SolidWorks 2006

What's New

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Introduction

About this Book

This book highlights and helps you learn the new functionality in the SolidWorks® 2006 software. It introduces concepts and provides step-by-step examples for many of the new functions.

This book does not cover every detail of the new functions in this software release. For complete coverage of the new functions in the SolidWorks 2006 software, refer to the *SolidWorks Online User's Guide* by clicking **Help**, **SolidWorks Help Topics**.

Intended Audience

This book is for experienced users of the SolidWorks software and assumes that you have a good working knowledge of an earlier release. If you are new to the software, you should read the *Getting Started* book, complete the *Online Tutorial* lessons, then contact your reseller for information about SolidWorks training classes.

Additional Resources

Other resources where you can learn about the new functionality of the SolidWorks software include:

- **SolidWorks 2006 What's New Highlights**. This book provides the highlights of the new functionality in the SolidWorks software. This book is available in printed format for subscription customers.
- **Interactive What's New**. Click next to new menu items and the title of new and changed PropertyManagers to read what is new about the command. A help topic appears with the text from this manual.
- **Online Help**. The online help contains brief descriptions of the new functionality with links to the complete, related help topics.

Late Changes

Due to software deadlines, this book may not include all of the enhancements in the SolidWorks 2006 software. For enhancements that are not in this book, refer to the *SolidWorks Release Notes* by clicking **Help**, **SolidWorks Release Notes**.

Using This Book

Use this book with the part, assembly, and drawing files provided. Read this book from beginning to end, and open the proper part, assembly, or drawing document for each example.

To use the example files:

1. Install the SolidWorks 2006 software.
2. Be sure to select the option to install the **Example Files**.

   The example files are placed in the <*install_dir*>**samples****whatsnew** folder.

   Because some of the example files are used with more than one example, do not save changes to these files unless instructed to do so.

Conventions Used in this Book

This book uses the following conventions:

Convention	Meaning	Example
Bold	Headings or titles that are not menu items	**Measure**. Measure the distance between two entities.
Bold Sans Serif	Any SolidWorks tool, menu item, or example file	Click **Insert**, **Mate References**.
Italic	Refers to books and other documents, or emphasizes text	Refer to the SolidWorks *Read This First*.
	Tip	When you create a 3D model, first make the 2D sketch, then create the extruded 3D feature.

Moving to SolidWorks 2006

Converting Older SolidWorks Files to SolidWorks 2006

Opening a SolidWorks document from an earlier release may take longer than you are used to experiencing. After the file is opened and saved, subsequent opening time returns to normal.

The SolidWorks Conversion Wizard automatically converts all of your SolidWorks files from an earlier version to the SolidWorks 2006 format. To access the Conversion Wizard, click the Microsoft® **Start** button, select **All Programs**, **SolidWorks 2006**, **SolidWorks Tools**, **Conversion Wizard**.

The utility offers you the choice to back up all of your files before the conversion. If you choose to back up 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 convert, and leads you through the process.

At the end of the conversion process, two report files exist in the folder where 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

If you are a SolidWorks subscription customer, you can take advantage of SolidWorks service packs that are regularly posted on the SolidWorks Web site. These service packs contain software updates and enhancements to the SolidWorks 2006 software. To check for a new service pack, click **Help**, **Service Packs**, and click **Check**. Select the check box if you want the software to automatically check the SolidWorks Web site for a new service pack once a week.

1

SolidWorks Fundamentals

This chapter describes enhancements to SolidWorks fundamentals and user interface in the following areas:

- Viewports
- Appearance Callouts
- Cameras
- Mouse Wheel
- Selection
- View Toolbar
- Options
- Conversion Wizard

Viewports

You can view models and drawings through viewports with the **Window**, **Viewport** menu or tools on the Standard Views toolbar.

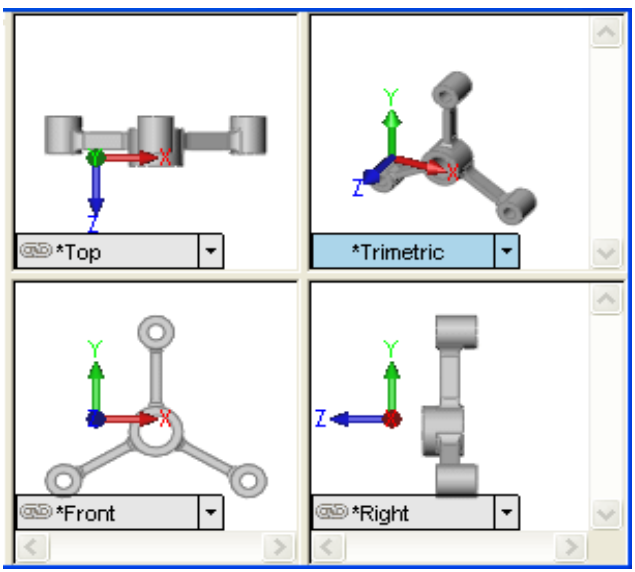
Single View		
Two View - Horizontal		Front and Top
Two View - Vertical		Front and Right
Four View		Front, Right, Top, and Trimetric
Link Views		Link orthogonal views

Specify either **First Angle** or **Third Angle** for Four View orientation under **Projection type for four view viewport** in **Tools**, **Options**, **System Options**, **Display/Selection**.

View pop-up menus in the lower left corner of each viewport display the current orientation. The menu for the active window is a different color than the menus in other viewports. If a viewport is linked, appears on the menu. Click the menu to choose an orientation or viewport tool.

When you switch from multiple viewports back to a single viewport, the model returns to the active view. If you then switch back to multiple viewports, the orientations are the defaults again.

You can change the orientation or rotation of any viewport independently as long as the viewports are not linked. When you link views (click **Link Views** on the View toolbar), zooming and panning are linked for all orthogonal views. Non-orthogonal views are not linked. View rotation using the pointer is not available in linked views, but if you rotate a view out of its orthogonal orientation, the view is no longer linked. If linked views is turned on, views become linked when rotated into orthogonal orientation.



Appearance Callouts

Appearance callouts display the colors and textures of the face, feature, body, and part under the item selected and are a shortcut to editing colors and textures. The display is similar to the Display Pane (see **Display States** on page 5-2) but the Display Pane does not show the states for faces.

To enable the callouts, click **View**, **Appearance Callouts**. With this menu item selected, a callout appears when you select a face, feature, or body in the graphics area or right-click a feature or body in the FeatureManager design tree and select **Appearance**, **Appearance Callouts**.

To edit a color or texture, select the display square in the callout.

You can specify file locations for **Appearance Folders** in **Tools**, **Options**, **System Options**, **File Locations**.

Cameras

You can now add cameras to model documents, then view the model from the camera perspective. For information on using cameras with SolidWorks Animator and PhotoWorks, see **SolidWorks Animator** on page 10-18 and **Camera** on page 10-15.

Camera View has been added to the View toolbar.

A typical procedure:

1 Add a camera.

2 Position the camera and define its properties in the **Camera** PropertyManager, or use the on-screen manipulators.

3 View the model from the camera's perspective.

Additional ways to use cameras:

- Split the screen and view the model in one window and the camera view in another.
- Move the camera through and into the model, and view the model from inside.

You can position the following items with on-screen manipulators (when the **Camera** PropertyManager is open):

- Camera
- Target
- Field of view

Controls in the **Camera** PropertyManager:

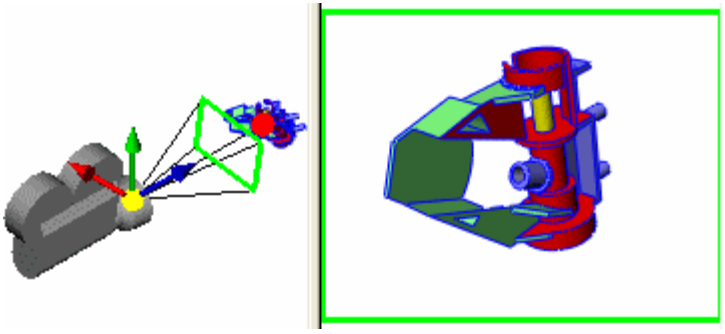
- Coordinate system (Cartesian or Spherical)
- Camera position and target point position
- Camera type (with or without fixed orientation)
- Perspective
- Field of view
- Depth of field (the effects are visible only in PhotoWorks renderings)
- Camera orientation (yaw, pitch, and roll)

To add a camera:

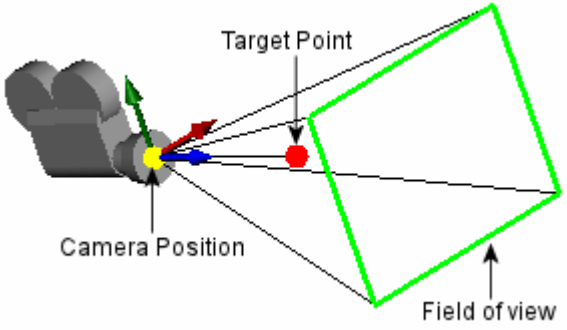
1 Open **fundamentals\sprinkler.sldprt**.

2 In the FeatureManager design tree, right-click **Lights and Cameras** and select **Add Camera**.

- **Camera1** appears under **Lights and Cameras**.
- The graphics area splits into two viewports with the camera in the left viewport and the camera view in the right viewport with a rectangle outlining the field of view.
- The **Camera** PropertyManager opens.

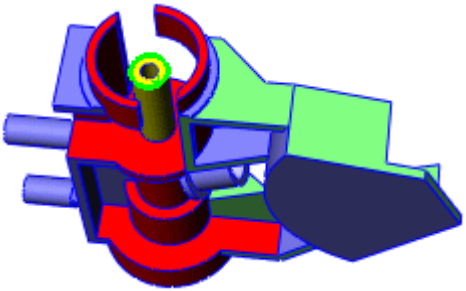


3 In the window on the left, drag the field of view rectangle in the graphics area to decrease its size and increase the display of the model.



4 Drag the camera closer to the model and around to the left to view this orientation.

5 Click to close the PropertyManager and return to a single viewport.



Mouse Wheel

The mouse wheel now zooms or scrolls in the window beneath the pointer. You can zoom a graphics window without clicking in it (for example, when the PropertyManager is active), and you can zoom a viewport in a multi-view layout without activating the view.

Selection

Invert Selection

When you select items (usually a small number) and click **Invert Selection** (Selection Filter toolbar) or **Tools**, **Invert Selection**, all other similar items in the document are selected and the original items are released.

Shift and Ctrl with Selection

Box and cross selection with **Shift** and **Ctrl** in the graphics area and in the FeatureManager design tree now work the same way as in Microsoft Windows Explorer.

- **Shift** with box selection selects everything within the box whether currently selected or not. Any prior selections outside the box remain unchanged.
- **Ctrl** with box selection inverts the current selection within the box. Any prior selections outside the box remain unchanged.

You can now initiate box or cross selection in the FeatureManager design tree by holding the pointer down in an empty region of the panel and dragging.

View Toolbar

Tools have been added to the View toolbar to toggle visibility of the following items, which are also available on the **View** menu.

Planes

Axes

Temporary Axes

Origins

Coordinate Systems

Curves

Sketches

3D Sketch Planes

3D Sketch Dimensions

All Annotations

Points

Routing Points

Parting Lines

Sketch Relations

Options

Save eDrawings data in SolidWorks document has been removed from **Tools**, **Options**, **System Options**, **General** and from **File**, **Save As**. The eDrawings Viewer can read native SolidWorks documents directly.

Conversion Wizard

The option **Create a backup for files to convert** is now on the last page of the wizard.

2

Sketching

This chapter describes enhancements to sketching in the following areas:

- General sketch entities
- Blocks
- 3D sketch functions
- Expanded spline control
- Formatting attributes for sketch entities
- Resolving sketch definition conflicts
- User Interface changes
- Transparency with raster data

General Sketch Entities

Offset with Parabolas and Ellipses

You can offset parabolas, ellipses, and partial ellipses. The interface and behavior are the same as offsets between other sketch entities.

Dynamic Sketch Updating

When creating lofts, you can edit the sketches as you create the features (see **Lofts** on page 3-5).

Tangency to a Face

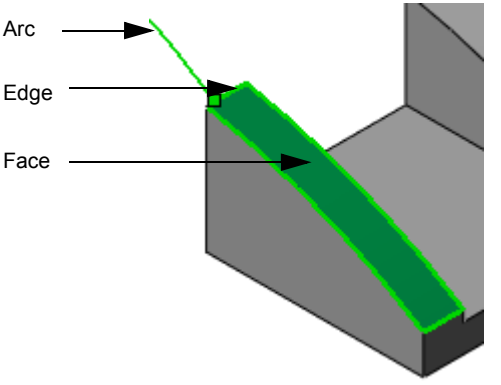
You can select a face or a surface, and apply a tangent relation between the face and another sketch entity.

To apply a tangent relation between a face and a sketch entity:

1 Open **Sketches\sketch_curve_tangent.sldprt**.

2 Right-click **Sketch3** and select **Edit Sketch**.

3 Press **Ctrl** and from the feature on the right, select the arc, the edge closest to the arc, and the sloping face, as shown.

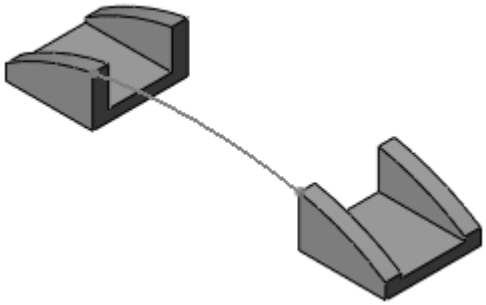


4 Click **Add Relation** (Dimensions/Relations toolbar) or **Tools**, **Relations**, **Add**.

5 In the PropertyManager, under **Add Relations**, click **Tangent Face**.

6 Click .

7 Repeat steps 3 to 5 to apply a tangent face relation between the arc and the left face.



Blocks

You create blocks from sketch entities. With blocks you can create and save sketches, and integrate them into multiple sketch structures. Blocks enable you to:

- Create layout sketches using a minimum of dimensions and relations.
- Select complex sketches when parts of one sketch overlap parts of another sketch.
- Move complex sketches within the graphics area.
- Add new sketches to an existing block.
- Create sketch structures by positioning sketches and using dimensions and relations.

