection of infinitely long strip lights on a very shiny surface. With **Zebra Stripes**, you can easily see wrinkles or defects in a surface, and you can verify that two adjacent faces are in contact or are tangent.

For more information on **Zebra Stripes**, see **Zebra Stripes** on page 1-8, and **Curvature Continuous Fillets** on page 5-7.

To activate Zebra Stripes:

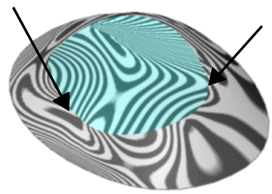
1 Open **surface_zebra_stripes.sldprt**. The light blue center patch is a **Surface Fill**.



2 Click **View**, **Display**, **Zebra Stripes**.

The **Zebra Stripes** PropertyManager opens, and the part displays a black and white striped pattern.

Notice that the stripes do not match at the boundary of the center patch. This indicates contact at the boundary, with no tangency or curvature continuity.



3 Rotate the part to view the changing reflection on the surfaces.

4 Click **OK**.

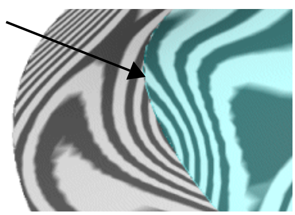
5 Right-click **Surface-Fill2** and select **Edit Definition**.

The **Surface-Fill** PropertyManager appears.

6 Under **Patch Boundary**, change from **Contact** to **Tangent**.

7 Click **OK**.

The stripes now match at the boundary indicating tangency. The slopes of the stripes do not match indicating curvature discontinuity.



Surface Features

PropertyManager Interface

The following surface features now use the PropertyManager:

- Boss/Base Thicken
- Knit Surface
- Surface Cut
- Offset Surface
- Planar Surface
- Radiate Surface

All previous dialog box functionality is retained in the PropertyManager.

Move/Copy Surfaces

You can now move, rotate, and make copies of surfaces.

To copy a surface in a linear pattern:

1 Open **surface_move.sldprt**.

2 Click **Move/Copy Surface** on the Surfaces toolbar, or **Insert**, **Surface**, **Move/Copy**.

The **Move/Copy Surface** PropertyManager appears.

3 Under **Surfaces to Move/Copy**:

a) Select the surface in the graphics area.

Surface Body<1> appears in the **Surfaces to Move/Copy** list.

b) Select the **Copy** check box.

c) Enter 3 for **Number of Copies**.

4 Under **Translate**, do the following:

a) Click to highlight the **Translation Reference** box, then click the edge shown.

b) Enter 40mm for **Distance** using the spin box.

Three copies of the surface appear at 40mm intervals along the direction specified by the selected edge.

5 Click **OK**.

The **Surface-Move/Copy** feature appears in the FeatureManager design tree.

To move a surface:

1 Click **Move/Copy Surface**, or **Insert**, **Surface**, **Move/Copy**.

2 Under **Surfaces to Move/Copy**, select the last surface from the previous procedure.

3 Under **Translate**, do the following:

a) Click to highlight the **Translation Reference** box, then click the same edge used in the previous procedure.

b) Enter 40mm for **Distance** using the spin box.

The surface moves 40mm. The original position of the surface is visible.

4 Click **OK**.

The surface appears in the new position.

To copy surfaces in a circular pattern:

1 Click **Move/Copy Surface**, or **Insert**, **Surface**, **Move/Copy**.

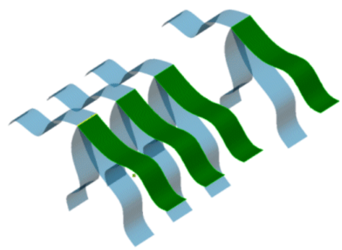
2 Under **Surfaces to Move/Copy**:

a) Select all the surfaces in the graphics area.

b) Select the **Copy** check box.

c) Enter 3 for **Number of Copies**.

3 Under **Rotate**, do the following:



a) Click to highlight the **Rotation Reference Entity** box, then click the same edge used in the previous procedure.

b) Enter 45° for **Angle** using the spin box.

Three copies of the surfaces appear at 45° intervals around the selected edge.

4 Click **OK**.

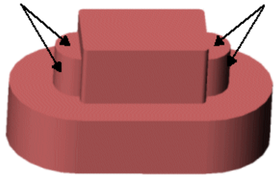
Mirror Surfaces

You can now mirror surfaces using **Insert**, **Pattern/Mirror**, **Mirror Feature**.

Deleting Faces in Surface Bodies

You can delete a face from a surface body and automatically patch it.

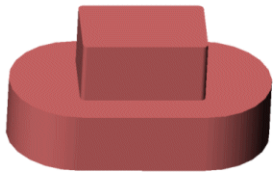
To delete and patch a surface body:



1. Open **delete_and_patch.sldprt**.
2. Click **Delete Face**  on the Surfaces toolbar, or **Insert**, **Face**, **Delete**.

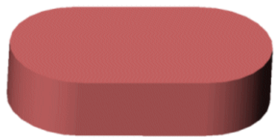
   The **Delete Face** PropertyManager appears.
3. Under **Selections**, click the faces shown to add them under **Faces to delete**.
4. Under **Options**, select **Delete and Patch**.
5. Click **OK** .

   The half-cylinder bosses disappear and the adjoining faces extend to form an unbroken surface.



6. Repeat steps 2 through 5 to delete the rectangular boss. Make sure you select the fillet faces as well as the larger faces for deletion.

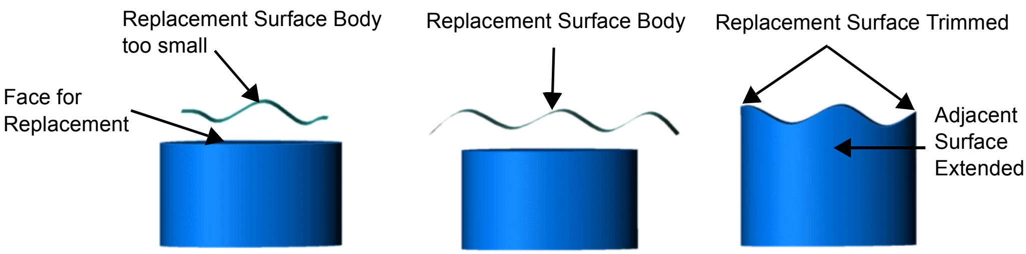
   The rectangular boss disappears and the adjoining face extends to form an unbroken surface.



Replacing a Face in a Surface Body

You can replace a face in a surface body or solid body with a replacement surface body. If the edges of the replacement surface body do not match the edges of the original face exactly, the adjacent faces are trimmed and extended.

The replacement surface body is trimmed to the adjacent surfaces. In most cases, you should use a replacement surface body larger than the face for replacement.



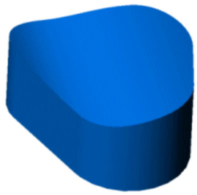
You can replace one or more sets of connected faces in a surface body with replacement surface bodies. When you replace more than one set of connected faces, the selection order of the **Target faces for replacement** and **Replacement surfaces** should correspond. See the SolidWorks 2001Plus online help for more information.

For an example of replacing a face in a solid body, see page 5-16.

To replace a face in a surface body:

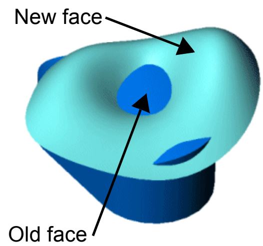
1 Open **replace_surface.sldprt**.

The knit surface body appears.



2 Right-click **Surface-Fill2** in the FeatureManager design tree and select **Show Surface Body**.

The light blue replacement face overlays the knit surface body.



3 Click **Replace Face** , or **Insert**, **Face**, **Replace**.

The **Replace Face** PropertyManager appears.

4 Under **Replace Parameters** do the following:

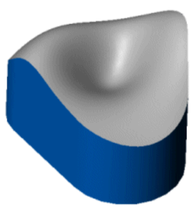
- Click the top face of the knit surface body (old face) to place it in the **Target faces for replacement** box.
- Click to highlight the **Replacement surfaces** box, then click the replacement face (new face) to place it in the box.

5 Click **OK** .

The face is replaced and the surface body is trimmed to fit.

6 Right-click on **Surface-Fill2**, and select **Hide Surface Body**.

The replacement face is hidden and the knit surface body is shown with the replacement face intact.



7

Assemblies

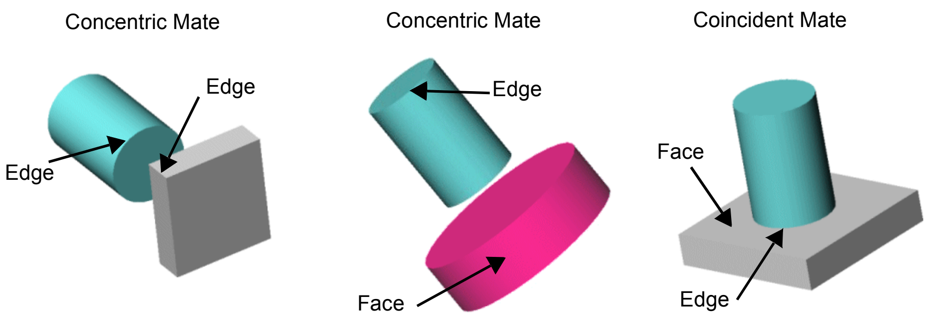
This chapter describes general information about working with assemblies of any kind. The following enhancements are discussed:

- Enhanced mate selections
- Diagnosing mate problems
- Moving and rotating multiple components
- Mirror components interface
- Explode lines
- Selection of interior parts
- Transparency in assemblies
- Large assembly performance
- Physical Dynamics
- MateGroup consolidation

Mates

Enhanced Mate Selections

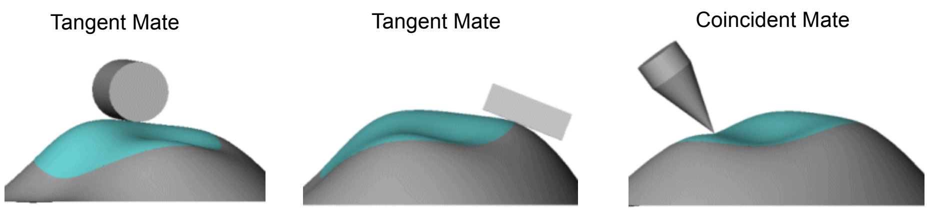
Circular Edges - You can add concentric and coincident mates to circular edges.



NOTES:

- You can add a concentric mate between circular edges. You can add a coincident mate between circular edges only if those edges have the same radius.
- You cannot mate arc edges.

General Surfaces - You can add cylinder tangent, plane tangent, and point coincident mates to any surface.

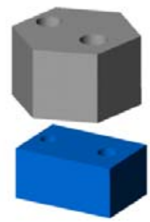


Mate Diagnostics

Mate Diagnostics is a new tool that allows you to identify mating problems in an assembly. You can examine the details of mates that are not satisfied, and identify groups of mates which over define the assembly.

Mate Diagnostics may find a mating error that went undetected in earlier versions of SolidWorks. These are primarily conflicts between mates and in-context features. Remember that when this error occurs, it is important to understand that the mate problem is not a new one. It existed in earlier versions of SolidWorks but was not detected by the software.