Attributes and functions that apply to regular sketches apply to blocks.

Practical applications for blocks include:

- **Designing circuit boards**. Represent various components, such as integrated circuits on a circuit board.
- **Displaying and designing mechanical devices**. Sketch layouts using relations between blocks to create motion with mechanical devices such as actuators.

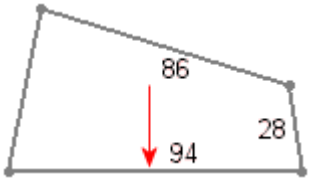
Block Attributes

- Create blocks using one or more sketch entities.
- Pre- or post-select entities to include in the block.
- Create nested blocks (blocks within blocks).
- Save blocks (**.sldblk** extension) and retrieve them in parts, assemblies, or blocks.
- Add or remove sketch entities, or change dimensions and relations to edit blocks.
- Browse for and drag blocks from the Task Pane.
- Add annotations to blocks.
- Mirror, scale, or pattern blocks.

To create blocks:

1 Open a new part.

2 Starting the horizontal line at the origin, create a sketch *approximating* the dimensions as shown.

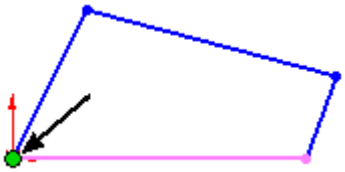


3 Select the horizontal line and click **Make block** (Block toolbar) or **Tools**, **Blocks**, **Make**.

4 In the PropertyManager, click .

5 Select the horizontal line, and add a **Horizontal** relation.

6 Select the vertex, as shown, and add a **Fix** relation.



7 Repeat steps 3 and 4 for each of the three remaining entities.

This creates a linkage mechanism comprised of four blocks that are identified in the FeatureManager design tree by icons.

8 Exit the sketch, save the part as **four_bar.sldprt**, and leave the part open to use later.

Editing Blocks

- Add or remove sketch entities from the block.
- Add or modify dimensions displayed during editing.
- Add relations between sketch entities within the same block. You cannot add relations to external sketch entities.
- Create multiple instances of the same block using the **Insert block** tool.

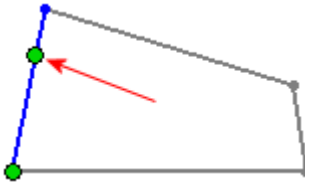
The **Insert block** command also enables you to add new blocks to the sketch you are editing.

- Explode blocks to remove block attributes from all the corresponding sketch entities. Exploding one instance of a block only affects that instance of the block.
- Sketch new entities while editing a block to create relations between the new sketch entities and the block you are editing.
- Use blocks to create an assembly. Convert the blocks to create the individual components within the assembly (see **Layout of Sketch Blocks** on page 5-6).

To edit blocks:

1 In **four_bar.sldprt**, edit **Sketch1**.

2 Select the left block as shown:



a) In the PropertyManager, under **Definition**, click **Edit**.

You can also right-click the block in the FeatureManager design tree, and select **Edit block**.

b) Click **Point** (Sketch toolbar) or **Tools**, **Sketch Entities**, **Point**, and add a point towards the top of the block, as shown.

3 Click the block confirmation corner , but do not close the sketch.

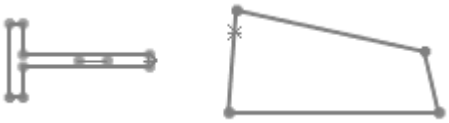
To insert and scale blocks:

1 While still in the sketch edit mode, click **Insert block** (Block toolbar) or **Tools**, **Blocks**, **Insert**.

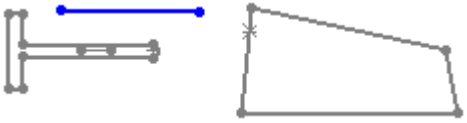
2 In the PropertyManager, under **Blocks to Insert**, click **Browse** and select **Sketches\hammer.SLDBLK**.

3 Under **Parameters**, set **Scale** to **.3**.

4 Click to position the hammer block near the sketch point, as shown, and then click .



5 Sketch a line with a length of approximately 60, as shown.



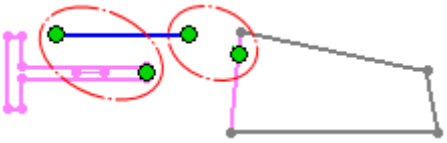
6 With the line selected, click **Make block** (Block toolbar) or **Tools**, **Blocks**, **Make**.

7 In the PropertyManager, click .

8 Add **Coincident** relations between:

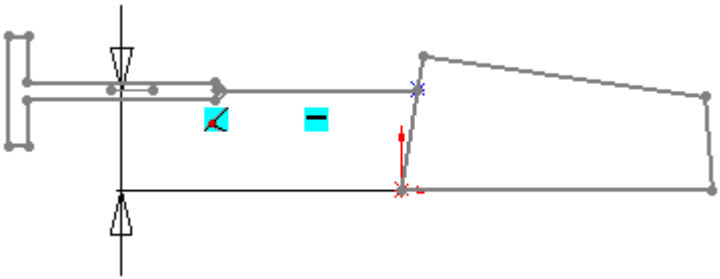
a) The sketch point on the upright block and the right end point on the line.

b) The center of the tangent arc on the hammer and the left end point on the line.



9 Click .

10 Add a dimension between the hammer centerline and the horizontal block.



11 Select the upper right vertex on the linkage, and move it to activate the hammer in a horizontal back-and-forth motion.

3D Sketching Functions

Sketch Planes with Manipulators

You can create planes while in a 3D sketch. Sketching on a plane in 3D is similar to sketching on a plane in 2D. You can resize or rotate planes using plane edges and manipulators. Double-click the plane to activate it and to display the manipulators; double-click outside the plane to de-activate it.

Additional 3D Sketch Entities

- Arcs (center point, 3 point, and tangent)
- Circles (center and perimeter)
- Rectangles

Arcs and circles allow you to create such entities as loft features or routing paths using single 3D sketches.

3D sketch entities use the same interface as their 2D counterparts. All 3D sketch entities are rooted where you start the sketch in 3D space.

Use the Sketcher Triad to limit the drag of entities.

To sketch in 3D using planes:

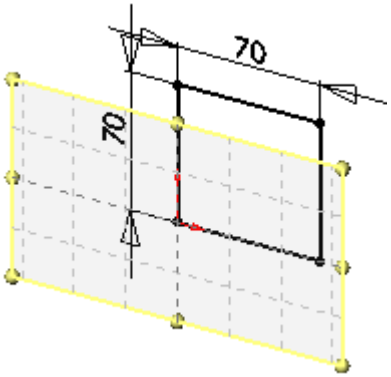
1 Open a part document and click **3D Sketch** (Sketch toolbar) or **Insert**, **3D Sketch**.

2 Double click **Front** in the FeatureManager design tree to add a sketch plane.

This enables you to sketch in 3D on a 2D plane.

3 Click **Rectangle** (Sketch toolbar), or **Tools**, **Sketch Entities**, **Rectangle**.

4 Starting at the origin, sketch a square and set the dimensions to **70**.



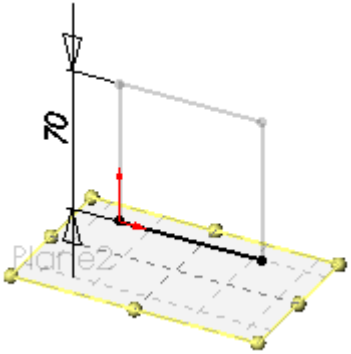
5 Click **Plane** (Sketch toolbar) or **Tools**, **Sketch Entities**, **Plane**.

6 In the PropertyManager:

a) Under **First Reference**, select either vertical line, and click **Perpendicular** .

b) Under **Second Reference**, select the bottom horizontal line, and click **Coincident** .

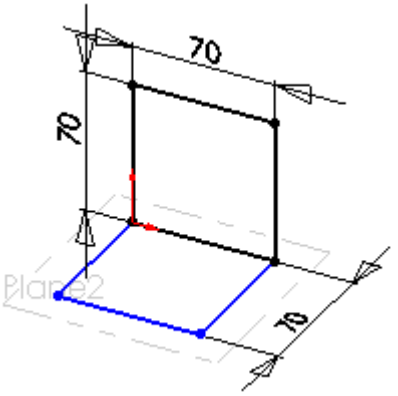
c) Click .



7 Click **Line** (Sketch toolbar) or **Tools**, **Sketch Entities**, **Line**.

8 Sketch two parallel lines, coincident to the bottom vertices of the square, and perpendicular to the vertical lines.

9 Then, connect the two end points and set the dimensions to **70**.



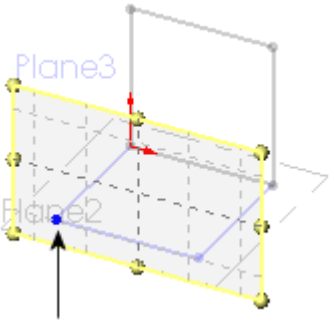
10 Click **Plane** (Sketch toolbar) or **Tools**, **Sketch Entities**, **Plane**.

11 In the PropertyManager:

a) Under **First Reference**, select one of the vertices connected in step 9.

b) Under **Second Reference**, select the **Front** plane, and click **Parallel**.

c) Click .



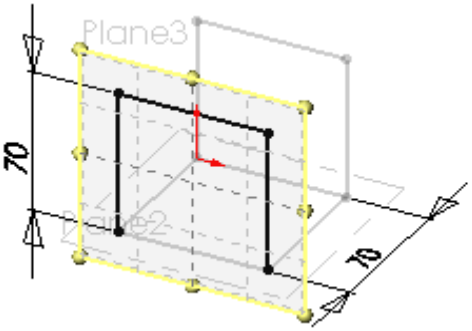
12 Click **Line** (Sketch toolbar) or **Tools**, **Sketch Entities**, **Line**.

a) Sketch two lines, coincident and perpendicular with each vertex.

b) Sketch a horizontal line connecting the two end points.

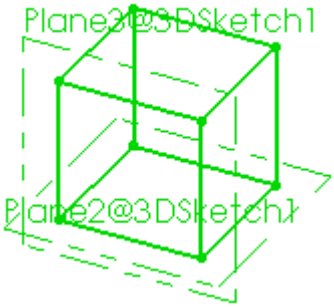
c) Set the dimension to **70**.

13 Double-click in the graphics area to clear the sketch plane.



14 Click **Line** (Sketch toolbar) or **Tools**, **Sketch Entities**, **Line**, and sketch two lines **AlongZ** to create the cube.

15 Close the sketch.

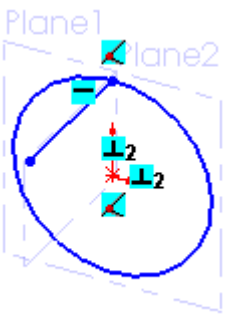


Planes also facilitate using 3D sketch entities using arcs to create lofts (see **Lofts** on page 3-5).

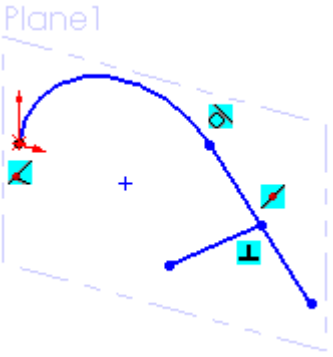
Sketch Relations

Additional sketch relations are supported with 3D sketching. For example:

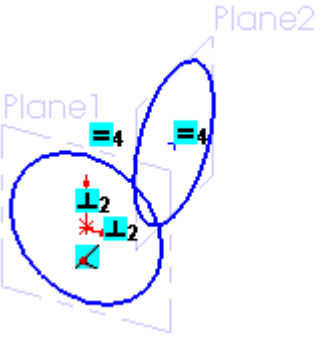
- Perpendicular relations between a line through a point on a surface.
- Relations between 3D sketch entities created on one sketch plane, and 3D entities created on other sketch planes.
- Midpoint relations.
- Relations between arcs and other sketch entities.
- Relations between arcs such as concentric, tangent, or equal.



Arc with perpendicular line created on perpendicular planes

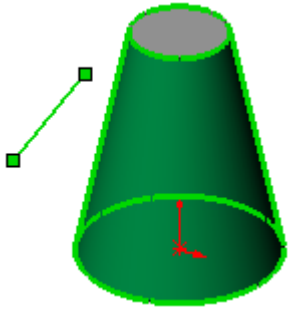


Arc with tangent line and perpendicular line to the midpoint between entities

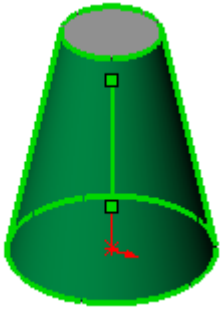


Equal arcs created on perpendicular planes

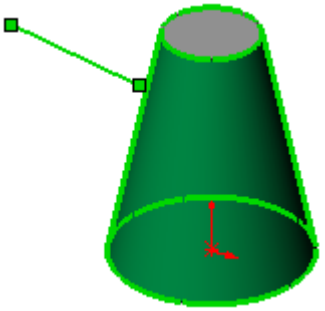
- Concentric and perpendicular relations between a cone and a 3D sketch line.



Conical entity and 3D line sketch



Concentric relation between entities



Perpendicular relation between entities

Dimension Control in 3D Sketches

Enhancements include:

- Dimensions between 3D sketch planes and other reference planes.
- Dimensions between 3D sketch planes and model faces or planar surfaces.
- Linear dimensions between a 3D sketch point and a 3D sketch plane, along the direction of the normal of the selected plane.
- Radius and diameter dimensions to arcs and circles.
- Linear dimensions between arcs and circles on one 3D sketch plane and any entity located normal to that plane.

3D Sketch Contour Selection

When creating lofts with 3D sketch contours as opposed to individual sketches, you can select one or more contours, as well as join contours while in the **Loft** PropertyManager (see **Lofts** on page 3-5).

Splines

Control Polygons

Control polygons are sequences of control points in space used to manipulate an object's shape. Control polygons display when you sketch and edit a 2D or 3D spline. Control points manipulate the shape of the spline.

Dragging control points as opposed to spline points localizes the area of change, giving you more precise control over the spline's shape.

To clear default display of control polygons:

1 Click **Options** (Standard toolbar) or **Tools**, **Options**.

2 Click **Sketch**.

3 Clear **Show spline control polygon by default**.

To display control polygons:

Right click the spline and select or clear **Display Control Polygon**

- or -

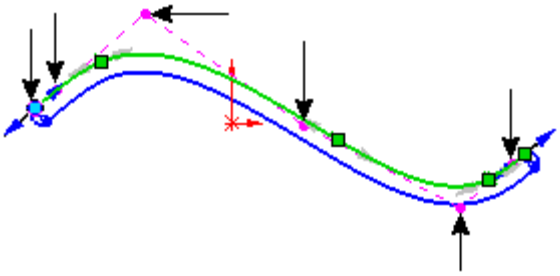
Select or clear **Display Control Polygon** (Spline Tools).

To use control polygons:

1 Open **Sketches\spline_control.sldprt**.

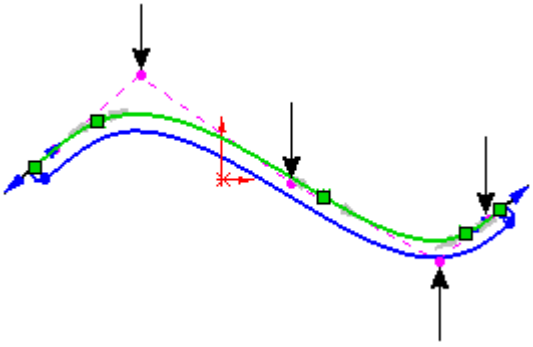
2 Under **Extrude1**, edit **Sketch1**.

3 Select the spline to display the PropertyManager and the control polygon.

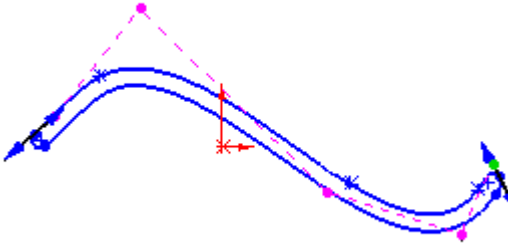


Control points on the spline control polygon are shown by the arrows

4 Drag the four points, as shown, to adjust the shape of the spline.



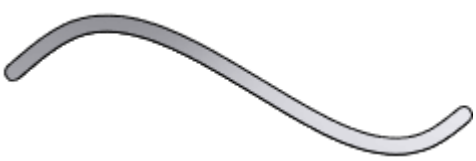
Starting position of the control polygon



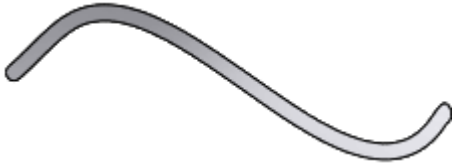
Ending position of the control polygon

5 Close the sketch.

Manipulating splines with control points causes subtle shape changes.



Original shape



Modified shape

Relax Spline

Relax Spline was added to address the following behavior encountered when dragging a spline point along the spline:

- Spline points behave as if they were clamped into position.
- Spline shape is extensively modified.

Select **Relax Spline** to smoothly drag spline points along the entity without adversely affecting the shape of the spline.

You cannot reverse the **Relax Spline** option once you select it.

To use relax spline selected:

1. Open **Sketches\relax_spline.sldprt**, right-click **Sketch1** and select **Edit Sketch**.

   The top spline has **Relax Spline** selected, while the bottom spline has it cleared.
2. Select the top spline and drag the middle point along the spline.

   The spline point moves along the entity because **Relax Spline** is selected.
3. Select the bottom spline, and drag the middle point along the spline.

   The spline point does not move along the entity because **Relax Spline** is cleared.

Dimension to Spline Handles

Using the **Smart Dimension** tool with spline handles, you can add dimensions for:

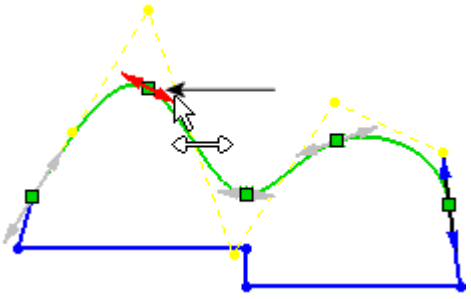
- **Tangent Magnitude**
- **Tangent Radial Direction**

This allows:

- Setting either value in the graphics area rather than the PropertyManager.
- Controlling tangent magnitude and direction in Design Tables.

To dimension to spline handles:

1. Open **Sketches\spline_handles.sldprt**.
2. Expand **Revolve1** to edit its sketch.
3. Select the spline to display the spline handles (handles are grey).
4. In the PropertyManager under **Parameters**, select **Tangent Driving**.
5. Select the second spline handle from the left, as shown:



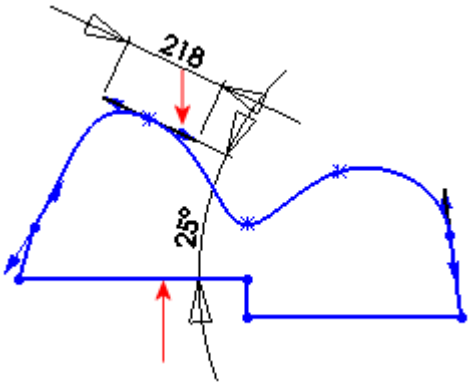
- Drag to adjust the tangent magnitude.
- Rotate to adjust the tangent dimension.

6 Click the spline point to set the handle, then right-click and select **Smart Dimension**.

7 Dimension as shown:

a) Select either arrow tip, and set the tangent magnitude to **218**.

b) Select the right arrow tip and the closest horizontal line, as shown, and set the tangent direction to **25**.



8 Close the sketch.

Formatting Attributes for Sketch Entities

You can assign color and font attributes to individual sketch entities. Capabilities are similar to drawings, including:

- Pre-or post-selection
- Switching back to the originally defined color
- Exporting DXF/DWG sketches

To change color and font attributes:

1 Open **Sketches\bracket.sldprt**.

2 Expand **Sketched Bend1**, right-click **Sketch3** and select **Edit Sketch**.

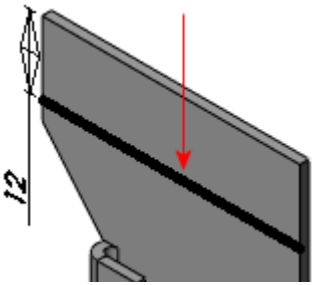
3 Select the bend line as shown.

4 Click **Line Color** (Line Format toolbar).

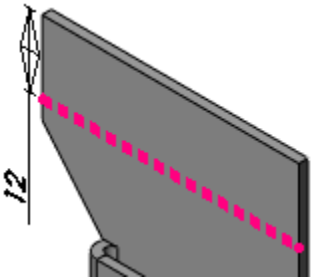
5 In the **Set Sketch Color** palette, select magenta then click **OK**.

6 Select the bend line again:

- Click **Line Style** (Line Format toolbar).
- Select the dashed line style, and click in the graphics area to apply the changes.



7 Select the bend line, click **Color Display Mode** (Line Format toolbar) to return the dashed line to its original color, and click in the graphics area to apply.



> The **Color Display Mode** does not affect **Line Style** or **Line Thickness**. You can change either of these attributes by using the appropriate line tool.

8 Click .

Resolve Conflicts

Resolve Conflicts generates a list of causes for over-defined sketches. The list is organized into **Solution Sets**. This tool enables you to delete a solution set of over defined dimensions or redundant relations without compromising your design intent.

Conflicts occur when sketches are over defined, and because the SolidWorks software allows some redundancy.

To resolve sketch conflicts:

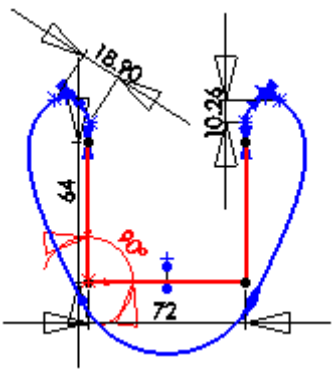
1 Open **Sketches\hood_scoop.sldprt**, and click **Close** in the **What's Wrong** dialog.

2 On the bottom right of the status bar, select **Over Defined** to open the PropertyManager.

3 Under **Conflicting Relations/Dimensions**, click **Diagnose** to display four **Solution Sets**.

Select to delete the **Solution Set** whose relations and dimensions do not conflict with your design intent

In this example, assume that the 90° is the dimension not required for your design intent.



4 Select **Solution Set 1**, click **Delete**, then click .

The sketch is no longer **Over Defined** and your design intent remains intact.

User Interface Changes

- **Move**, **Rotate**, **Scale**, and **Copy** sketch commands each opens its own PropertyManager.
- **Linear** and **Circular** sketch patterns use a PropertyManager that resembles the functions used in features.

Move, Rotate, Scale, and Copy

Capabilities and changes include:

- Enable **Object Snaps** while in the **Move**, **Rotate** and **Scale** commands.
- Duplicate prior **Move** or **Rotate** coordinates using **Repeat function**.
- Remove base point coordinates from all commands.
- Remove destination pointer from **Move** and **Copy** commands.
- Allow **Copy** while in the **Scale** command.

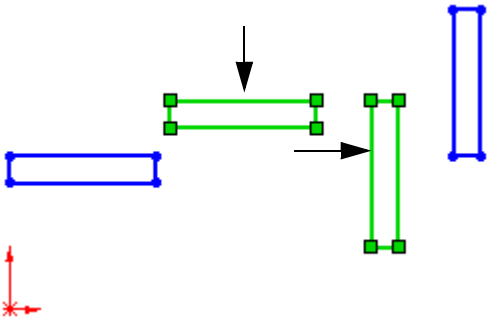
Move Sketch Entities

To move sketch entities:

1 Open **Sketches\move_entities.sldprt**.

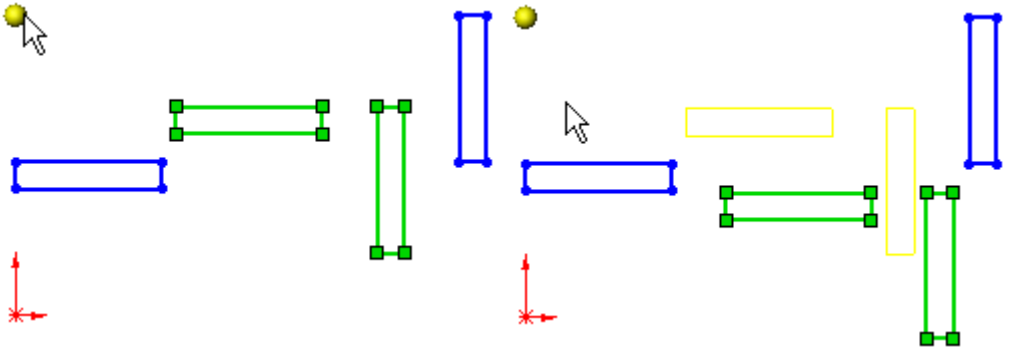
2 Click **Move Entities** (Sketch toolbar) or **Tools**, **Sketch Tools**, **Move**.

3 Block select the two rectangles as shown, for **Entities to Move** .



4 Under **Parameters**, click **Base Point** .

5 Click in the graphics area as shown, to place the **Base Point** .



Base point from which to move

New position after drag

6 Drag right to position, double-click , and close the sketch.

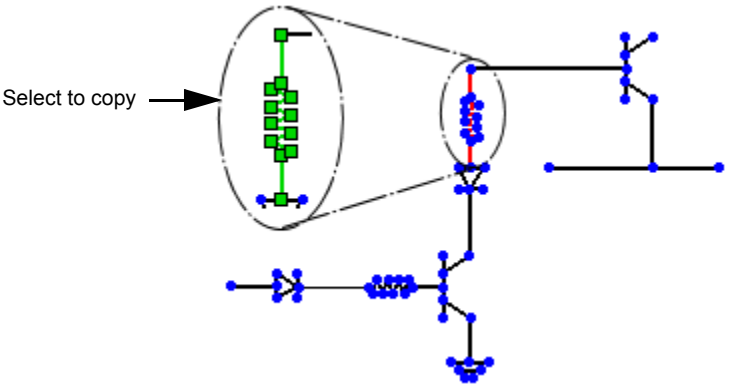
Copy Sketch Entities

To copy sketch entities:

1 Open **Sketches\copy_resistor.sldprt**.

2 Click **Copy Entities** (Sketch toolbar) or **Tools**, **Sketch Tools**, **Copy**.

3 Select the resistor in red for **Entities to Copy** .



4 In the PropertyManager:

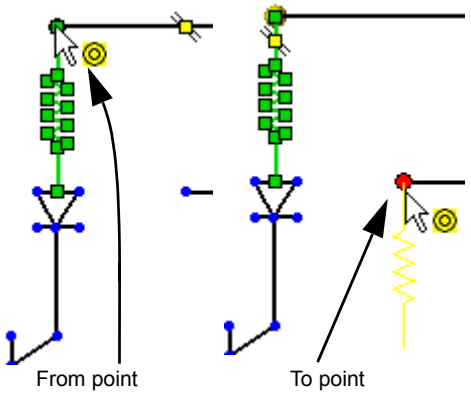
a) Under **Entities to Copy**, select **Keep relations**.

b) Under **Parameters**, click **Base Point** .