To use Mate Diagnostics:



1 Open **\concentric_circles\concentric_circles.sldasm**.

   Notice the down arrow next to the assembly name and **MateGroup1**. This indicates rebuild errors in the assembly.

2 Expand **MateGroup1**.

   Two mates are shown:

   - **Concentric1** is highlighted with a yellow exclamation point. This means the mate is satisfied, but over defines the assembly.
   - **Concentric2** is highlighted with a red exclamation point. This means the mate is not satisfied.

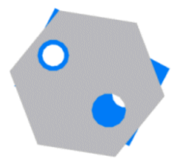
3 Click **Tools**, **Mate Diagnostics**.

   The **Diagnostics** PropertyManager appears.

4 Under **Analyze problem**, click **Analyze**.

   The over defined set of mates is listed. The message states that the holes in the two components have different spacings, and lists the spacings.

5 Under **Not satisfied mates**, click **Concentric2**.



   The entities in the not satisfied mate are highlighted in the graphics area. A message states the cylinders are not concentric, and lists the distance by which the holes in the two components are currently misaligned.

6 Click **OK**.

NOTE: This example shows only one type of problem that Mate Diagnostics can identify. You can find problems with other types of mates such as parallel, coincident, and so on. Mate Diagnostics analyzes any size assembly, but it is more likely to identify specific problems in assemblies with a small number of mates.

To fix the mate problems:

1 Expand ⊞ **Part1**.

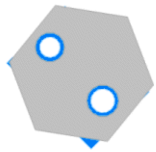
2 Right-click **Sketch1** under **Base-Extrude**, and select **Edit Sketch**.

Part1 turns pink in the FeatureManager design tree, and the sketch appears.

3 Add a dimension of 125mm between the hole centers.

4 Close the sketch, and click **Edit Part** on the Assembly toolbar to return to editing the assembly.

5 Repeat steps 1-4 for **Part2**.



Make sure that the 125mm dimension you add is a true distance, and not a horizontal component.

The components move into the correct orientation. The warning symbols disappear. Both mates are satisfied and they do not over define the assembly.

General Enhancements

Moving and Rotating Multiple Components

You can now select more than one component at a time to move or rotate. When you move multiple components, all the mates are solved. Dynamic clearance and collision detection work as they do for a single component.

Mirror Components Interface

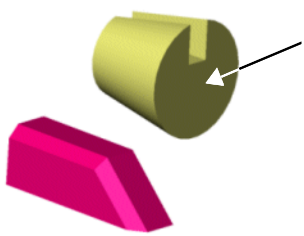
Mirror Components now uses the PropertyManager instead of a dialog box.

To mirror components in an assembly:

1 Open **\component_mirror\component_mirror.sldasm.**

2 Click **Insert**, **Mirrored Components**.

The **Mirror Components** PropertyManager appears with **Step 1: Selections** displayed.



3 Select the face on the wide end of the cone as the **Mirror plane**.

4 Under **Components to Mirror**, select **cone_and_block-1** in the Flyout FeatureManager design tree.

5 Click ⊞ to expand **cone_and_block-1** in the PropertyManager.

The components of the sub-assembly are listed.

6 Click ☐ next to **block-1** to toggle from copying to mirroring ☑.

With mirroring, a new part is made.

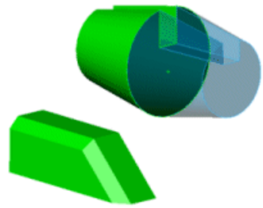
The sub-assembly **cone_and_block-1** is automatically checked.

7 Click **Next**.

Step 2: Filenames appears.

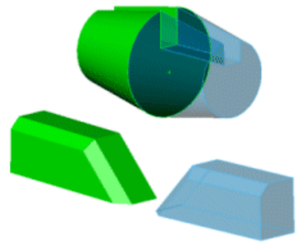
8 Click **Next** again to accept the default file names.

Step 3: Orientation appears. The copied cone is highlighted.



9 If the grooves in the cones do not line up, click **Reorient Component** until they do.

10 Select the **Preview mirrored components** check box to view the orientation of the mirrored block.



11 Click **OK** to create the new sub-assembly and component and close the PropertyManager.

Assembly Explode Lines

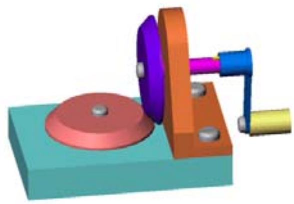
You can create explode lines to show the part relationships in an exploded view of an assembly. Draw explode lines between two components to indicate that they are aligned in the final assembly.

Explode lines can connect the following:

- *Cylindrical faces/circular edges*. Select the circular edges or cylindrical faces of the boss or cut. The explode line appears along the axis of the cylinder.
- *Straight edges*. The explode line extends from the edge.
- *Planar faces*. The explode line extends perpendicular to the face. Click the face at the point where you want the explode line to attach.

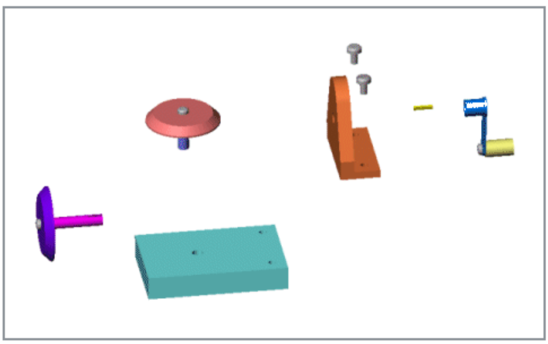
NOTE: You can create an **Explode Line Sketch** only in an assembly with an exploded view.

To open an explode line sketch in an assembly:



1. Open **\gear_assy\gear_assy.sldasm**.
2. Click the **ConfigurationManager** tab.
3. Expand **Default**.
4. Right-click **ExplView1** and select **Explode**.

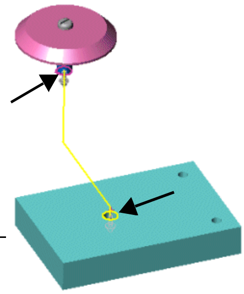
   The assembly explodes.
5. Click **Zoom to Fit**.



6. Click **Explode Line Sketch** on the Assembly toolbar, or **Insert**, **Explode Line Sketch**.

   A new 3D sketch opens. The **Route Line** PropertyManager appears.

To create explode lines:



1. Click the bottom edge of the vertical shaft and the edge of the hole in the base as shown.

   The entities appear under **Items to Connect**.

   An explode line appears.

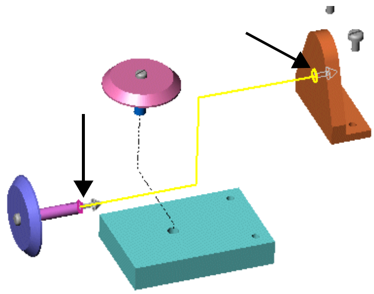
2. Click the handles to reverse the line direction if necessary.

TIP: You can click the handles, or use the options in the PropertyManager to change the shape of the explode line.

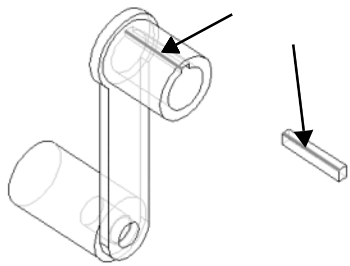
---

3. Click **OK** to place the explode line in the explode line sketch.

   In the sketch, you can drag the lines to reposition them if needed.



4. Click **Route Line** on the Routing toolbar, or **Tools**, **Sketch Entity**, **Route Line**.
5. Create an explode line between the edge of the horizontal shaft, and the edge of the hole in the bracket as shown.
6. Click **OK**.
7. Click **Route Line**, or **Tools**, **Sketch Entity**, **Route Lines**.



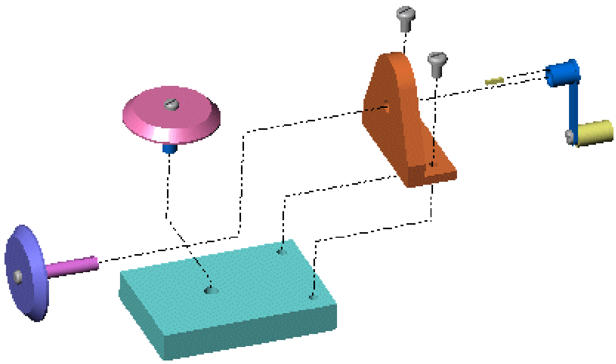
8. Create an explode line between the edge of the key, and the upper edge of the key slot in the handle as shown.
9. Click **OK**.

10 Continue to add explode lines to the assembly as needed.

11 Exit the sketch.

NOTE: Notice that the explode line sketch appears in the ConfigurationManager, and not in the FeatureManager design tree.

The completed sketch may look like this:



Exploded Assembly Enhancement

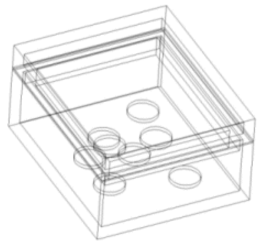
If you delete a component in an exploded assembly, the exploded configuration remains the same. In earlier versions, if you deleted a component that was included in an explode step, the explode step was deleted.

You can test this using the assembly from the previous procedure. Collapse the exploded assembly, delete the screws, then explode the assembly again. The exploded configuration does not change. You can edit the explode line sketch to remove any lines that are no longer needed.

Selecting Interior Parts

You can select interior parts in an assembly using a new option in **Advanced Component Selection**. Interior parts are those parts that you cannot see when the assembly is opaque and shaded.

To select interior parts:



1 Open **\box\box.sldasm**.

2 Click **Wireframe**  to see the disks inside the box.

3 Click **Tools**, **Advanced Select**.

The **Advanced Component Selection** dialog box opens.

4 Under **Define additional criteria**, do the following:

- Under **Property**, select **Part is interior detail - SW Special**.
- Under **Condition**, select **is yes**.

5 Click **Add**.

Part is interior detail - SW Special is yes appears under **Criteria**.

6 Click **Apply**.

The eight discs are selected.

7 Click **Change Suppression State**  on the Assembly toolbar, then click **Suppress**.

The **SolidWorks 2001Plus** dialog box appears.

8 Click **Yes** to save your changes.

The discs become suppressed.

9 Close the assembly without saving.

Using Transparency in Assemblies

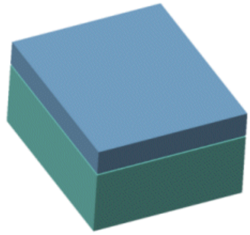
You can select from several options to display assembly transparency during part editing. (See **Transparency in Assemblies** on page 1-7.) The part you are editing is always opaque, but you can vary the appearance of the other components.

During selection, the opaque component is selected first, if the other components in front of it are more than 10 percent transparent. As you move your cursor over the opaque component, you can select its face with a single click.

To select a transparent component with an opaque component behind it, press **Shift** when selecting.

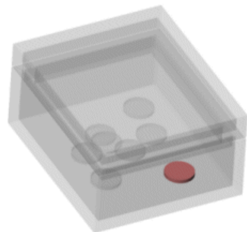
To select the part you are editing through an opaque component, press **Tab** when selecting.

To set assembly transparency during part editing:



1. Open **\box\box.sldasm**.
2. Click **Tools**, **Options**. On the **System Options** tab, click **Display/Selection**.
3. Under **Assembly transparency for in context edit**, select **Force assembly transparency**, and move the slider about 3/4 of the way to the right.
4. Click **OK**.
5. Right-click **disc<1>** in the FeatureManager design tree, and select **Edit Part**.