5 Select the vertex at the top of the resistor as the **From** point.

6 Select the left vertex on the horizontal line as the **To** point.

7 Click to place the copy, and then click .



Circular and Linear Sketch Patterns

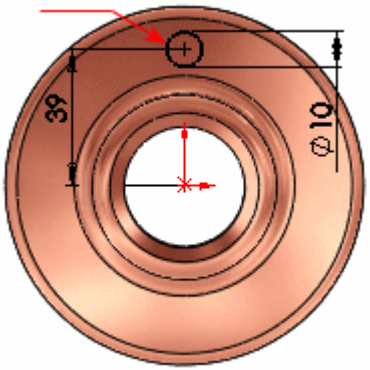
Circular and linear sketch patterns use a PropertyManager.

To add a circular sketch pattern:

1 Open **Sketches\circular_pattern.sldprt**.

2 Click **Circular Sketch Pattern** (Sketch toolbar) or **Tools**, **Sketch Tools**, **Circular Pattern**.

3 Select the circle for **Entities to Pattern**.

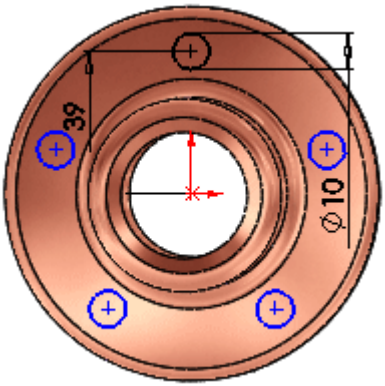


Point-1 is added under **Parameters** as the center of the pattern .

Circular sketch patterns default to the sketch origin as the center for the pattern, but you can select any point as the center.

4 In the PropertyManager under **Parameters**, set **Number** to **5**.

5 Click .



To edit the sketch pattern, right-click a pattern instance and select **Edit Circular Pattern**.

Preserve Background Transparency for Raster Files

You can display and control the transparency of images added to 2D sketches (**Sketch Picture** command, Sketch toolbar). **Transparency** settings include:

- **None**. Use no transparency attributes.
- **From file**. Keep the transparency attributes that exist in the file.
- **Full Image**. Select to set the whole image transparent.
- **User defined**. Select a color from the image, define a tolerance level for that color, and apply a transparency level to the image.

To define sketch transparency:

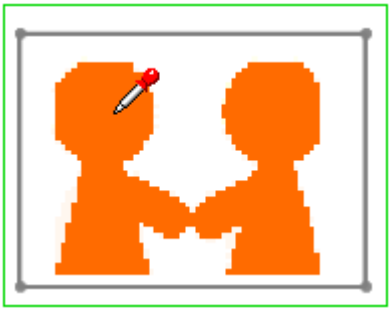
1 Open **Sketches\transparency.sldprt**.

The part has two sketches on parallel planes.

2 Under **Sketch1**, double-click **Sketch Picture1**.

3 In the PropertyManager, under **Transparency**, select **User defined**.

4 With the eye dropper, click in the orange area, as shown, to define the color.



5 In the PropertyManager, under **Transparency**, move the **Matching tolerance** slider to approximately **.80**.

Matching tolerance determines how closely you want to match variations of the color you select. Each color in most images includes varying shades. Use **Matching tolerance** to control the degree to which the selected color is displayed.

6 Click .

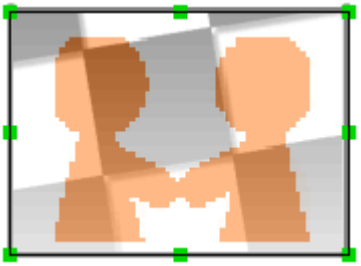
7 Press the left arrow key to rotate the sketch 360°, so that the **Sketch1** is in the background, and **Sketch2** is in the foreground.

8 In the FeatureManager design tree, under **Sketch2**, double-click **Sketch Picture2**.

9 In the PropertyManager, under **Transparency**, select **Full image**.

10 In the PropertyManager move the **Transparency** slider to approximately **.45**.

Sketch1 is displayed through **Sketch2**.



3

Features

This chapter describes enhancements to features in the following areas:

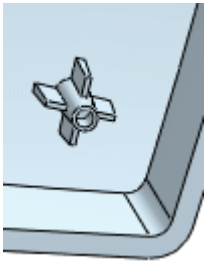
- Fastening features
- Hole Series
- Hole Wizard
- Lofts
- Patterns

Fastening Features

The new Fastening Features streamline creation of common features for plastic parts. You can create:

- Mounting bosses
- Snap hooks
- Snap grooves
- Vents

To insert mounting bosses:



1 Open **Features/Fastening_Features.sldprt**.

2 Click **Mounting Boss** (Fastening Features toolbar) or **Insert**, **Fastening Feature**, **Mounting Boss**.

3 In the PropertyManager:

a) Under **Position**, select the face in the lower-right corner as shown for **Select a face** to position the mounting boss.

b) Select **Axis1** for **Select Direction** to align the mounting boss with **Axis1**. If necessary, click **Reverse Direction** so the mounting boss points towards you.

You can control the parameters for the boss, fins, and mounting hole or pin.

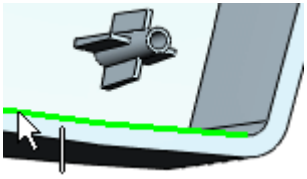
c) Under **Fins**, select **Axis2** as a vector to define the fin orientation.

d) Click .

To insert snap hooks:

1 Click **Snap Hook** (Fastening Features toolbar) or **Insert**, **Fastening Feature**, **Snap Hook**.

2 In the PropertyManager, under **Snap Hook Selections**:



a) Select the edge shown in the lower-right corner for to position the snap hook. The selection point defines the center of the leading edge of the hook's base.

b) Select **Axis1** for to define the vertical direction of the hook.

c) Select **Reverse direction** under **Axis1** to display the snap hook.

d) Select **Axis2** for to define the direction of the hook.

e) Select **Enter body height**, then set the base height to **15**.

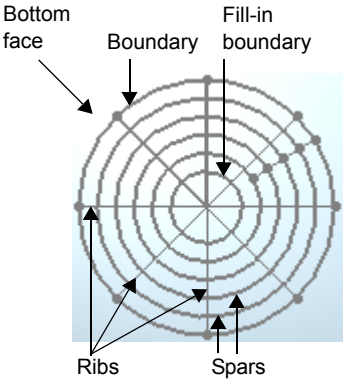
f) Click .

To add fastening features to each corner:

1 Mirror **Mounting Boss1** and **Snap Hook1** about the **Right Plane** to create **Mirror1**.

2 Mirror **Mirror1** about the **Front Plane** to create all the mounting bosses and snap hooks.

To add a vent.:



1 Click **View**, **Sketches** to show **Sketch Vent**.

2 Click **Bottom** (Standard Views toolbar).

3 Click **Vent** (Fastening Features toolbar) or **Insert**, **Fastening Feature**, **Vent**.

4 In the PropertyManager:

a) Select the outer circle shown for **Boundary** .

b) Under **Geometry Properties**, select the bottom face for **Face** .

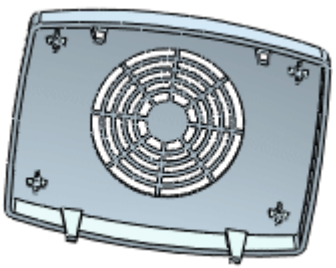
c) Select the eight lines for **Ribs**. Set the width to **2**.

d) Select the four outer circles for **Spars**.

e) Select the inner circle for **Fill-In Boundary** .

f) Click .

5 Hide the sketches, then rotate the model to view the fastening features.



Hole Series

You can now create a hole series, commonly used for mounting holes, using the **Hole Series** PropertyManager.

Hole Wizard

The Hole Wizard functionality now uses a PropertyManager. Feature Scope is now available for Hole Wizard holes.

To insert a Hole Wizard hole:

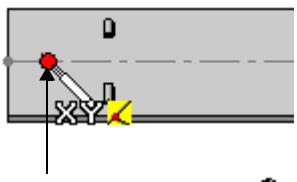
1 Open **Features**/**connector**/**PC_connector.sldprt**.

2 Click **Hole Wizard** (Features toolbar) or **Insert**, **Features**, **Hole**, **Wizard**.

> You can also drag a favorite hole from the **Design Library** onto the part.

3 On the **Type** tab, under **Hole Specification**:

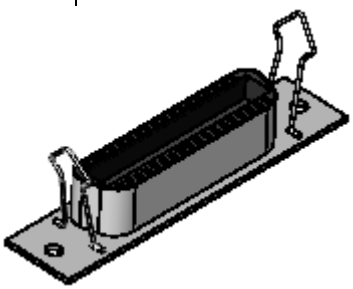
- Click **Hole** .
- Select **Ansi Metric** in **Standard**.
- Select **M3.0** in **Size**.



4 On the **Positions** tab, select the sketch point as shown.

5 Select the sketch point on the opposite end of the part to place a second hole.

> Both holes must be the same type. To create different types of Hole Wizard holes, add them separately.



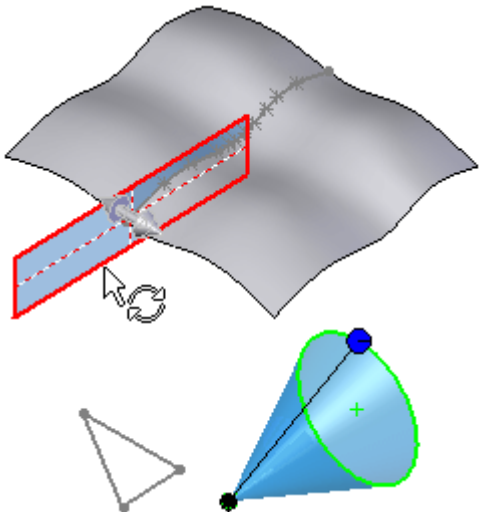
6 Click , then save the part.

7 Open **Features**/**connector**/**HoleWiz.sldasm** to see the entire connector assembly.

Lofts

General enhancements:

- ❑ You can add loft sections with greater accuracy and fewer control points.
- ❑ Loft sections added using the **Add Loft Section** command automatically create pierce points at their end points. Previously these sections were added with no relation to existing guide curves.
- ❑ You can create a loft to a point, even if the point is part of a sketch containing other sketch entities. Previously the point had to be the only entity in the sketch.



Contour Selection

You can enable contour selection through the new **Sketch Tools** box in the PropertyManager. You can select single contours, automatically chain multiple contours, or select multiple contours and join them to form a single sketch.

Dynamic Sketch Update

When you create or edit a loft, you can drag 3D sketch geometry to change the feature's shape. You can also edit the sketch dimensions.

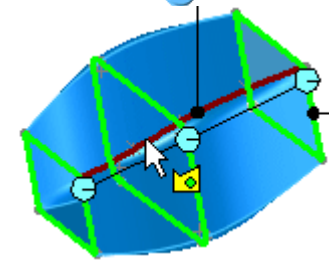
The feature can have only one 3D sketch. You cannot dynamically edit 2D sketches.

To dynamically update 3D sketches while editing a feature:

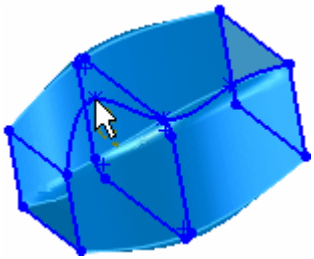
1. Edit the feature.
2. In the PropertyManager, under **Sketch Tools**, click **Drag Sketch** to enter 3D sketch drag mode.

3 Drag a guide curve or profile to reshape the feature.

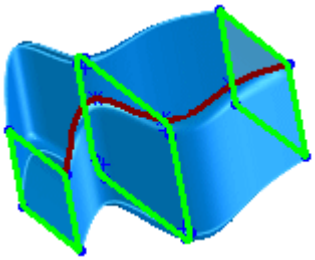
4 Click .



Select a 3D guide curve contour



Drag the contour



Preview of new loft

Single 3D Sketch

All sketch entities (including guide curves and profiles) can be contained in a single 3D sketch.

Start/End Constraint

Loft incorporates a new default start/end constraint called **Default**. The **Default** tangency is approximated by a parabola that is scribed between the first and last profile. Tangency is derived from the parabola to drive the loft surface.

Incorporating this new default condition results in a more predictable and natural loft surface when matching conditions are not specified.

Previously, the **None** constraint applied zero curvature to the start/end constraints of the loft.

Patterns

Curve Driven Patterns

Curve driven patterns now support 3D geometry. You can pattern features along 3D curves or 3D edges.

Data Propagation

A new option propagates SolidWorks colors, textures, and cosmetic thread data to all pattern instances.

To propagate data to pattern instances:

1 Open **Features**/ **Pattern_Data.sldprt**.

The boss has a cosmetic thread and the cut extrude is blue.

2 Click **Linear Pattern** (Features toolbar) or **Insert**, **Pattern/Mirror**, **Linear Pattern**.

3 In the PropertyManager, under **Direction 1**:

a) Select the edge shown for **Pattern Direction**. Make sure the direction arrow points left as shown.

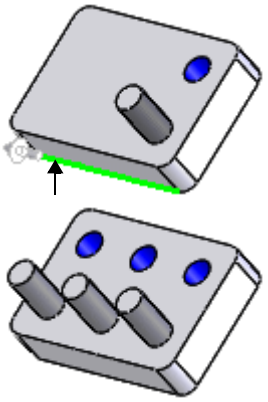
b) Set **Spacing** to **50**.

c) Set the **Number of Instances** to **3**.

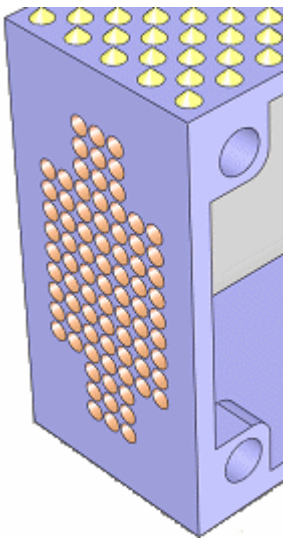
4 Under **Features to Pattern**, select **Extrude2** and **Cut-Extrude1**.