   Disc<1> turns pink and the other components turn transparent.



6. Move the cursor over **disc<1>** in the graphics area.

   Notice that the edge of **disc<1>** highlights. Because the components in front of it are transparent, **disc<1>** is the default selection.
7. Right-click **disc<1>** in the graphics area and select **Edit Sketch**.
8. Change the disc diameter dimension to 17mm and close the sketch.

   The disc diameter changes.
9. Click **Edit Part**  on the Assembly toolbar to return to editing the assembly.

   The assembly turns opaque

TIP: Once you have set the transparency level, click **Force Transparency**  on the Assembly toolbar to enable transparency during part editing.

Large Assembly Mode

Large Assembly Mode allows you to specify various performance-related options that apply only to large assemblies.

To specify Large Assembly Mode options:

1 Click **Tools**, **Options**. On the **System Options** tab, click **Large Assembly Settings**.

The options appear pre-set with recommended settings for use with large assemblies.

Large assembly threshold is the number of components above which Large Assembly Mode activates or sends a message, depending on your settings.

2 In the **Automatically activate Large Assembly Mode** list, select from the following:

- **Prompt**. When you reach the threshold, a message appears with the option to activate Large Assembly Mode.
- **Never**. Ignores the threshold.
- **Always**. When you reach the threshold, Large Assembly Mode automatically activates.

3 Modify any of the other settings as needed.

Many of the options also appear in other areas of the **Options** dialog box. The settings you choose under **Large Assembly Settings** apply only when you are in Large Assembly Mode. Options that you set elsewhere apply when Large Assembly Mode is off.

4 Click **OK**.

To activate Large Assembly Mode:

Click **Large Assembly Mode** on the Assembly toolbar, or click **Tools**, **Large Assembly Mode**.

-or-

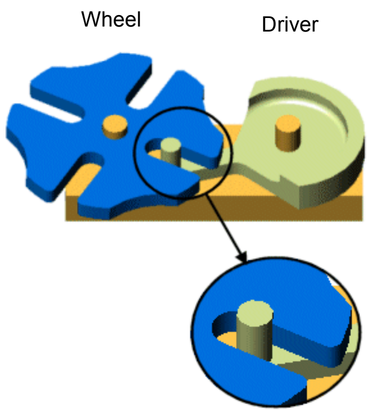
Set the threshold settings as described previously to activate Large Assembly Mode when the number of components in your assembly reaches the specified threshold.

Physical Dynamics

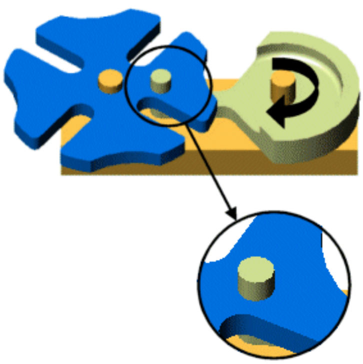
Physical Dynamics is a new option in Collision Detection that allows you to see the motion of assembly components in a realistic way. With Physical Dynamics enabled, when you drag a component, the component applies a force to components that it touches. The effect is to move or rotate contacted components within their allowable degrees of freedom. The dragged component reacts to a collision by rotating within its allowable degrees of freedom or by sliding against a constrained or partially constrained component to allow the drag to continue.

Physical Dynamics propagates throughout the assembly. The dragged component can push aside a component, which then moves into and pushes aside another component, and so on.

In the following example of a geneva wheel mechanism, the driver and the wheel are constrained to rotate about fixed shafts. During motion of the mechanism, each full turn of the driver turns the wheel 1/4 turn, as the pin on the driver engages the slot in the wheel, and pushes against the side of the slot. As the pin slides out of the slot, the wheel stops turning. The driver continues turning, bringing the pin around to engage in the next slot. The effect is to turn constant rotary motion into periodic rotary motion.



Geneva wheel mechanism in starting position.



If neither Physical Dynamics nor Collision Detection is enabled, the pin passes through the wheel as shown. If Collision Detection is enabled without Physical Dynamics, the pin stops when it contacts the edge of the slot.

Physical Dynamics activated. As the driver rotates, the pin contacts the side of the slot and forces the wheel to rotate about its axis. The pin slides down, and then up the slot, pushing the wheel. As the pin loses contact with the side of the slot, the wheel stops moving.

To activate Physical Dynamics:

1. Click **Move Component** on the Assembly toolbar, or click **Tools**, **Component**, **Move**.

   The **Move Component** PropertyManager appears.

2. Select the **Collision Detection** check box.
3. Select the **Physical Dynamics** check box.
4. Drag a component.
5. Click **OK**.

TIPS: Physical Dynamics works best (and is most meaningful) on assemblies that have only a few degrees of freedom. Add all appropriate mates prior to running Physical Dynamics.

You can select specific components in Collision Detection to improve the performance of Physical Dynamics. Choose only those components that are directly involved in the motion you are testing.

MateGroup Consolidation

In earlier versions of SolidWorks, an assembly could have more than one mategroup. In SolidWorks 2001Plus, each assembly contains only one mategroup. The program solves all the mates together.

When you save an existing assembly with multiple mategroups in SolidWorks 2001Plus, the software consolidates all the mates in that assembly into a single mategroup. The other mategroups no longer appear in the FeatureManager design tree.

There are several advantages to solving all the mates in a single mategroup:

- *Improved drag behavior*. The program solves all mates during a drag. In earlier versions of SolidWorks, not all mates were solved during a drag. It depended on which mategroups the dragged component referenced.
- *No mategroup propagation*. All mates are created in and stay in **MateGroup1**. In earlier versions, you could not create some types of mates until an existing group of mates moved to a new mategroup.
- *Fewer mating errors.* You can create some mates to assembly-level features that previously were not possible.

8

Configurations

- ❑ You can set the following items with different values in specified configurations:
  - Individual features
  - End conditions
  - Sketch planes
  - Equations
  - Sketch constraints
  - Sketch dimensions
  - External sketch relations
- ❑ Other new functionality in configurations include:
  - Nested configurations
  - Design table icon

Individual Features

You can suppress individual features of parts in specified configurations. Instead of clicking **Edit**, **Suppress** to suppress individual features in certain configurations, you can use a shortcut menu.

To suppress individual features using the shortcut menu:

1. Right-click the feature you want to suppress in the FeatureManager design tree and select **Properties**.
2. In the **Feature Properties** dialog box, do the following:
   - Select the **Suppressed** check box.
   - Select **This Configuration**, **All Configurations**, or **Specify Configuration(s)**.
3. Click **OK**.

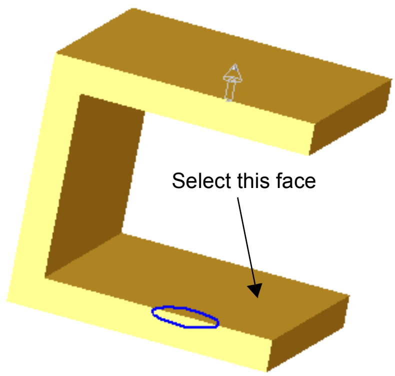
End Conditions

You can change the end condition of extruded features in specified configurations.

To change the end condition of an extruded feature in a configuration:

1. Open **configuration.sldprt**.
2. In the FeatureManager design tree, right-click **Boss-Extrude-Thin1** and select **Edit Definition.**

   The **Boss-Extrude-Thin1** PropertyManager appears.



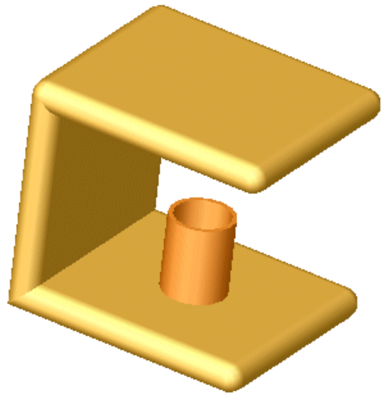
3. Under **Direction 1**, do the following:
   - Change **End Condition** to **Offset From Surface**.
   - Select the face shown.
   - Set **Offset Distance**  to 25mm.

4 Under **Configurations**, select **This configuration**.

5 Click **OK** .

The end condition of the boss-extrude changes in the active configuration, **Default**, only.

6 **Close** the part without saving it.



Sketch Planes

The plane on which a sketch lies is now configurable through the **Edit Sketch Plane** PropertyManager. You can place a single sketch on different planes in different configurations.

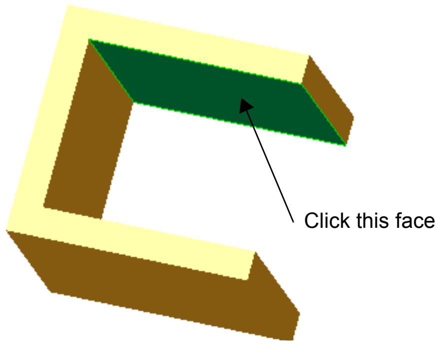
To change a sketch plane in a configuration:

1 Open **configuration.sldprt**.

2 In the FeatureManager design tree, click the ⊞ to expand **Boss-Extrude-Thin1**.

3 Right-click **Sketch2** and select **Edit Sketch Plane**.

The **Sketch Plane** PropertyManager appears.



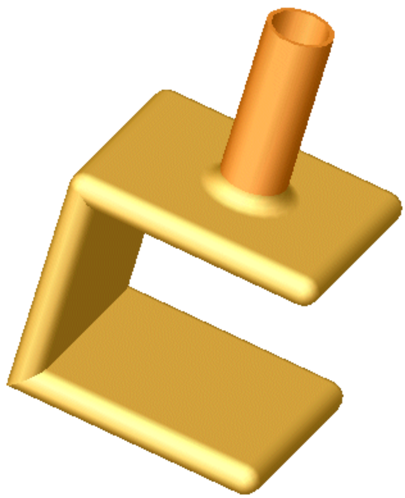
4 Under **Sketch Plane/Face**, click the face shown.

5 Under **Configurations**, select **This configuration**.

6 Click **OK** .

The sketch changes planes in the active configuration only.

7 **Close** the part without saving it.



Equations

You can suppress or unsuppress equations in specified configurations.

To suppress or unsuppress equations in configurations:

1 Open **configuration.sldprt**.

2 Click **Tools**, **Equations**.

The **Equations** dialog box appears.

3 Under the **Active** column, do the following:

- Clear the check box next to the equation **“D1@Fillet1” = 5**.

  Suppressed appears in the **Evaluates To** column.

- Select the check box next to the equation **“D1@Fillet1” = 1**.

  1mm appears in the **Evaluates To** column.

4 Click **Configs**.

The **configuration.sldprt** dialog box appears.

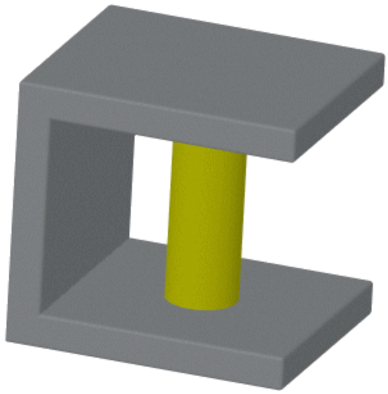
5 Select **All configurations**, and click **OK**.

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

7 Click **Rebuild** .

The fillet size changes in both configurations.

8 **Close** the part without saving it.



Sketch Relations

You can control the suppression state of sketch relations per configuration.

To suppress or unsuppress sketch constraints:

1 Open **sketch_relations.sldprt**.

This sketch contains three vertical lines with the top points in a fixed position.

2 Edit the sketch, click **Add Relation**  and an **Equal** relation to the three lines.

3 Exit the sketch.

4 In the ConfigurationManager, add a new configuration.

5 Edit the sketch again.

6 Click **Display/Delete Relations**  or **Tools**, **Relations**, **Display/Delete**.

The **Sketch Relations** PropertyManager appears.

7 Under **Relations**, do the following:
   - Select one of the **Equal radius/length** relations from the list.
   - Select the **Suppressed** check box.

     The status changes to **Driven**.
8 Under **Configurations**, do the following:
   - Select **Specify configurations**.
   - Select **Default**.
   - Click to clear the configuration you added in step 4.
9 Click **OK** ✔, but do not close the sketch.
10 Drag the lines by the bottom end points to see how the **Equal radius/length** relation is suppressed.

Sketch Dimensions

You can control the driving state of sketch dimensions in specified configurations to control the behavior of your model.

To control the driving state of sketch dimensions:

1. Sketch and dimension a line.
2. Right-click the dimension, and select **Properties**.

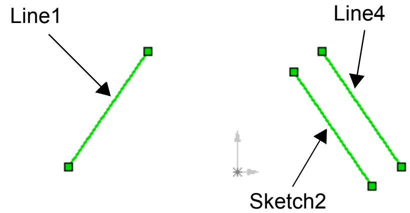
   The **Dimension Properties** dialog box appears.
3. Select the **Driven** check box.
4. Click **Specify Configs**.

   A dialog box appears.
5. Select the configurations you want to apply the new settings to: **This configuration**, **All configurations**, or **Specify configurations**.
6. Click **OK**.
7. Click **OK** again to close the **Dimension Properties** dialog box.

External Sketch Relations

You can set different external sketch relations in different configurations.

To set external sketch relations:



1 Open **config_sketch.sldprt**.

2 Right-click **Sketch2** and select **Edit Sketch**.

3 Click **Display/Delete Relations** or **Tools**, **Relations**, **Display/Delete**.

   The **Sketch Relations** PropertyManager appears.

4 Under **Entities**, select **Line4** from the table in the **Entity** column.

   Line4 of **Sketch1** is highlighted in the graphics area.

5 In the graphics area, select **Line1**.

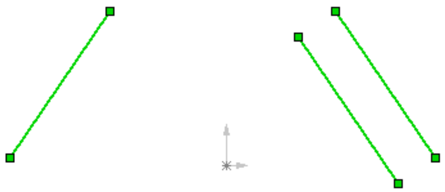
   Line1@Sketch1 appears in the **Replace** box as a replacement for **Line4**.

6 Under **Configurations**, select the following:
   - **Specify configurations**
   - **config2**

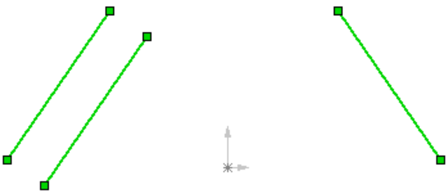
7 Under **Entities**, click **Replace**.

8 Click **OK** and exit the sketch.

   The relation is updated in **config2** only.



Sketch before replacing external sketch relation



Sketch after replacing external sketch relation

Nested Configurations

You can now create nested configurations. This allows you to create a parent-child relationship within a configuration. By default, all parameters in the child configuration are linked to the parent configuration. However, you can override any configurable parameter in the child configuration so that the parameter is no longer linked to the parent.

To create a nested configuration:

1 In the ConfigurationManager, right-click a configuration and select **Add Derived Configuration**.

  The **Add Configuration** dialog box appears.

2 Type a name in the **Configuration Name** box.

3 Under **Properties for newly inserted items**, select from the following check boxes. These options control what happens to the nested configuration when you add new items to another configuration:

  - **Suppress features** (parts only). New features are suppressed.
  - **Suppress features and mates** (assemblies only). New features (including feature cuts and holes, sketches, and so on that belong to the *assembly*) and mates are suppressed.
  - **Hide component models** (assemblies only). New components are hidden.
  - **Suppress component models** (assemblies only). New components are suppressed.

4 Under **Part number displayed when used in a bill of materials**, do the following:

  a) Select one of the following options from the list:

  - **Document Name**. The part number is the same as the document name.
  - **Configuration Name**. The part number is the same as the configuration name.
  - **Link to Parent Configuration**. The part number is the same as the parent configuration name.
  - **User Specified Name**. The part number is a name that you type.

  b) (Optional). Select the **Don't show child components in BOM when used as sub-assembly** check box (assemblies only). When checked, the sub-assembly is *always* shown as a single item in the bill of materials.

5 Click **OK**.

  The new configuration is added to the ConfigurationManager underneath its parent.

Design Tables

When you insert a design table into a part or assembly document, the design table icon appears in the FeatureManager design tree. To edit the design table, right-click **Design Table** in the FeatureManager design tree and select **Edit Definition**.

9

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:

- Opening drawings from model documents
- Displaying drawing views in shaded mode
- New functionality in RapidDraft drawings
- Setting automatic hidden components list
- New tools to hide and show edges
- Displaying assembly explode lines in drawings
- Chinese (GB) detailing standard now available
- New functionality in dimensions
- New annotation tools
- Annotations now in the PropertyManager

Opening Drawings from Model Documents

You can now open drawings directly from inside part and assembly documents. Right-click the top item in the FeatureManager design tree or anywhere on the model in the graphics area and select **Open drawing**.

SolidWorks looks for a drawing with the same name as the model, in the same folder as the model. If the drawing exists, it opens automatically. If such a drawing is not found, a browse window appears so you can locate a drawing manually.

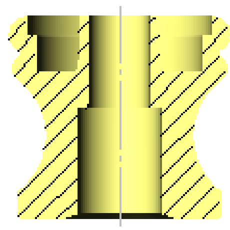
Displaying Drawing Views in Shaded Mode

Shaded display mode helps clarify representation of a model. Many drawing views can now be displayed in shaded mode and with hidden lines removed (HLR) when in shaded mode.

The system options for drawings called **Default Edge Display** is now named **Default Display Type**. You can set the default display type for new views in options. You can also select the display type for individual views from the View toolbar.

Shaded mode is available for the following types of views:

- **Standard 3 View**
- **Named View**
- **Projected View**
- **Auxiliary View**
- **Section View**
- **Relative View**

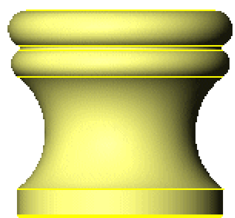


To set the default display mode for new drawings:

1. Click **Tools**, **Options**, **System Options**.
2. Under **Drawings**, click **Default Display Type**.
3. Under **Default display mode for new drawing views**, choose **Shaded**.
4. If necessary, also select the **HLR edges when shaded** check box.
5. Click **OK**.

To display an individual drawing view in shaded mode:

1. Open **drw_shaded.slddrw** and select the drawing view.
2. Click **Shaded**  on the View toolbar.
3. Toggle **Display HLR Edges in Shaded Mode**  to see its effect.



New Functionality in RapidDraft Drawings

The following new functionality is available in RapidDraft drawings without loading the model:

- Create Detail Views
- Create Crop Views
- Create and edit Auxiliary and Projected Views of parts
- Create Broken-out Sections of parts
- Create Section Views of parts
- Shift between Hidden Lines in Gray (HLG) and Hidden Lines Removed (HLR) display for parts
- Choose Tangent edge display (Visible, With Font, Removed) available in HLG and HLR modes
- Copy and Paste, and Cut and Paste, of drawing views
- Hide Edges and Show Edges

Setting Automatic Hidden Components List

You can now set an option to automatically list hidden components when you create a drawing view.

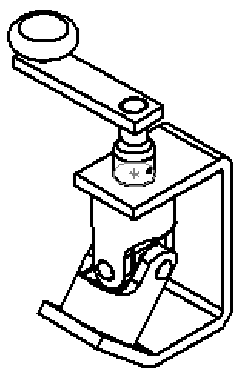
To populate the hidden components list automatically:

1 Click **Tools**, **Options**, **System Options**, **Drawings**.

2 Select the **Automatic hiding of components on view creation** check box, then click **OK**.

3 Open a new drawing and open **\samples\tutorial\universal_joint\ujoint.sldasm**.

4 Click **Named View** , click in the graphics area of the assembly, select ***Isometric** from the **View Orientation** list, and click in the graphics area of the drawing to place the view.



NOTE: If the drawing is in **Shaded** mode, click **Hidden Lines Removed** on the View toolbar.

5 Right-click the view and select **Properties**.

6 Select the **Hide/Show Components** tab.

Note the hidden component listed.

7 Click **OK**.

8 Keep the drawing open to use in the next procedure.

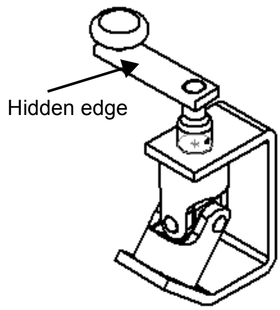
New Tools to Hide and Show Edges

Two new tools on the Line Format toolbar facilitate hiding and showing edges in drawings. These tools are the same as the shortcut menu items **Hide Edge** and **Show Edge**.

To hide and show edges:

1. Click **Tools**, **Options, System Options**, **Drawings**, make sure the **Select hidden entities** check box is selected, and click **OK**.
2. In the drawing of **ujoint.sldasm** (created in the previous procedure), select an edge and click **Hide Edge** on the Line Format toolbar, or right-click and select **Hide Edge**.

   The edge disappears.

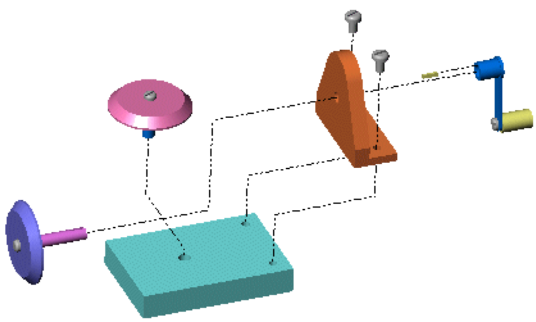
3. Select the edge again and click **Show Edge** on the Line Format toolbar, or right-click and select **Show Edge**.

   The edge reappears.

Displaying Assembly Explode Lines in Drawings

You can now insert explode lines into assemblies. You can also edit the explode lines and other lines with new sketch tools (see **New Routing Toolbar** on page 2-6). The explode lines are displayed in drawings of exploded configurations.

See **Assembly Explode Lines** on page 7-6 for an example of inserting explode lines into an assembly. Here is the drawing of the assembly, in shaded mode, showing the explode lines.



Chinese (GB) Detailing Standard Now Available

The Chinese dimensioning standard (GB) is now available. The standard is similar to ISO.

To use the Chinese dimensioning standard:

1. In a drawing, click **Tools**, **Options**, **Document Properties**, **Detailing**.
2. Under **Dimensioning standard**, choose **GB** from the list.