5 Under **Options**, select the new option **Propagate Visual Properties**.

6 Click .



Fill Patterns



The new Fill Pattern feature lets you select an area defined by co-planar faces or a sketch that lies on co-planar faces. The command fills the defined region with a pattern of features or a predefined cut shape.

You set parameters to control the way the grid of instances is laid out. You can create a grid for a sheet metal perforation-style pattern, or a grid of concentric shapes which you would typically use on aesthetic parts.

The predefined cut shapes available are circles, squares, diamonds, and polygons. You can control the parameters for each shape.

Typical uses include:

- Weight reduction
- Ventilation holes
- Grip Surfaces

Multibody Parts

You can create the patterns below in multibody parts:

- Sketch driven and curve driven patterns. Select **Geometry pattern** in the PropertyManager. If **Geometry Pattern** is cleared, the pattern appears only on the body on which the seed feature is located (for the sketch pattern).
- Linear patterns. Select or clear **Geometry pattern** in the PropertyManager.

Feature Scope is now available for sketch and curve driven patterns.

To create patterns in multibody parts:

1 Open **Features**/**Multibody_Patterns.sldprt**.

2 Click **Linear Pattern** (Features toolbar) or **Insert**, **Pattern/Mirror**, **Linear Pattern**.

3 In the PropertyManager, under **Direction 1**:

a) Select the edge shown for **Pattern Direction**.

b) Set **Spacing** to **30**.

c) Set **Number of Instances** to **2**.

4 Under **Direction 2**:

a) Select the same edge as **Direction 1**. If necessary, click **Reverse Direction** so the arrow points in the opposite direction from **Direction 1**.

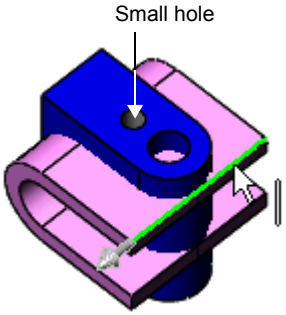
b) Set **Spacing** to **30** and **Number of Instances** to **2**.

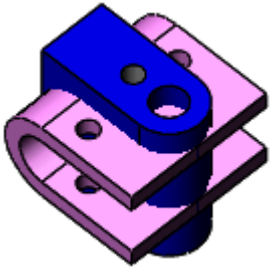
5 Under **Features to Pattern**, select the small hole in the graphics area.

6 Under **Options**, clear **Geometry pattern**.

7 Click .

The linear pattern appears across the multiple bodies. Previously this pattern failed.





4

Parts

This chapter describes enhancements to parts in the following areas:

- Layout of sketch blocks
- Mates in multibody parts
- Smart Components

Layout of Sketch Blocks

You can create a layout of sketch blocks in a part document (see **Blocks** on page 2-3). Using this method, you can conceptually design an assembly, then automatically create a full assembly, generating one component for each block (see **Layout of Sketch Blocks** on page 5-6).

Mates in Multibody Parts

In multibody parts, you can precisely place bodies using mates. The following mates are supported:

Angle
Coincident
Concentric
Distance
Parallel
Perpendicular
Tangent

You can apply mates:

- While inserting a body into a part
- After the body is already in the part

Additional capabilities:

- Apply multiple sets of mates to the same body. Mates specified within different sets can conflict with each other. For example, you can apply a perpendicular mate between two faces in one set, and in a different set, apply a parallel mate between the same two faces.
- Select several bodies at once to be positioned by mates. The selected bodies move as a single entity. The bodies that are not selected are treated as fixed.

Multibody parts should not replace the use of assemblies. A multibody part consists of multiple solid bodies that are not dynamic. If you need to represent dynamic motion among bodies, use an assembly. Tools such as **Move Component**, **Dynamic Clearance** and **Collision Detection** are available only with assembly documents.

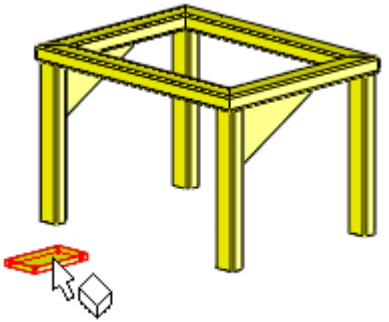
To add mates between bodies:

1 Open **Parts\weldment.sldprt**.

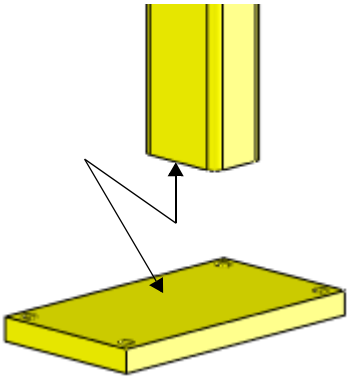
2 Click **Move/Copy Bodies** (Features toolbar) or **Insert**, **Features**, **Move/Copy**.

In the PropertyManager, click **Constraints** if you do not see the **Mate Settings** group box.

3 In the PropertyManager, for **Bodies to Move**, select the foot in the graphics area.



4 Under **Mate Settings**, select the top face of the foot and the bottom face of the weldment for **Entities to Mate**.

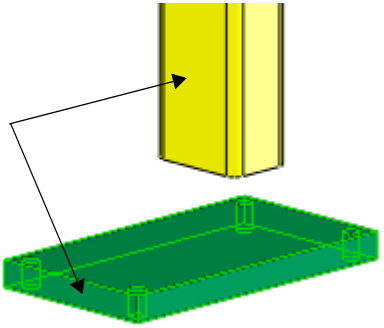


The two faces align, and **Coincident** is selected in the PropertyManager.

5 Click **Add**.

Coincident appears under **Mates**.

6 Select the sides of the foot and weldment as shown for **Entities to Mate**.

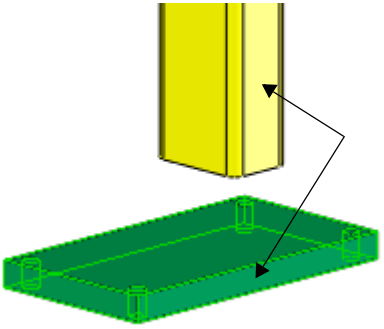


7 Click **Distance** and set **Distance** to **70**. Select **Flip Dimension** if necessary to center the foot on the weldment leg.

8 Click **Add**.

Distance appears under **Mates**.

9 Select the sides of the foot and weldment as shown for **Entities to Mate**.



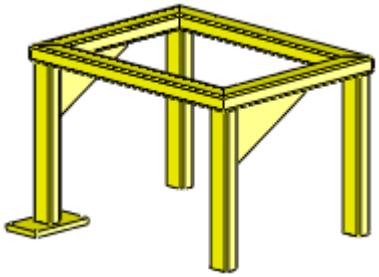
10 Click **Distance** and set **Distance** to **20**. Select **Flip Dimension** if necessary to center the foot on the weldment leg.

11 Click **Add**.

Distance appears under **Mates**.

12 Click .

The mates appear under **Body-Move/Copy** in the FeatureManager design tree.



To add mates when inserting a body, select **Launch Move Dialog** in the **Insert Part** PropertyManager.

Smart Components

You can create smart components to insert into assemblies. When you make a component smart, you associate other components and features with the component, such as bolts and mounting holes. When you insert the component into an assembly, you can choose to insert the associated components and features. The following feature types can be associated with a smart component:

- Extruded bosses and cuts
- Revolved bosses and cuts
- Simple holes
- Hole Wizard holes

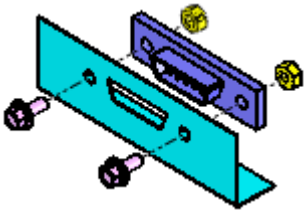
You can map configurations of the smart component to configurations of the associated components and features.

Creating Smart Components

You define a smart component in a temporary defining assembly. After you establish the component as smart, the defining assembly is no longer needed. All the defining data, including the information necessary to reconstruct the defining assembly, is captured in the component document.

To create a smart component:

1 Open **Parts\defining_assembly.sldasm**.



Use this assembly to define **d_connector.sldprt** as a smart component, with the screws, nuts, and holes as associated components and features. The connector has 3 configurations (11-pin, 17-pin and 21-pin). The sheet metal part has three corresponding configurations with different cutout sizes. The screws and nuts are the same for each configuration.

2 Click **Make Smart Component** (Assemblies toolbar) or **Tools**, **Make Smart Component**.

3 In the PropertyManager:

a) Under **Smart Component**, select **d_connector<1>** for **Component to make smart** .

b) Under **Components**, select the two screws and the two hex nuts for **Components associated with Smart Component** .

c) Under **Features**, select **D Cutout and holes** of **smetal_part<1>** for **Features to make smart** .

The components are hidden to facilitate selection of features in the graphics area. After selecting features, click **Show Components**.

4 Click **Configurator Table**.

The Configurator Table lists the configurations of **d_connector**, and has a column for every component in the defining assembly. You specify the configurations for every associated component and feature for each configuration of the smart component.

You can enter information in the table by:

- Typing
- Selecting from drop-down menus
- Copying and pasting from other cells

5 Click in the first cell under **D Cutout and holes <smetal_part-1>**, and select **11pin**.

6 For the second and third cells under **smetal_part-1**, select **17pin** and **21pin**.

7 In each of the cells for the screws and the hex nuts, select **6mm diameter**.

Every cell now has a value.

8 Click **OK**.

9 Click .

In the FeatureManager design tree, the icon for **d_connector<1>** changes to , to indicate that it is a smart component.

10 Save the assembly and click **Yes** to save the referenced models that have been modified.

All the smart component's defining data, including the information necessary to reconstruct the defining assembly, is saved in the component document **d_connector.sldprt**.

11 Close the assembly.

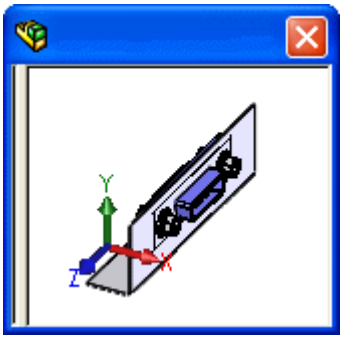
The defining assembly is no longer needed and can be deleted.

To examine the smart component definition:

1 Open **Parts\d_connector.sldprt**.

The new folder, **Smart Feature**, in the FeatureManager design tree, indicates that the part is a smart component. The subfolders **Features**, **Components**, and **References** contain information about the associated components and features.

2 Right-click **Smart Feature** and select **Preview**.



A preview window appears, showing a temporary assembly of the component and its associated components and features.

3 Expand the **Features** folder under **Smart Feature**, and select **D cutout and holes**.

The preview shows the **D cutout and holes** feature.

4 Expand **Components** and **References**, and select items in these folders.

The preview shows the selected component or reference face.

5 Click in the graphics area to close the preview window.

6 Close the part.

Inserting Smart Components into Assemblies

You insert a smart component into an assembly and position it with mates, exactly as you do for any other component. Then you activate the Smart Feature, and choose which of the smart component's associated components and features to add.

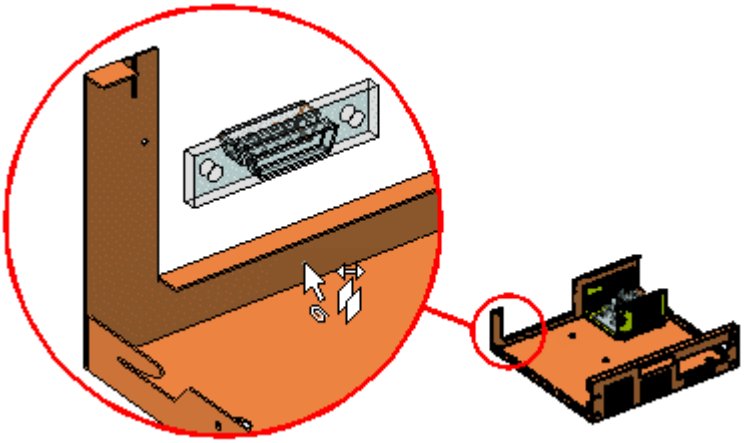
To add a smart component to an assembly:

1 Open **Parts\computer.sldasm**.

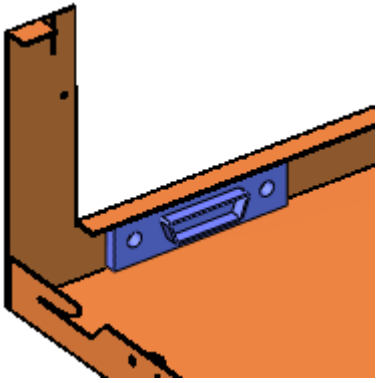
2 Click **Insert Components** (Assemblies toolbar) or **Insert**, **Component**, **Existing Part/Assembly**.

3 Browse to **Parts\d_connector.sldprt** and click **Open**.

4 Click to place the part on the inside face of the chassis, as shown, and click **Add/Finish Mate** (Mate pop-up toolbar) to accept the coincident mate.



5 Move the connector if necessary to position it over the face.

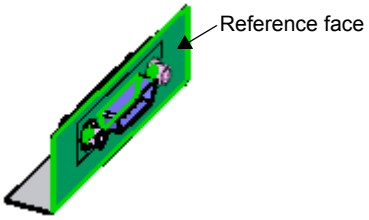


6 Select the connector and click **Insert**, **Smart Features**.

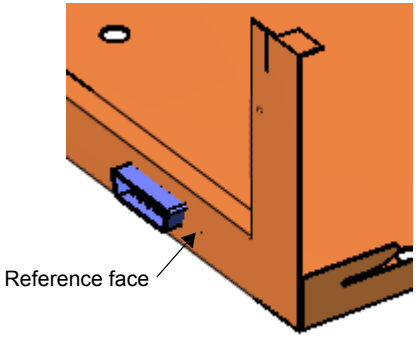
Other methods for inserting smart features:

- In the graphics area, right-click the connector and select **Insert Smart Features**.
- In the FeatureManager design tree, select **d_connector<1>**, then click the icon that appears over the connector in the graphics area.

A preview window appears that highlights the reference face required to insert the cutout and screws.



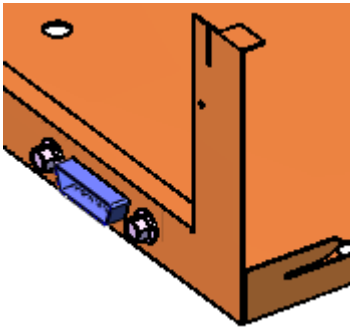
7 Select the *outside* face of the chassis.



8 In the PropertyManager, under **Options**, select **Update feature and component size/location when smart component moves/changes** so that if you change the smart component, the associated components and features also change.

9 Click .

The screws, nuts, and cutout and holes are added to the assembly.



If you move the connector, the associated components and features move when you click **Rebuild** (Standard toolbar).

Editing Smart Components

The defining data for a smart component, including the information necessary to reconstruct the defining assembly, is stored in the smart component document.

To edit the definition of a smart component:

1 Open **Parts\d_connector.sldprt**.

2 In the FeatureManager design tree, right-click on **Smart Feature** and select **Edit in Defining Assembly**.

A temporary assembly, **d_connector_ta.sldasm**, opens.

3 Click **Edit Definition** on the pop-up toolbar.

4 In the PropertyManager, click **Configurator Table**.

5 For **d_connector** configuration **21pin**, change the screw and hex nut configurations to **8mm diameter**.

6 Click **OK**.

7 Click .

8 On the ConfigurationManager tab, double-click **21pin** to see the updated configuration.

9 Save and close the assembly.

The temporary assembly disappears. The updated information is stored in the smart component **d_connector.sldprt**.

Auto Size

You can add auto-sizing capability to cylindrical smart components. When you insert the smart component onto a cylindrical component in an assembly, the size of the smart component is set to fit the cylindrical component.

To add auto-sizing to a smart component:

1 Open **Parts\ring_defining_assembly.sldasm**.

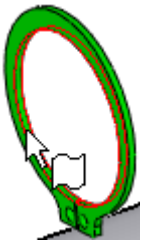
2 Click **Make Smart Component** (Assemblies toolbar) or **Tools**, **Make Smart Component**.

3 In the PropertyManager:

a) Under **Smart Component**, select **retaining_ring<1>** for **Component to make smart** .

b) Under **Features**, select **Retaining ring groove** of **shaft<1>** for **Features to make smart** .

4 Under **Auto Size**:



a) Select **Diameter**.

b) Under **Concentric mate reference**, select the inside cylindrical face of the retaining ring for **Cylindrical Face or Axis** . This defines the diameter that will automatically resize.

5 Click **Configurator Table**.

6 For each configuration of **retaining_ring**, map the appropriate **shaft-1** retaining ring groove and define the minimum and maximum shaft diameters allowed, as follows:

retaining_ring	shaft-1	Minimum Diameter	Maximum Diameter
40mm shaft	Ø40	39	40
65mm shaft	Ø65	64	65

7 Click **OK**.

8 Click .

In the FeatureManager design tree, the icon for **retaining_ring<1>** changes to .

9 Save and close the assembly.

To add an auto-sizing smart component to an assembly:

1 Open **Parts\shaft_assembly.sldasm**.

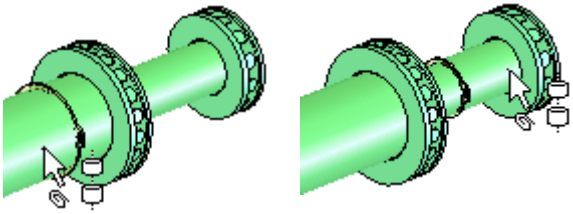
2 Click **Insert Components** (Assemblies toolbar), then:

a) Browse to **retaining_ring.sldprt** and click **Open**.

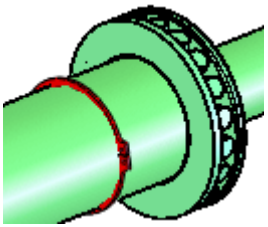
b) Under **Options**, select **Graphics preview**.

3 Move the pointer over the large shaft, then over the small shaft.

The preview of the retaining ring changes size, depending on the size of the shaft.



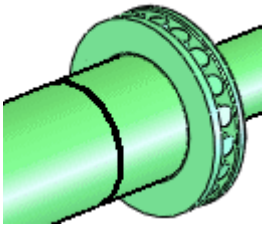
4 Click to place the ring on the large shaft, and click **Add/Finish Mate** (Mate pop-up toolbar) to accept the concentric mate.



In the FeatureManager design tree, the icon indicates that **retaining_ring<1>** is a smart component.

5 Right-click the retaining ring and select **Insert Smart Features**.

The groove is added to the shaft. You can hide the retaining ring to see the groove.



In the FeatureManager design tree:

- **Smart-Feature1<retaining_ring-1>** appears under **shaft2<1>**.
- **Retaining_ring-1** appears and contains the smart component **retaining_ring<1>** and its associated feature.

If you move the ring, the groove moves when you click **Rebuild** (Standard toolbar).

5

Assemblies

This chapter describes enhancements to assemblies in the following areas:

- Assembly performance
- Display states
- Feature scope
- Layout of sketch blocks
- Physical simulation
- Load references
- Width mates
- General enhancements

Assembly Performance

Large Assembly Mode

In **System Options**, the **Large Assembly Mode** page has been removed and its options moved to the **Assemblies** page. Many seldom-used options are now automatically set by the system.

When you open an assembly whose number of components exceeds the threshold you specified in **Options**, you can select or clear **Large Assembly Mode** in the **Open** dialog box.

To change Large Assembly Mode options:

1 Click **Options** (Standard toolbar), or click **Tools**, **Options**.

2 Select **Assemblies**.

3 Set options under **Large Assemblies**.

4 Click **OK**.

Lightweight Components

Enhancements:

- Components need to be resolved less frequently.
- You can create section views of assemblies without resolving lightweight components.

You need to convert documents to SolidWorks 2006 format to enable lightweight mode for components.

Display States

You can set a separate display mode (Wireframe, Hidden Lines Removed, etc.) for each component in an assembly. You can define different combinations of the following settings for each component, and save them in Display States:

- Display mode
- Hide/Show state
- Component color
- Component texture
- Transparency

You can:

- Create multiple display states for each configuration of the model
- Copy display states from one configuration to another

You can toggle between display states without having to change configurations, which can result in improved performance, especially in large assemblies.

The Display Pane helps you visualize the display settings for each component in a display state. You can click in the Display Pane to change display settings for each component.

In design tables, the new keyword **$DISPLAYSTATE** controls the display state of a configuration. The keyword **$SHOW** is obsolete, because the hide/show state of components is now stored in Display States.

In assembly drawings, you can create separate display states for each view.

To create a new display state:

1 Open **Assemblies\cordless_drill.sldasm**.

2 Click **Shaded With Edges** (View toolbar) or **View**, **Display**, **Shaded With Edges**.

3 On the ConfigurationManager tab, expand **Default** and **Display State**.

Display State-1 is the default display state for this configuration.

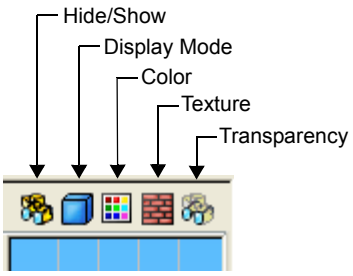
4 Right-click **Display State** and select **Add Display State**.

Display State-2 is added to the list and becomes the active display state.

5 Rename **Display State-2** as **demo**.

6 On the FeatureManager tab, click (to the right of the tabs) to show the Display Pane.

The Display Pane has five columns.

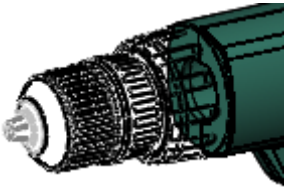


7 Move the pointer over **Transmission** in the FeatureManager design tree, then:

a) Move the pointer to the right until it is in the Display Mode column and the pointer changes to .

b) Click, and select **Hidden Lines Removed**.

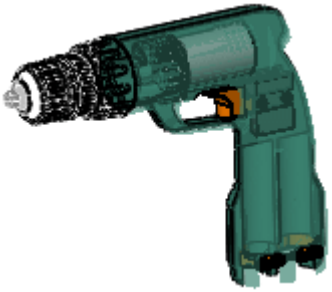
The component display mode changes, and replaces (default display mode) in the Display Pane.



Components set to the default display mode use the display mode of the top level assembly in the current window, as set on the View menu or View toolbar.

8 For the **BODY-RIGHT** component, click in the Transparency column, and select **Change transparency**.

The component becomes transparent, and appears in the Display Pane for **BODY-RIGHT**.

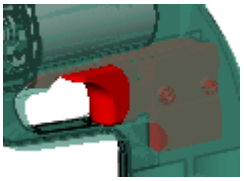


9 For **Trigger**:

a) Click in the Display Mode column and select **Shaded**.

b) Click in the Color column and select **Color**.

c) Select red and click .

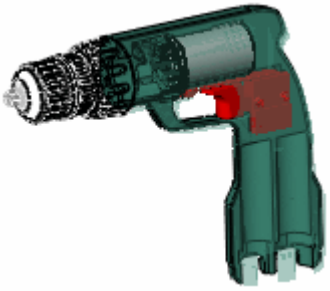


10 For **Retainer**, click in the Hide/Show column and select **Hide**.

11 On the ConfigurationManager tab, double-click **Display State-1** .

Display State-1 becomes the active display state. You can toggle back and forth by double-clicking each display state.

Display State-1



Display State-2

Feature Scope

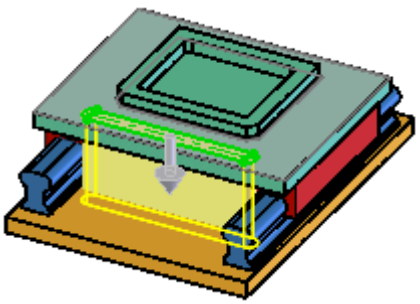
While defining assembly features (extruded and revolved cuts, simple holes, and Hole Wizard holes), you can specify (in the PropertyManager) the components that the feature affects:

- **All components**. All components in the assembly, including those added after the feature is created. When the feature changes, rebuilds act on all components in the assembly, which can result in slower performance.
- **Selected components**. Only those components listed in the selection box. You can select components manually, or by specifying **Auto-select**, which selects all components intersected by the feature. When the feature changes, rebuilds act only on the selected list.

To define the scope of an assembly feature:

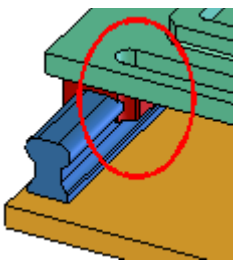
1. Open **Assemblies\lens_mount.sldasm**.
2. Click **Extruded Cut** (Features toolbar) or **Insert**, **Assembly Feature**, **Cut**, **Extrude**.
3. Select **Sketch1** at the bottom of the FeatureManager design tree.
4. In the PropertyManager, under **Direction1**, select **Through All** in **End Condition**.

   The preview shows the cut affecting all components.



5. Under **Feature Scope**, clear **Auto-select**.
6. Select **slider<1>**, **slider<2>**, and **baseplate<1>** for **Components to Affect**.
7. Click .

   The cut affects the baseplate and two sliders, but not the rails or table.



Layout of Sketch Blocks

You can create an assembly from a layout of sketch blocks (see **Blocks** on page 2-3). Using this method, you can conceptually design an assembly, then automatically create a full assembly, generating one component for each block. The generated components are:

- Positioned the same as their representative block
- Mated to the layout, so changes to the layout change the components

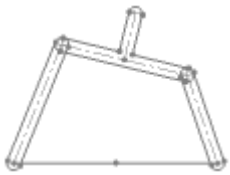
If a sketch block is from an existing component, that component is used in the assembly. Otherwise, a new component is created that consists of a single sketch identical to the sketch block.

You can create the new components in the same assembly as the layout, or create a new assembly, preserving the original layout.

To create an assembly from a layout:

1 Open **Assemblies\layout01.sldasm**.

The assembly contains a sketch, but no components.



2 In the FeatureManager design tree, expand **Sketch1**.

Sketch1 contains four blocks. The first block has its default name (**Block1-1**). The others have been renamed to **left_link**, **t_link**, and **right_link**.

3 Right-click **Sketch1** in the FeatureManager design tree, and select **Create Assembly from Layout Sketch**.

The dialog box contains the following columns:

- **Create**. Creates a component for the corresponding block listed under **Source**.
- **Source**. Lists the layout sketch and its blocks.
- **Target Component**. Lists the names of the assembly and part documents to be created or modified.
- **Path**. Lists the location of the assembly and part documents to be created or modified.

4 In the dialog box:

a) Select **Create new top level assembly**. The **Target Component** for **Sketch1** changes from **layout01** to a new assembly name.

b) In the **Create** column, clear the first check box, so a component will *not* be created for **Block1**. The **Target Component** and **Path** for **Block1** disappear.

c) Under **Target Component**, click in the first box and rename the new assembly to **linkage01**.

5 Click **OK**.

A new assembly (**linkage01**) opens. It contains **Sketch1**, plus 3 new components. Expand **Mates** to see three **LockToSketch** mates, which mate each component to **Sketch1**.

You can now edit the new component documents to further define the components of the new assembly.