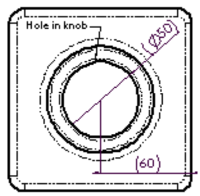
New Functionality in Dimensions

Hiding dangling dimensions

If you *delete* features in parts and assemblies, reference dimensions and annotations in the drawing document are left dangling. You now have the option to hide these dangling dimensions and annotations automatically.

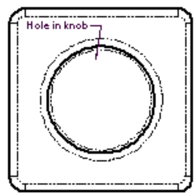
If you *suppress* features, SolidWorks automatically hides dangling reference dimensions in drawings, independently from the new option. SolidWorks does not automatically hide dangling reference *annotations* when features are suppressed; however, the new option hides them.

To hide dangling dimensions:



1. Open **tutor1.sldprt** and **dims_dangling.slddrw**.
2. Tile the windows.
3. In the drawing document, click **Tools**, **Options**, **Document Properties**. Under **Detailing**, click **Annotations Display**, clear **Hide dangling dimensions and annotations**, and click **OK**.
4. In the FeatureManager design tree in the part document, right-click **Hole_in_Knob** and select **Suppress**.

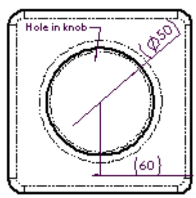
   Note the two dimensions in the drawing document.



5. Select the drawing window to display the change.

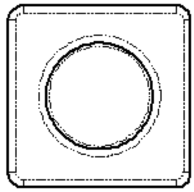
   The dangling dimensions are hidden automatically.
6. In the FeatureManager design tree in the part document, right-click **Hole_in_Knob** and select **Delete**. In the **Confirm Delete** dialog box, click **Yes**.
7. Select the drawing window.

   The dangling dimensions reappear.



8 Click **Tools**, **Options**, **Document Properties**. Under **Detailing**, click **Annotations Display**, select the **Hide dangling dimensions and annotations** check box, and click **OK**.

The dangling dimensions and the annotation are hidden automatically.



Dimension Favorites

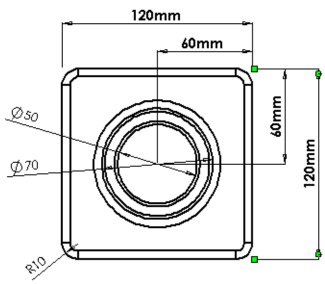
You can now define dimension styles, similar to paragraph styles in word processing documents. The functionality of these styles, called **Dimension Favorites**, includes:

- Any dimension attribute can be saved as part of a style.
- Styles can be applied to multiple dimensions.
- Styles are named so they can be referenced.
- Styles can be saved and loaded in part, assembly, and drawing documents. You can access styles created in other documents and located in other folders.
- Styles can be added, updated, and deleted in a document.
- Favorites styles cannot be applied to dimensions created by **Hole Callout**.

To add a dimension favorite to a document:

1 Open **dims_favorites.slddrw**.

2 Select one of the linear dimensions (120, for example).



3 In the **Dimension** PropertyManager, under **Dimension Text**, type **mm** after **<DIM>** in the middle box.

4 Under **Dimension Favorites**, click **Add or Update a Favorite** .

5 In the **Add or Update a Favorite** dialog box, type **favorite_mm** under **Enter a new name or choose an existing name**, then click **OK**.

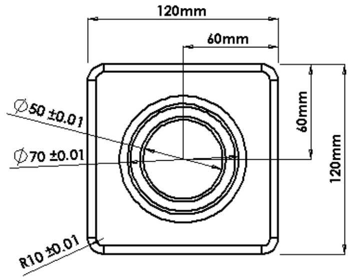
6 Hold **Ctrl** and select the other linear dimensions, then select **favorite_mm** from the **Dimension Favorites** list.

The **mm** appears after all the selected dimensions.

7 Under **Dimension Favorites**, click **Save a Favorite**  and save the file with the default name and extension **.sldfvt**.

NOTE: You can use a favorite within a document without saving it. If you **Save a Favorite** , you can load it into any other document with **Load Favorites** .

To update a dimension favorite:



1. With **dims_favorites.slddrw** still open, select one of the diameter dimensions (50, for example).
2. In the **Dimension** PropertyManager, under **Tolerance/Precision** choose **Symmetric** from the **Tolerance Type** list and type 0.01mm for **Maximum Variation**.
3. Under **Dimension Favorites**, click **Add or Update a Favorite**.
4. In the **Add or Update a Favorite** dialog box, type **favorite_tol** under **Enter a new name or choose an existing name**, then click **OK**.
5. Click in the graphics area to exit the **Dimension** PropertyManager, then hold **Ctrl** and select the other circular dimensions, then select **favorite_tol** from the **Dimension Favorites** list.
6. Click in the graphics area to exit the **Dimension** PropertyManager, then right-click one of the circular dimensions and select **Properties**.
7. In the **Dimension Properties** dialog box, clear the **Use document's font** check box, and click **Font**.
8. In the **Choose Font** dialog box, under **Font Style** select **Bold**, under **Height** select **Points** and **26**, then click **OK**.
9. Click **Apply** to preview the changes, then click **OK**.
10. Click **Add or Update a Favorite**.
11. In the **Add or Update a Favorite** dialog box, select **favorite_tol** from the **Enter a new name or choose an existing name** list, select **Update all annotations linked to this favorite**, then click **OK**.

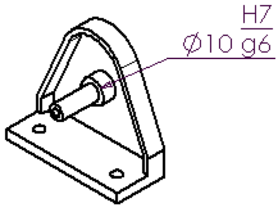
    All the dimensions in the **favorite_tol** style are updated with the new formatting.

To reset dimensions to default attributes:

1. Select a dimension or multiple dimensions.
2. Click **Apply the default attributes to selected dimensions**.

    The dimensions are reset to their default values.

Fit Tolerance



When you select **Fit** or **Fit w/Tolerance** under **Tolerance/ Precision** in the **Dimension** PropertyManager, the tolerances defined in the standards (such as **H11** or **k6** in ISO) are now available from lists for both the **Hole Fit** and **Shaft Fit**. You can also type in any text, and you can display the hole fit and stack fit as **Stacked with line display**, **Stacked without line display**, or **Linear display**.

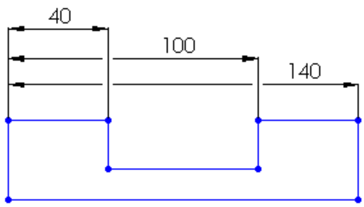
Parentheses in Tolerances

You can add parentheses around **Bilateral**, **Symmetric**, and **Fit w/Tolerance** tolerances.

Break Dimension Lines

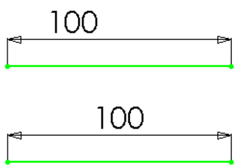
You can now break dimension witness lines in drawings when they cross each other. The gap distance default can be set for the document in options, or in the **Dimension** PropertyManager for individual dimensions.

To break dimension witness lines:



1. Open **dims_break_lines.slddrw**.
2. Click **Tools**, **Options**, **Document Properties**. Under **Detailing**, click **Dimensions**, under **Break Dimension Witness/Leader Lines** set the value for **Gap** to 4mm, and click **OK**.
3. Select the dimension of 140.
4. In the **Dimension** PropertyManager, select the **Break Dimension Lines** check box.
5. Select the **Use document's gap** check box.
6. Select the dimension of 100 and repeat steps 4 and 5.
7. Click **OK**.

Center Text

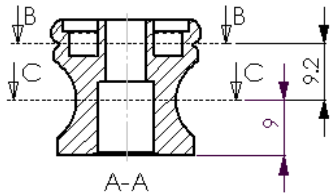


Center Text has been changed to **Center between witness lines** in both options and the shortcut menu.

To set the default for the document, click **Tools**, **Options**, **Document Properties**. Under **Detailing**, click **Dimensions** and select the **Center between witness lines** check box.

For individual dimensions in a document, right-click the dimension (or multiple dimensions) and select **Display Options**, **Center between witness lines**.

Dimensioning to Section Lines



You can now add dimensions to section lines without editing the section line sketch.

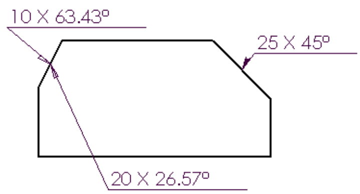
- You can dimension from one section line to another.
- You can dimension from a section line to any other appropriate geometry within the same view.
- The dimensions are driving dimensions.

Click **Dimension** and select a section line to dimension

Chamfer Dimensions

You can now specify chamfer dimensions from the **Dimension** shortcut menu, or from the **Tools**, **Dimensions** menu, with various display properties and separate tolerances in the **Dimension** PropertyManager.

To add a chamfer dimension to a drawing:



1 Open **drw_chamfer_dim.slddrw**. The drawing contains a part with two chamfers. The chamfer on the left is defined by two distances (10mm and 20mm), and the chamfer on the right is defined by a distance and an angle (25mm and 45 deg).

2 Click **Dimension** on the Sketch Relations toolbar, then right-click in the graphics area and select **Chamfer Dimension**.

The pointer changes to .

3 Select the chamfer edge at the left, select the top horizontal edge, then click in the graphics area to place the dimension. Under **Dimension Text**, click **Distance X Angle** 1x45°.

NOTE: You must select the chamfer edge first. Nothing happens until you select the second edge.

The tool remains active for you to select other chamfers.

4 Select the chamfer edge at the left again, select the left vertical edge, then click in the graphics area to place the dimension.

5 Select the chamfer edge at the right, select either the top horizontal edge or the right vertical edge, then click in the graphics area to place the dimension.

You can edit any chamfer dimension properties in the **Dimension** PropertyManager. For example, you can display the dimension as **Distance X Distance** 1x1, with **Angled Text** , or with tolerances.

6 Click **OK** .

New Annotation Tools

Dowel Pin Symbol

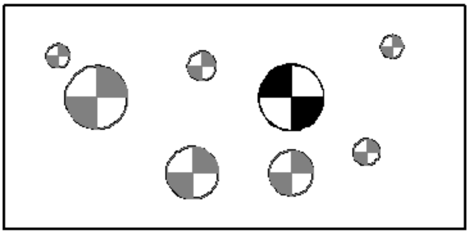
A **Dowel Pin Symbol** tool has been added to the Annotation toolbar for drawings. You can select the tool first or select multiple holes (circular edges or sketch circles) first. The symbol conforms to the size of the selected hole.

You cannot move dowel pin symbols with cut and paste operations.

To add dowel pin symbols to a drawing:

1 Open **drw_dowel_pin.slddrw**.

2 Click **Dowel Pin Symbol** on the Annotation toolbar, or click **Insert**, **Annotations**, **Dowel Pin Symbol**.



3 Select the edge of a hole.

A dowel pin symbol is inserted, filling the hole.

4 Click **OK**.

5 Hold **Ctrl** and select multiple holes.

6 Click **Dowel Pin Symbol**.

Dowel pin symbols appear in all the selected holes.

7 Click **OK**.

8 Select one of the dowel pin symbols.

The pointer changes to when it is over a symbol.

9 In the **Dowel Pin Symbol** PropertyManager, under **Display Attributes**, select the **Flip Symbol** check box and click **OK**.

The symbol rotates 90 degrees counterclockwise.

NOTE: You can also right-click a dowel pin symbol and select **Flip Symbol**.