6 Right-click **t_link<2>** in the FeatureManager design tree and select **Open Part**.

7 Click **Extruded Boss/Base** (Features toolbar) or **Insert**, **Boss/Base**, **Extrude**.

8 Select **Sketch1** in the FeatureManager design tree.

9 In the PropertyManager, under **Direction 1**, set **Depth** to **10**.

10 Click .

11 Save and close the part.

12 In **linkage01.sldasm**, change the view to **Isometric** .

Physical Simulation

You can assign numerical values for:

- Linear acceleration of a linear motor feature
- Acceleration of gravity of a gravity feature
- Angular velocity of a rotary motor feature

The values are used by the COSMOSMotion application to define the loads on the components when you perform a motion analysis based on the physical simulation.

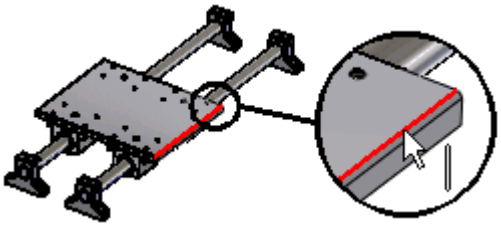
To assign a numerical value:

1 Open **Assemblies\linear_slide.sldasm**.

2 Click **Linear Motor** (Simulation toolbar) or **Insert**, **Simulation**, **Linear Motor**.

3 In the PropertyManager:

a) Select the upper right edge of the plate in the graphics area for **Direction Reference**.



b) Select **Numeric**.

c) Set **Numeric motor value** to **100**.

d) Click ✔.

4 Click **Calculate Simulation** (Simulation toolbar).

5 Click **Replay Simulation** (Simulation toolbar) to see the simulation.

Values outside the default range for physical simulation affect only the COSMOSMotion application. For example, the default maximum speed of a linear motor is 300 mm/s. If you specify a value greater than 300 mm/s, the component does not move at a correspondingly faster rate when you replay the physical simulation, but does move faster in the COSMOSMotion application.

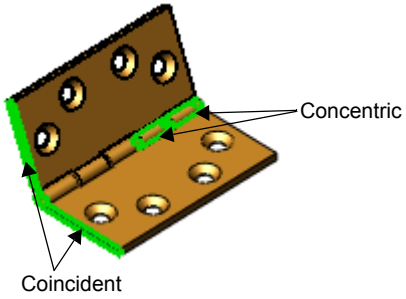
Load References

You can associate additional faces with a mate, that can then be used to define load-bearing surfaces in COSMOSMotion analysis.

To insert load references:

1 Open **Assemblies\hinge.sldasm**.

The hinge assembly has a coincident mate between two planar surfaces and a concentric mate between two cylindrical surfaces as shown.



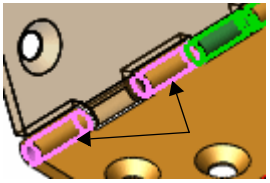
2 In the FeatureManager design tree:

a) Expand **Mates** .

b) Right-click **Concentric1** and select **Insert Load Reference**.

3 In the PropertyManager, under **Component1(hinge<1>)**, select **Display component2 transparent**.

4 In the graphics area, select the two inner cylindrical surfaces of **hinge<1>** that are not yet part of the mate.



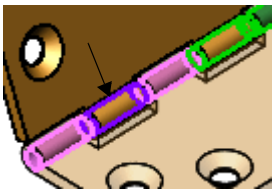
5 Clear **Display component2 transparent**.

6 Under **Component2(hinge<2>)**:

a) Click in the selection box.

b) Select **Display component1 transparent**.

7 In the graphics area, select the inner cylindrical surface of **hinge<2>** that is not yet part of the mate.



8 Click .

9 In the FeatureManager design tree, under **Mates** , expand **Concentric1** and select **Concentric1 Load Reference**.

The faces you defined in steps 4 and 7 highlight in the graphics area, and are now associated with the **Concentric1** mate.

Width mates

A width mate centers a tab within the width of a groove.

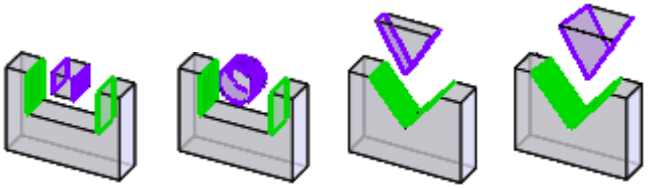
Tab references can include:

- Two parallel planar faces
- Two non-parallel planar faces (with or without draft)
- One cylindrical face or axis

Groove width references can include:

- Two parallel planar faces
- Two non-parallel planar faces (with or without draft)

Examples:



To add a width mate:

1 Open **Assemblies\trigger.sldasm**.

2 Click **Mate** (Assembly toolbar) or **Insert**, **Mate**.

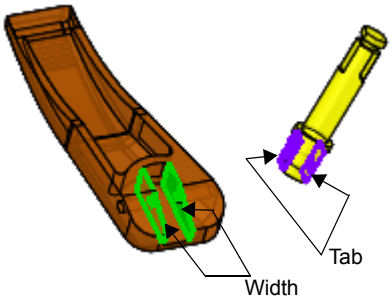
3 Under **Advanced Mates**, click **Width**.

4 Under **Mate Selections**:

a) For **Width Selections**, select the two walls of the groove in the base.

b) For **Tab Selections**, select the two faces on the arm.

The components align so that the tab is centered between the faces of the groove.



5 Click twice to add the mate and close the PropertyManager.

6 Change view to **Front**.

7 Move and rotate the arm.

The arm can translate along the center plane of the groove and rotate about an axis normal to the center plane. The width mate prevents the arm from translating or rotating side to side.

If **Enable selection through transparency** is selected in **Tools**, **Options**, **System Options**, **Display/Selection**, press **Shift** when you select the arm to temporarily override the option.

General Enhancements

Assembly Statistics

In multi-configuration drawings, **Assembly Statistics** (Tools toolbar) now works for all drawing views, including section views, even if the view is not of the current configuration.

Smart Components

You can insert smart components into assemblies. When you create a smart component, you define hardware and features associated with the component, such as bolts and mounting holes. When you insert the component into an assembly, the associated hardware and features are also inserted. See **Smart Components** on page 4-5.

Smart Fasteners

Smart Fasteners functionality has been improved, especially inserting fasteners in a pattern of holes.

6

Drawings

This chapter describes enhancements to detailing in the following areas:

- Drawing view quality
- Model views
- Detail views
- Broken-out section views
- Broken views
- Section views
- 3D drawing view
- Edge display

Drawing View Quality

There are no longer high quality and draft quality view settings in drawing view PropertyManagers. All drawing views are automatically high quality. As a result, lightweight drawings use the high quality view style. To override the high quality setting, click **Tools**, **Options**, **System Options**, **Display Style**, and set **Display quality for new views** to **Draft quality**.

If you open a drawing saved as draft quality in a previous release, it remains in this state in SolidWorks 2006. However, if you change the drawing view to high quality, you cannot change it back to draft quality.

The exception to draft quality views is when the software detects very large assembly drawings. In this case, you are asked if you want to create draft quality views.

Model Views

When you insert a model view, the PropertyManager allows you to:

- Insert single or multiple views of the model.
- Select standard view orientations or annotation views. See **3D Annotations** on page 7-3.

To insert multiple model views with standard view orientations:

1 Open a new drawing.

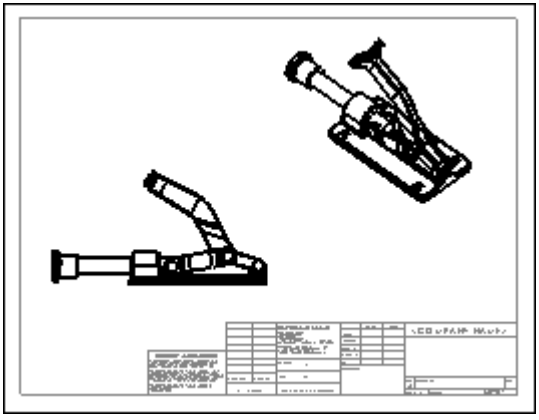
2 In the PropertyManager:

a) Click **Browse** and open **drawings\drill_fixture\drill_fixture.sldasm**.

b) Under **Number of Views**, select **Multiple views**.

c) Under **Orientation**, click ***Front** and ***Isometric** .

d) Click .



Detail Views

Bend lines in flat pattern sheet metal drawings remain visible in detail views.

Broken-Out Section Views

When you create a broken-out section view, you can exclude components and fasteners.

To exclude components from a broken-out section view:

1 Open **drawings\chain\chain.slddrw**.

2 Select the spline, then click **Broken-out Section** (Drawing toolbar) or **Insert**, **Drawing View**, **Broken-out Section**.

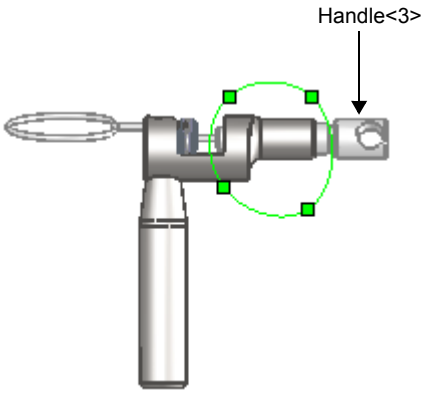
3 In the graphics area, select **Handle<3>** for **Excluded components**.

4 Click **OK**.

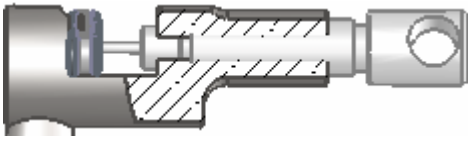
5 In the PropertyManager:

a) Set **Depth** to **50**.

b) Click .

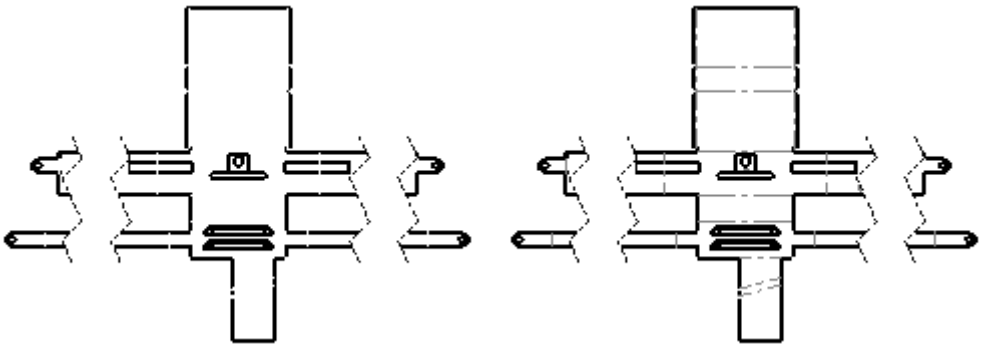


The broken-out section applies to only one component.



Broken Views

When you create a broken view for sheet metal parts, bend lines remain visible. Previously, bend lines were not supported. The bend lines are also trimmed appropriately to the break lines.



SolidWorks 2005 SolidWorks 2006

Section Views

When you create a section view, you can specify the distance of the cut so the section view does not create a cut of the entire drawing view.

To specify the distance of a section view:

1 Open **drawings\board\board.slddrw**.

A full section view already exists.

2 Select the section line.

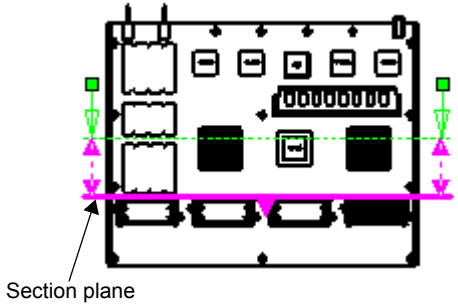
3 In the PropertyManager, under **Section Depth**:

a) Select **Distance**.

b) Set **Depth** to **65**.

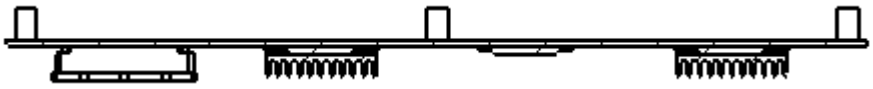
c) Click **Preview**.

All components between the section line and section plane will be shown in the section view.



4 Click .

The section view cuts the model to the distance specified.



3D Drawing View

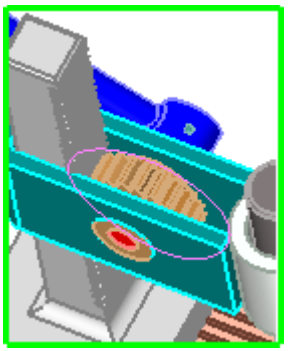
3D drawing view lets you rotate a drawing view out of its plane so you can see components or edges obscured by other entities. When you rotate a drawing view with 3D drawing view, you can save the orientation as a new view orientation.

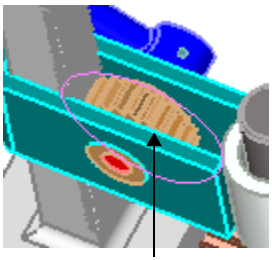
To manipulate a drawing view in 3D drawing view:

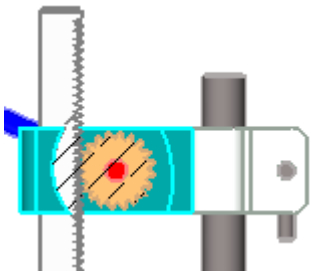
1. Open **drawings\press\press.slddrw**.
2. Select the spline, then click **Broken-out Section** (Drawing toolbar) or **Insert**, **Drawing View**, **Broken-out Section**.
3. In the dialog box, click **OK**.
4. Click **3D Drawing View** (View toolbar), or **View**, **Modify**, **3D Drawing View**.

   A pop-up toolbar appears with **Rotate** selected.
5. Rotate the drawing view to the orientation so you can see the top of the model. Use **Zoom to Area** on the pop-up toolbar to zoom in to the model if necessary.
6. Press **Esc** to exit rotate mode.
7. Select the edge as shown, then click **Exit** in the pop-up toolbar.
8. Click in the PropertyManager.