Multi-jog Leader

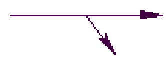
You can use the new **Multi-jog Leader** tool with other annotations or by itself to create leaders with as many bends as required, or to create simple arrows or block diagrams. For example, the multi-jog leader can point a note to an entity in a drawing that is difficult to reach with a straight or bent leader.

- Each segment of the leader is previewed as you move the pointer.
- You can attach the leader to a dock point of an annotation (Note, Surface Finish Symbol, Geometric Tolerance Symbol, and so on), or you can start it anywhere on a drawing sheet. A leader attached to an annotation moves with the annotation.

- Multi-jog leader is one of the leaders you can choose in the PropertyManagers of annotations with leaders, such as Notes, Geometric Tolerance Symbols, and so on.
- If you end the leader at an entity, the leader attaches to the entity with an arrow. You can also end the leader anywhere on the drawing sheet by double-clicking.
- To add an arrow at either endpoint of a leader, right-click the endpoint and select an arrow style from the menu.
- To delete the leader, right-click anywhere along the leader and select **Delete leader**.
- To delete a jog point, right-click the point and select **Delete jog point**.
- To add a jog point, right-click a segment and select **Add jog point**.
- To add a branch, right-click a jog point and select **Insert new branch**.
- You can drag the jog points.
- You can add and drag horizontal bends.

To create a multi-jog leader:

1. Open a new drawing and click **Multi-jog Leader** on the Annotations toolbar, or click **Insert**, **Annotations**, **Multi-jog Leader**.
2. Click in the graphics area, move the pointer to create a horizontal line, and double-click.
3. Move the pointer over the leader endpoint (the pointer changes to), right-click and select an arrow style from the menu.
4. Move the pointer over the middle of the leader (the pointer changes to), right-click and select **Add jog point**.
5. Move the pointer over the new jog point (the pointer changes to), right-click and select **Insert New Branch**.
6. Move the pointer to create another leader, double-click, and add an arrow as in step 3.

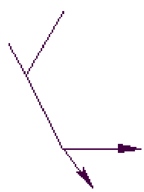


To attach a multi-jog leader to another annotation:

1. Click **Surface Finish Symbol** on the Annotation toolbar, or click **Insert**, **Annotations**, **Surface Finish Symbol**.
2. Click in the graphics area to place a symbol and click **OK**.
3. Select the beginning endpoint of the multi-jog leader and drag it to the surface finish symbol.

   A dock point highlights at the base of the symbol.
4. Drop the multi-jog leader endpoint on the surface finish symbol dock point.
5. Drag the surface finish symbol.

   The multi-jog leader moves with the surface finish symbol.



Annotations Now in the PropertyManager

Blocks

Block properties are now available in the **Block** PropertyManager. The name of the selected block appears in a **Name** box.

In the **Text Display** section, you can choose **Normal**, **All**, or **None**.

Layers are now handled through the existing Layer toolbar, so the **Layer** box that appeared in the **Block Properties** dialog box does not appear in the **Block** PropertyManager.

Hole Callouts

The **Hole Callout** tool now uses the **Dimension** PropertyManager. You can select either the tool or the hole first. The hole callout contains the hole diameter and depth.

Center Marks

The properties for center marks are now available in the **Center Mark** PropertyManager.

Center marks rotate automatically when the view is rotated. You can also rotate a center mark individually, specifying the degrees of rotation.

To insert a center mark into a drawing view:

1 Open **drw_center_mark.slddrw**.

2 Click **Center Mark** on the Annotation toolbar, or click **Insert**, **Annotations**, **Center Mark**.

   The **Center Mark** PropertyManager opens and prompts you to select a circular edge or an arc. The pointer changes to .

3 Select the center circle.

   A center mark appears as shown in the first example, with lines of length 2.5mm.

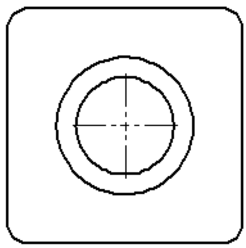
4 Under **Display attributes**:

   - Clear **Use document's defaults**
   - Under **Mark size**, click the up arrow to increase the size to 12.50mm

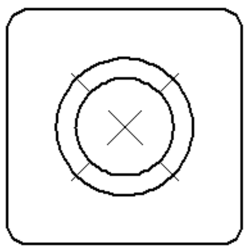
   A preview of the extended lines appears in the drawing view.

5 Under **Angle** , enter 45deg, then click **OK** .

   The center mark rotates 45 degrees counterclockwise as shown in the example at right.



Center mark using default values



Center mark with extended, rotated lines

Notes

Functionality for note properties is now in the PropertyManager. Changes in functionality in notes include:

- **Type text on screen**. The text appears in the graphics area as you type.
- **Layer**. Use the Layer toolbar to place notes and other annotations in named layers.
- **Tag**. A tag is an attribute of a note in a Block imported from AutoCAD. The tag can be **Read Only** or **Invisible**, or both.

To insert a note into a document:

1. Open a new drawing and click **Note** on the Annotation toolbar, or click **Insert**, **Annotations**, **Note**.

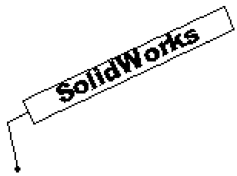
   The **Note** PropertyManager appears.
2. Under **Arrow/Leaders**, click **Leader** and **Bent Leader** , select the filled circle from the **Arrow Style** list.
3. Under **Text Format**, do the following:
   - Type **Angle** - 25deg
   - Click **Center Align**
   - Clear **Use document's font** and click **Font**.
   - In the **Choose Font** dialog box, choose the following:

     Font - Arial Black

     Font style - Bold

     Height - Click **Points** and select 20 from the list

     Click **OK**



4. Under **Border**, select **Box** and **Tight Fit**.
5. Click in the graphics area to place the leader, and again to place the note.
6. Type **SolidWorks**. The text appears in the graphics area.
7. Click **OK** .

You can drag the note and leader to any position. To edit the text, double-click the note. To edit properties, select the note and edit in the **Note** PropertyManager.

10

Import and Export

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

- General information
- DXF/DWG files
- STEP files

General Information

Import Diagnosis and Improve Geometry

The **Import Diagnosis** dialog box and the **Improve Geometry** menu item have been moved to a new **Import Diagnosis** PropertyManager. The functionality of these items has not changed.

To open the **Import Diagnosis** PropertyManager, right-click an imported feature in the FeatureManager design tree, and select **Diagnosis**. The gap and face check results appear under **Gaps** and **Faces**.

To improve and rebuild the feature geometry, click **Improve Geometry**. Results are still reported in the **Improve Geometry** dialog box.

Import Options

There are two new import options to help you properly import data:

- Perform full entity check and repair errors
- Customize tolerance

Entity Check and Repair

For ACIS®, IGES, STEP, and VDAFS files, you now have the option to perform a full entity check and repair errors (default is enabled). With this option enabled, import performance is slower because the software spends more time checking and repairing the model entities. If the quality of the imported data is good, you may not have to enable this option.

To set the entity check and repair option:

1. Click **File**, **Open**.
2. Set **Files of type** to either ACIS, IGES, STEP, or VDAFS, then click **Options**.

   The **Import Options** dialog box appears. Note that the new option, **Perform full entity check and repair errors**, is enabled.
3. Click **OK**.
4. Browse to the file to import and click **Open** to perform the check.

Customize Tolerance

For ACIS, IGES, STEP, and VDAFS files, you now have the option to customize the tolerance when you import models with very small entities (smallest values on the order of 1.0e-6 to 1.0e-7 meters). With this option disabled (default), SolidWorks uses internal tolerance settings which are too large to properly import and display these small models.

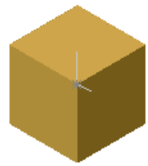
To customize the tolerance when importing models with very small entities:

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

   The **Open** dialog box appears.

2. Set **Files of type** to **IGES Files (*.igs, *.iges)**, select **SmallPart.igs**, and click **Open**.

   The file opens, but all the surfaces were not properly imported because of their very small size.



3. Close the file without saving changes.

   Now you will enable the **Customize tolerance** option to see the difference.

4. Click **File**, **Open**.

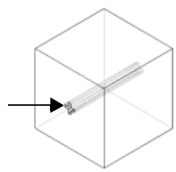
   The **Open** dialog box appears.

5. Set **Files of type** to **IGES Files (*.igs, *.iges)**, select **SmallPart.igs**, then click **Options**.

   The **Import Options** dialog box appears.

6. Select the **Customize tolerance** check box, then move the slider completely to the left towards **Tight**.

   This tightens the tolerance to 1.0e-8m. The tolerance size appears in the **Tolerance** box.



7. Click **OK**, then click **Open**.

   The file opens with all the very small surfaces properly displayed.

8. Click **Hidden in Gray**  on the View toolbar, or **View**, **Display**, **Hidden in Gray** to better display the surfaces.

DXF/DWG Files

Import Topics

AutoCAD Attributes

SolidWorks now supports attributes when you import DXF/DWG files. Previously, attributes were imported as note text only.

To see an example of attribute import support:

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

   The **Open** dialog box appears.

2. Set **Files of type** to **Dwg Files (*.dwg)**, select **Attributes.dwg**, and click **Open**.

   The **DXF/DWG Import - Document Type** dialog box appears.
   Import to a new drawing is selected by default.

3. Click **Next**.

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

4. Click **Next**.

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

5. Make sure that **Import all data to the sheet** is selected. Select the **Explode blocks** check box, then click **Finish** to import the file.

6. Zoom to the title block and in the **DRAWING NO.** box, click the number **12345-67**.

   The **Note** PropertyManager appears. Under **Block Settings**, the attribute **Tag name** shows **DRAWING#**. In previous versions, the attribute value 12345-67 imported only as note text.

7. Close the file without saving changes.

AutoCAD Blocks

You now have the option to import AutoCAD® blocks as fully-editable SolidWorks blocks. This capability makes them selectable while preserving their attributes and values. All attributes from the AutoCAD drawing display as annotations in the SolidWorks block. Previously, these entities were exploded into basic geometry, losing the attribute tag name.

To import AutoCAD blocks as SolidWorks blocks:

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

   The **Open** dialog box appears.

2. Set **Files of type** to **Dwg Files (*.dwg)**, select **AcadBlocks.dwg**, and click **Open**.

   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 Click **Next**.

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

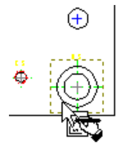
5 Make sure that **Import all data to the sheet** is selected.

NOTE: If you want to explode the AutoCAD blocks when you import them, select the **Explode blocks** check box. SolidWorks then imports the blocks as individual entities.

6 Click **Finish**.

The drawing opens with the AutoCAD blocks imported as blocks.

7 Move the pointer over a block.



The pointer changes to , indicating you are over a block. The block is highlighted with a dashed line.

8 Select the block.

The **Block** PropertyManager opens. You can now edit the block attributes.

9 Click **OK** to close the **Block** PropertyManager.

AutoCAD Fonts

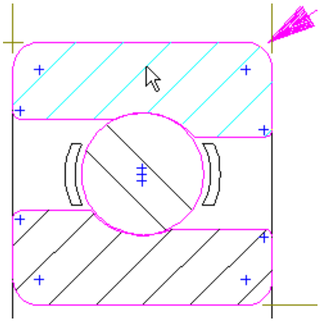
SolidWorks now accurately displays AutoCAD fonts when you import DXF/DWG drawing files, even if AutoCAD is not installed on your computer. SolidWorks has added True Type fonts to support the AutoCAD fonts.

Crosshatches

When you import DXF/DWG files, associative crosshatches with a closed boundary now import as grouped lines that use the SolidWorks area hatch format. Previously, hatches were imported as individual lines. AutoCAD version 14 files and higher are supported.

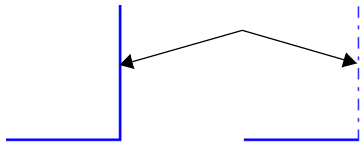
If you import an AutoCAD R12 or R13 file with crosshatching, you can combine the crosshatching into a SolidWorks block annotation. See **AutoCAD Blocks** on page 10-4 for more information.

You also have the option to not import crosshatches from the original AutoCAD file. In this case, all crosshatches are permanently deleted from the new SolidWorks document when you import the DXF or DWG file. The crosshatches are not restored if you later save the SolidWorks document as a DXF/DWG file. This option is useful when converting 2D files to 3D files. See **Converting a 2D Drawing to a 3D Part** on page 3-5 for details.



DXF Files with Dashed Line Font

When you import a DXF file as a SolidWorks part, any line with a dashed line font is now imported as a construction line. Previously, this geometry was displayed as a solid line when imported as a part.



Data Units

When you import a DXF/DWG file into SolidWorks, you can now specify the following new **Data units**:

Data unit	Value
Angstroms	1.0e-10 meters
Nanometers	1.0e-9 meters
Microns	1.0e-6 meters
Microinches	1.0e-6 inches
Mils	1.0e-3 inches

- Import to a Drawing

When you import a DXF/DWG file to a new drawing, you specify these data units in the **DXF/DWG Import - Document Settings** dialog box, under **Input file properties**.

- Import to a Part

When you import a DXF/DWG file to a new part, you specify these data units in the **DXF/DWG Import - Part Document Options** dialog box, under **Units of imported data**.

Export Topics

Export Option 1:1

You now have the option to export a SolidWorks drawing document using a model geometry scale of 1:1. The paper or sheet scale is not normally used when you enable this option.

Previously, SolidWorks software always exported the DXF/DWG file based on the sheet scale. The drawing border was the correct size and items dimensioned correctly, but the actual geometry line length was at the sheet scale, and could not be used with CAM software.

To export a SolidWorks drawing document as a DXF/DWG file with a 1:1 model geometry scale:

1 Open **ExportScale.slddrw**.

2 Click **File**, **Save As**.

The **Save As** dialog box appears.

3 Set **Save as type** to **Dwg Files (*.dwg)** or **Dxf Files (*.dxf),** then click **Options**.

The **DXF/DWG Export Options** dialog box appears.

4 Select the **1:1 Scale output** check box.

The **Base scale** box is activated and displays **view scale = 1/2: count = 2**.

Base scale refers to the basis used for the 1:1 scale output of the geometry, based on the various drawing view scales on the sheet. If you have a view selected, the **view scale** and **count** values for this selected view appear in the **Base scale** box. If no view is selected, the **view scale** with the highest **count** is displayed by default.

5 Display the base scale menu items:

- **sheet scale = 1/2: count = 1**
- **view scale = 1/3: count = 1**
- **view scale = 1/2: count = 2**
- **view scale = 1/1: count = 1**

The menu displays the sheet scale and view scales. There can be multiple view scales on a sheet. **Count** indicates the number of occurrences of this scale in the drawing document.

6 Make sure the **Custom Map SolidWorks to DXF** check box is cleared, then click **OK**.

7 Click **Save** in the **Save As** dialog box.

The drawing document is saved with a model geometry scale of 1:1 for the chosen views, with the remainder of the sheet scaled accordingly.

AutoCAD Line Fonts

When you export SolidWorks drawing documents to DXF/DWG files, you now have the option to map SolidWorks line fonts to the original AutoCAD stock line types. Line font thickness is mapped to the closest AutoCAD line weight value.

NOTE: AutoCAD line weight value mapping is only supported for AutoCAD **Version** setting **R2000** and higher in the **DXF/DWG Export Options** dialog box.

To export a SolidWorks drawing document as a DXF/DWG file using AutoCAD line types:

1. With a SolidWorks drawing document open, click **File**, **Save As**.

   The **Save As** dialog box appears.

2. Set **Save as type** to **Dwg Files (*.dwg)** or **Dxf Files (*.dxf)**, then click **Options**.

   The **DXF/DWG Export Options** dialog box appears.

3. Select **AutoCAD Standard Styles** from the **Line styles** box.
4. Click **OK**, then click **Save** in the **Save As** dialog box.
5. If a message box appears about scaling and dimensions, click **OK**.

   The drawing document is saved using AutoCAD line types.

STEP Files

Import

SolidWorks now supports import of wireframe geometry from STEP AP203 and AP214 files. When you import a STEP file that contains wireframe geometry, the **STEP Wireframe Import** dialog box appears with the following options:

- **Import STEP Curves as 3D Curves**. Imports STEP curves as 3D curves.
- **Import STEP Curves as 3D Sketches**. Analytic planar sketches (circles, ellipses, parabolas) import into individual sketches. All other sketch entities (lines, splines, points) are imported into a single 3D sketch.

Export

There is a new **STEP Export Options** dialog box that lets you set export options for part or assembly documents. There are new **3D curves** and **Export sketch entities** export options.

The **Set STEP configuration data** check box, which appears when you save documents to the **STEP AP203 (*.step)** format, was previously located in the **Save As** dialog box.

NOTE: For STEP files exported from SW2001Plus:

If you select the **3D curves** or **Export sketch entities** options, you *cannot* open these files in earlier versions of SolidWorks.

If you select the **Solid/Surface geometry** option, you *can* open these files in earlier versions of SolidWorks.

To export to a STEP file and set the export options:

1 Open **StepExport.sldprt**.

2 Click **File**, **Save As**.

The **Save As** dialog box appears.

3 Set **Files of type** to **STEP AP203 (*.step)**, then click **Options**.

The **STEP Export Options** dialog box appears, with the following items:

- **Solid/Surface geometry**. Exports the geometry as all solids and surface bodies, as done in previous versions of SolidWorks.
- **3D curves** with the **Export sketch entities** check box cleared. Exports the solid and surface bodies as wireframe entities. All 3D curves (composite curves, 3D wireframes, imported curves, and so forth) are also saved.
- **3D curves** with the **Export sketch entities** check box selected. The **Export sketch entities** item is available as an option to **3D curves**. Exports all the items in **3D curves**, plus all 2D and 3D sketches in the document.

4 Select **3D curves** and **Export sketch entities**, then click **OK**.

5 Click **Save** and save the file as **StepExport.STEP**.

Because you selected the **3D Curves** option:

- All edges of the solid and surface bodies are exported as wireframe entities.
- All 3D curves in the part are exported.
- No surface or solid from the model is exported.

Because you selected the **Export sketch entities** option, in addition to the above, all 2D and 3D sketches are also exported.

6 Open **StepExport.STEP** to view the results.

The **STEP Wireframe Import** dialog box appears.

- If you select **Import STEP Curves as 3D Curves**, all 3D curves are imported as 3D curves.
- If you select **Import STEP Curves as 3D Sketches**, all 3D curves are imported as 3D sketches. Additionally, all lines and splines are imported into a single 3D sketch.

Because you cannot group the sketch elements together in a STEP file, when you open the file in SolidWorks, circles, ellipses, and parabolas are imported into individual 2D sketches.

11

Sheet Metal

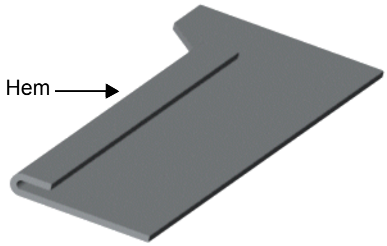
This chapter describes enhancements to the SolidWorks sheet metal functionality including:

- ❑ New sheet metal features such as:
  - Hem
  - Jog
  - Break Corner
- ❑ Improved sheet metal features such as:
  - Corner Treatments
  - Partial Miter Flange
  - Open Loop Miter Flange

New Sheet Metal Features

Hem

The Hem feature adds a hem to your sheet metal part at a selected edge.



Here are some additional items to note about the Hem feature:

- ❑ The selected edge must be linear.
- ❑ Mitered corners are automatically added to intersecting hems.
- ❑ The thickness is automatically linked to the thickness of the sheet metal part.
- ❑ If you select multiple edges to add a hem, the edges must lie on the same face.

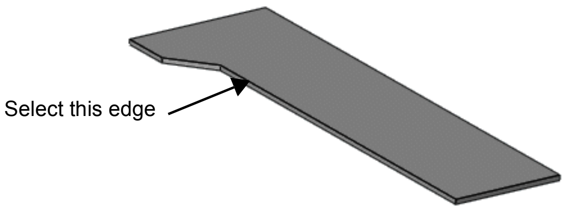
To create a Hem feature in a sheet metal part:

1 Open **bracket.sldprt**.

2 Click **Hem** on the Sheet Metal toolbar, or click **Insert**, **Sheet Metal**, **Hem**.

The **Hem** PropertyManager appears.

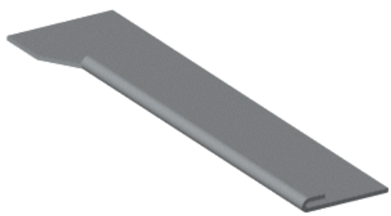
3 Under **Edges**, do the following:



- Select the edge as shown.
- Set the bend position to **Material Inside**.

4 Under **Type and Size**, do the following:

- Select **Open** as the hem type.
- Set the **Length** to 10mm.
- Set **Gap Distance** to 2mm.



5 Click **OK**.

The hem is created on the part.

Jog

The Jog feature adds material to a sheet metal part by creating two bends from a sketched line.

Here are some additional items to note about the Jog feature:

- The sketch must contain only one line.
- The line does not need to be horizontal or vertical.
- The jog can have a zero offset value.
- You can fix the projected length of the jog so that the part retains its original length after the jog is created.

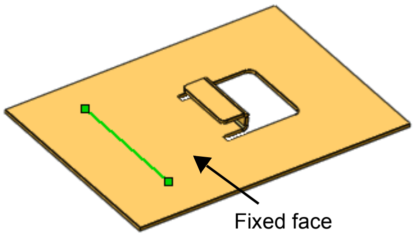
To create a Jog feature in a part with a Base-Flange feature:

1 Open **jog.sldprt**.

2 Select **Sketch6** in the FeatureManager design tree.

3 Click **Jog** on the Sheet Metal toolbar, or click **Insert**, **Sheet Metal**, **Jog**.

The **Jog** PropertyManager appears.



4 Select the face above the sketched line as the **Fixed Face**.

5 Under **Jog Offset**, do the following:

- Set **End Condition** to **Blind**.
- Set **Offset Distance** to 10mm.

NOTE: If you set the **Jog offset distance** to zero, the bends will be tangent, forming an "S" curve. If there is not enough material in your sheet metal part to create a flat surface jog feature, the offset must be zero.

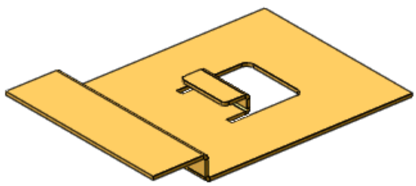
- Set the **Dimension position** to **Outside Offset**.
- Select the **Fix projected length** check box to preserve the original length of the part.

6 Set the **Jog Position** to **Bend Centerline**.

7 Use the default value for the **Jog Angle** of 90 degrees.

8 Click **OK**.

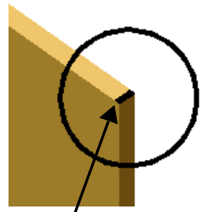
The jog is added to the sheet metal part.



Break Corner

The Break Corner feature cuts away material from a sheet metal part on an edge or a face.

Here are some additional items to note about the **Break Corner** feature:

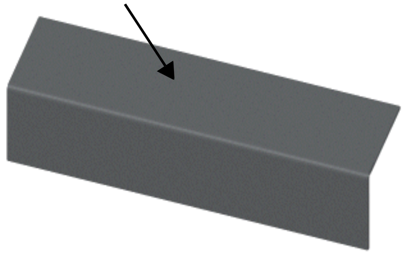


- ❑ Only corner edges, flange faces, or both may be selected.
- ❑ When selecting the corners to break, the software automatically identifies the edges or faces available when you pass the pointer over them, making selection easier.
- ❑ There are two different break types: chamfer or fillet.

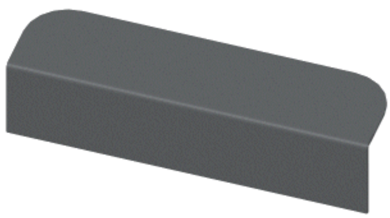
To create a Break Corner feature in a sheet metal part:

1. Open **break_corner.sldprt**.
2. Click **Break Corner** on the Sheet Metal toolbar, or click **Insert**, **Sheet Metal**, **Break Corner**.

   The **Break Corner** PropertyManager appears.
3. Under **Parameters**, do the following:
   - Click the face as shown.
   - Set the **Distance** to 20mm.
   - Select **Fillet** as the **Break type**.
4. Click **OK**.



The part appears with filleted edges.



Improved Sheet Metal Features

Partial Miter Flange

If you create a miter flange on a sheet metal part, you now have the option to create a partial miter flange. Instead of creating a miter flange across the entire edge of a sheet metal part, you can specify the offset of the flange.

Here are some additional items to note about the partial miter flange feature:

- At least one offset distance must be greater than zero.
- Negative offset distances are not allowed.

To create a partial miter flange feature in a part with a Base-Flange feature:

1 Open **partial_miter_flange.sldprt**.

Notice the miter flange on the part extends to the full length of the edges.

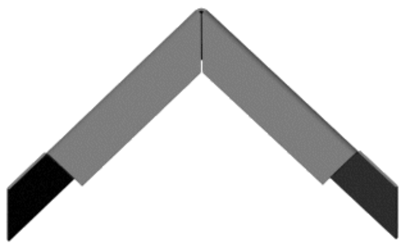
2 Right-click **Miter Flange1** in the FeatureManager design tree, and select **Edit Definition**.

The **Miter Flange1** PropertyManager appears.

3 Under **Start/End Offset**, set the **Start Offset Distance** and **End Offset Distance** to 10mm.

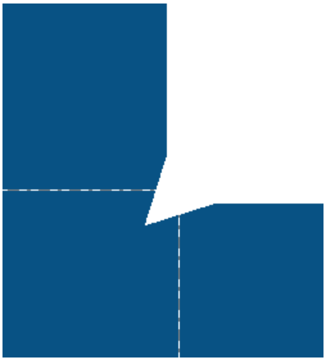
4 Click **OK** .

The miter flange is updated and bend reliefs are automatically added.

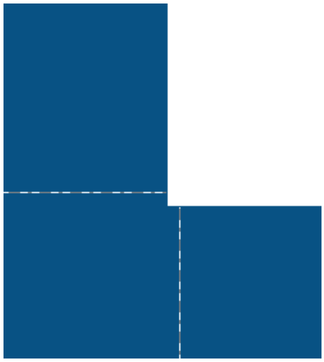


Corner Treatments

If you flatten a sheet metal part using the **Flattened**  tool, you can turn corner treatments on or off. In previous releases of SolidWorks, corner treatments were automatically enabled and could not be turned off. When your part is in flattened mode, right-click **Flat-Pattern** in the FeatureManager design tree and select **Edit Definition**. In the **Flat-Pattern** PropertyManager, select the **Corner treatment** check box. (This is the default.)



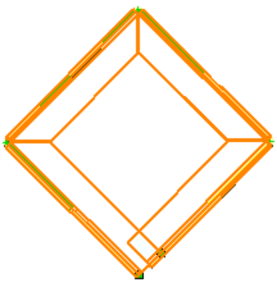
Without corner treatment



With corner treatment

Open Loop Miter Flange

In open loop sheet metal parts, the miter flange feature now trims corners properly. The beginning and end of the miter flange meet correctly so that the feature is added appropriately.



Miter Flange with incorrect corner trim

Miter Flange with correct corner trim

A

SolidWorks 2001 Service Pack Enhancements

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

Assemblies

Cam-Follower Mates

The cam-follower mate functionality allows a cylinder or plane to have a tangent mate to a closed series of tangent surfaces, such as would be found on a cam. It allows a point to have a coincident mate to a closed series of tangent surfaces.

Smart Fasteners

Smart Fasteners allows you to add standard bolts and screws to holes in your assembly with a single command. The fasteners are automatically mated. You can add nuts and washers to the fasteners, and edit the fasteners to change type, length, drive type, and thread display.

NOTE: Smart Fasteners requires the use of the SolidWorks Toolbox add-in. Please contact your reseller for local pricing and availability

Drawings and Detailing

Balloons

You can specify bent leaders separately for notes and for balloons. Bent leader options were added to **Tools, Options, Document Properties, Balloons**. Previously, the bent leader options for notes also controlled balloons.

Blocks

The **Block Properties** dialog box includes a check box for displaying **Note Text**. Right-click a block and select **Properties**. Under **Note Text** in the **Block Properties** dialog box, clear the **Display** check box to hide all notes and balloons in the block. Select the check box to display the text again.

Broken-Out Sections

You can create broken-out sections in drawing views by sketching a closed profile and specifying a depth. Material is removed to the specified depth to expose inner details of the model. The profile is generally a spline, but it can be any sketched closed profile. You can specify the depth by entering a value or by selecting an edge in a related view.

Cosmetic Threads

Cosmetic threads were inserted automatically into drawings in SolidWorks 2001. Cosmetic threads are now inserted automatically for part drawings, but not for assembly drawings. A **Cosmetic thread** check box was added to the **Insert Model Items** dialog box for manual insertion in assembly drawings.

Hole Wizard Dimensions

Check boxes for **Hole Wizard Profiles** and **Hole Wizard Locations** were added to the **Insert Model Items** dialog box under **Dimensions** for greater control in inserting dimensions into drawings.

Reference Dimensions

Reference dimensions are now automatically hidden when a feature is suppressed. The dimensions are shown again when the feature is unsuppressed.

eDrawings 2.0

SolidWorks announces the release of eDrawings 2.0 and eDrawings Professional.

With eDrawings 2.0, you can now do the following:

- Create eDrawing files from SolidWorks part or assembly documents, as well as drawing documents
- Open SolidWorks documents in the eDrawings Viewer
- Save eDrawing data to SolidWorks documents in SolidWorks
- View models using standard orthographic views

With the optional eDrawings Professional, you can do the following:

- ❑ Markup files
- ❑ Measure models
- ❑ Move components in files
- ❑ Create cross section views in files

See http://www.solidworks.com/edrawings for more information.

Features

Lofts

The behavior of lofts has changed. If the **Maintain Tangency** check box under **Options** is checked, the tangent faces are merged. If the **Maintain Tangency** option is not checked, the faces are not merged and the surfaces adjust to an angular tolerance of one half a degree. Prior to the SP3 release, if the **Maintain Tangency** option is checked, the faces are not merged. Any lofts using **Maintain Tangency** that were created with the SP2 release or before exhibit this behavior.

Hole Wizard

Helicoil® Inch and **Helicoil Metric** are now available as standards for **Tap** holes.

Fundamentals

Selection Filter

You have the option to filter for blocks using the selection filter.

Web Folders

When you save assembly documents to a web folder, you have the option to retain the same folder structure in the web folder as the current path for all component documents.

Import/Export

Autodesk Inventor

Support now includes through version Autodesk Inventor R4.

CATIA Graphics Files

You can open CGR part or assembly files in SolidWorks. CGR files contain only graphical information and are for viewing only.

DXF 3D

When you import a DXF 3D file that contains multiple bodies or an assembly, SolidWorks creates an assembly document.

IGES

In earlier versions of SolidWorks, a color was assigned to each face in an imported IGES model. Beginning with Service Pack 4, the most prevalent color in the IGES model becomes the part color, and the remaining faces are assigned a face color.

For example, import a block with five blue faces and one red face. Beginning with Service Pack 4, blue is the part color, and the one red face is assigned a face color.

Pro/ENGINEER

The SolidWorks Pro/E translator is available as an add-in. This translator exports SolidWorks part or assembly documents to Pro/ENGINEER part or assembly files.

Solid Edge

SolidWorks supports import of Solid Edge parts up to Solid Edge version 9.

Unigraphics II

You can import parts and assemblies from Unigraphics II version 10 through version 17.

Virtue Translator

The Virtue translator is available as an add-in. This add-in exports SolidWorks part or assembly documents to Virtue (VTU) streaming graphics files.

Sheet Metal

Bend Tables

SolidWorks includes two metric bend tables with default radius and thickness values. The bend allowance values are as follows:

- ❑ **metric base bend table.xls** - no bend allowance values are included.
- ❑ **table4 - metric bend allowance.xls** - sample bend allowance values are included.

Closed Corner

In the **Closed Corner** PropertyManager, you can select a **Corner type** of **Butt**, **Overlap**, or **Underlap**.

SolidWorks Utilities

SolidWorks Utilities is a set of applications that allows you to examine and edit individual parts, and compare the features and solid geometry of pairs of parts. The set of applications is a single, separately purchased add-in to the SolidWorks software; please contact your reseller for local pricing.

Surfaces

The **Surface Fill** PropertyManager has a new **Quality** control. This control can improve the appearance of the filled surface by changing the number of patches defining the surface.

Index

D

E

F

G

H

I

SolidWorks 2001Plus Document Order Form

Getting Started Manual

SolidWorks will ship the SolidWorks 2001Plus Getting Started manual, as ordered on this form, to registered customers who have upgraded or are on the SolidWorks Subscription Service plan.
Orders can be processed only if *all* information is received. Type or print clearly.

Quantity required____

Serial Numbers____

Include a serial number for each set requested. (For example: 0000 0000 1234 1234)
Attach additional sheet if necessary.

Language required:

- ❑ English
- ❑ French
- ❑ German
- ❑ Italian
- ❑ Russian
- ❑ Spanish

Mail your request to:

SolidWorks Corporation
Attn: Books
300 Baker Avenue
Concord, MA 01742 USA

Fax your request to:

+1-978-371-4996

Order online:

http://www.solidworks.com/docorders

Please send my package to:

Company____

Attention____

Address____

____

________________________________**Country**______________________

Telephone Number__________________