   The broken-out section view is created and the view returns to its original orientation.







Edge Display

New functionality for the **Hide Edge** and **Show Edge** tools include:

- Ability to hide and show multiple edges in a drawing view at the same time.
- Highlighting of hidden edges (in orange) for easier selection.
- Pre- or post-selection of edges to hide.

To hide and show multiple edges:

1 Open **drawings\bolt.slddrw**.

2 Click **Hide Edge** (Line Format toolbar).

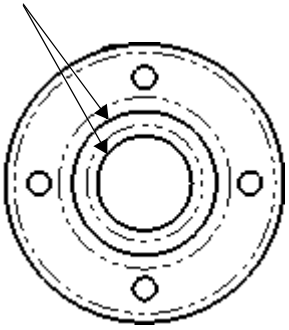
3 In **Drawing View1**, select the edges shown.

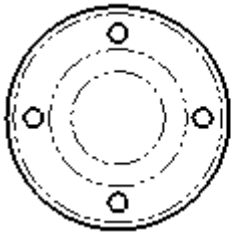
4 After you select the second edge, right-click to accept the selections.

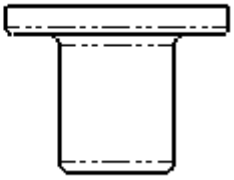
5 Click **Show Edge** (Line Format toolbar).

6 In **Drawing View2**, select the two orange edges.

7 After you select the second edge, right-click to accept the selections.







7

Detailing

This chapter describes enhancements to detailing in the following areas:

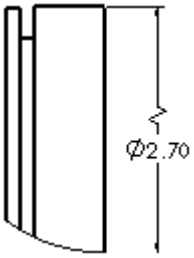
- Dimensions
- 3D annotations
- Notes
- Sheet metal bend notes
- Spelling
- Options
- Tables

Dimensions

You can now create foreshortened diameters in drawing documents. If the software detects that the diameter will be too large for the drawing view, the dimension is foreshortened.

You can also dimension to foreshortened radii and diameters. When you dimension to a foreshortened radius or diameter, the dimension also appears as a zigzag.

To create a foreshortened diameter:



1 Open **detailing\diameter.slddrw**.

2 Hold down **Shift** and drag the dimension **2.70** to the detail view.

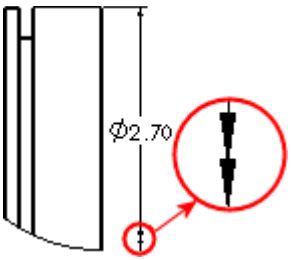
The dimension is shown with a zigzag and is properly scaled.

3 Click **Options** (Standard toolbar) or **Tools**, **Options**.

4 On the **Document Properties** tab, select **Arrows**.

5 Under **Foreshortened diameter**, select the double arrows, then click **OK**.

The diameter dimension is updated.



To create a foreshortened radius:

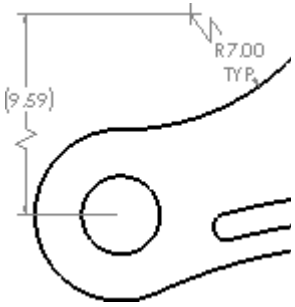
1 Open **detailing\radius.slddrw**.

2 Right-click the dimension **R7.00** and select **Display Options**, **Foreshortened**.

3 Click **Smart Dimension** (Dimensions/Relations toolbar) or **Tools**, **Dimensions**, **Smart**.

4 Insert a dimension from the foreshortened radius and the center of the hole as shown.

The dimension **9.59** appears as a foreshortened radius.



5 Press **Esc** to exit dimension mode.

6 Right-click the dimension **R16.00** and select **Display Options**, **Foreshortened**.

The dimension **16.85** becomes foreshortened because it applies to the radial dimension.

3D Annotations

SolidWorks part and assembly documents now support 3D annotations according to the ASME Y14.41-2003 standard. The annotations are displayed parallel to the plane on which they reside.

If you create a drawing from the model, the SolidWorks software organizes 3D annotations according to the model's orthographic views, such as front, bottom, isometric, etc. These orientations are called annotation views, and they replicate the standard drawing view orientations.

After you create annotation views in the model, you can use these views in a drawing. The annotation views are converted into 2D drawing views; the annotations you inserted in the model are retained in the drawing.

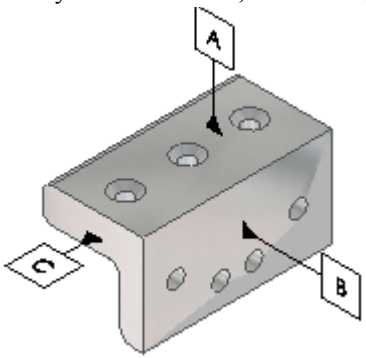
Inserting 3D Annotations

To insert 3D annotations:

1 Open **detailing\bracket.sldprt**.

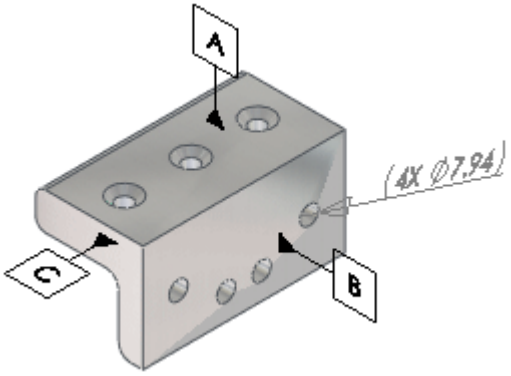
2 Click **Datum Feature** (Annotation toolbar) or **Insert**, **Annotations**, **Datum Feature Symbol**.

3 Insert the datum feature symbols as shown, then click .



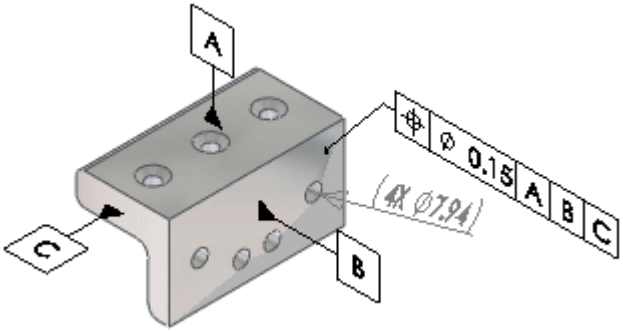
4 Click **Smart Dimension** (Dimensions/Relations toolbar), or **Tools**, **Dimensions**, **Smart**.

5 Dimension the hole as shown.



6 With the large model face selected, click **Geometric Tolerance** (Annotation toolbar) or **Insert**, **Annotations**, **Geometric Tolerance**.

7 Create the geometric tolerance shown, then click **OK**.



8 Save **bracket.sldprt**, and keep it open for the next procedure.

Creating Annotation Views

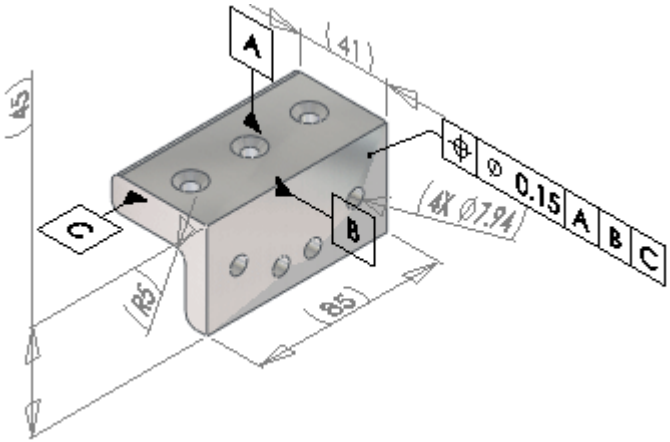
Annotation views represent 2D drawing views in a 3D model. If you have annotation views enabled when you insert 3D annotations, the annotations belong to specific annotation views.

To create annotation views:

1 Open **detailing\bracket.sldprt** if you do not have it open from the previous procedure.

2 Click **Smart Dimension** (Dimensions/Relations toolbar) or **Tools**, **Dimensions**, **Smart**.

3 Dimension the part as shown, then click .



4 Save **bracket.sldprt**, and keep it open for the next procedure.

Creating a Drawing from Annotation Views

After you create annotation views in a model, you can use these views in a drawing.

To create a drawing from a model with annotation views:

1 Open **detailing\bracket.sldprt** if you do not have it open from the previous procedure.

2 Click **Make Drawing from Part/Assembly** (Standard toolbar) or **File**, **Make Drawing from Part**.

3 In the PropertyManager, under **Number of Views**, select **Multiple views**.

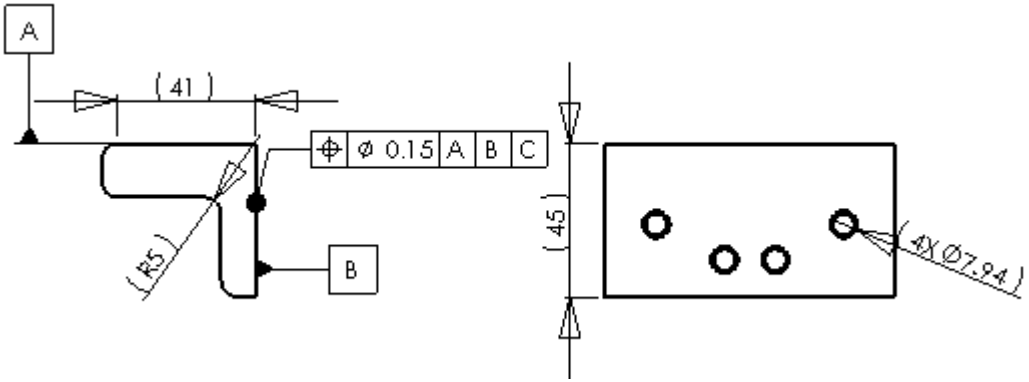
4 Under **Orientation**:

a) Select **Annotation view**.

b) Click ***Front** and ***Right** .

5 Click .

The drawing views appear with annotations.



Notes

In drawings, you can resize the bounding box around a note by:

- Typing a note first, then resizing the bounding box.
- Creating the bounding box first, then typing the text in the box.

You can also lock the note in place so it is not inadvertently moved.

To create a bounding box for a note:

1 Open a new drawing.

2 Zoom to the lower right corner of the title block.

3 Click **Note** (Annotation toolbar), or **Insert**, **Annotations**, **Note**.

4 Click and drag a bounding box inside the title block as shown.

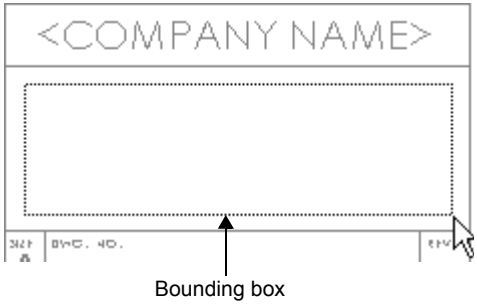
5 Drag the handles to size the box as necessary.

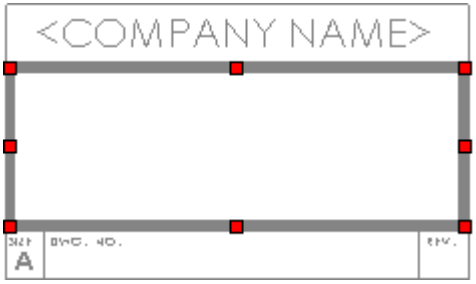
6 Type **USB PORT AND CABLES**.

7 In the PropertyManager, under **Text Format**, click **Lock/Unlock note**.

8 Click .

The note is locked in place.





Sheet Metal Bend Notes

Flat patterns inserted into drawing documents can display bend notes. See **Sheet Metal Bend Notes** on page 8-6.

Spelling

You can now check the spelling in documents with the **Spell Checker** tool. This tool spell checks notes, dimensions that include text, and drawing title blocks (when in **Edit Sheet Format** mode). For multiple users, you can specify a central dictionary location.

Spell Checker:

- Does not check words in a table
- Is available with Microsoft Word 2000 or later

To specify a central dictionary location:

1 Click **Options** (Standard toolbar), or **Tools**, **Options**.

2 Select **File Locations**.

3 Select **Spelling Folders** in **Show folders for**.

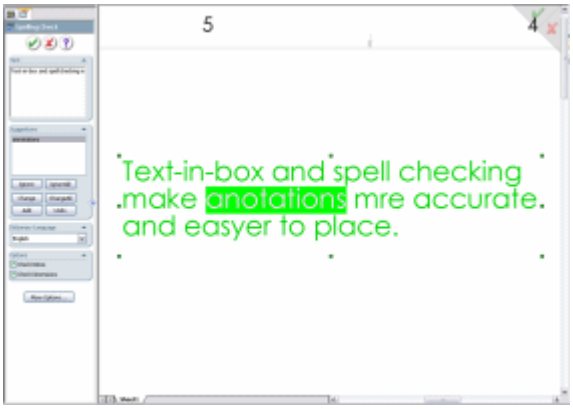
4 Click **Add** and browse to the dictionary location.

5 Click **OK** twice.

To check spelling:

1 In a document, click **Spell Checker** (Tools toolbar), or **Tools**, **Spelling**.

The first misspelling is found. In the PropertyManager, under **Suggestions**, the tool offers alternate spellings.



2 Click **Change**.

3 The tool continues to check misspelled words. Select the correctly spelled word in the PropertyManager and click **Change** until all misspellings are corrected.

4 When the spelling check is complete, click .

Options

In **Tools**, **Options**, **Document Properties**, **Detailing**, you can set **Trailing zeroes** to **Standard**. This option trims trailing zeroes to the ASME Y14.5M-1994 standard.

Tables

General Enhancements

Enhancements to tables include:

- **Row** PropertyManager. When you select a row in a table, this PropertyManager allows you to move the row up or down. You can also access this PropertyManager from several PropertyManagers.
- Moving tables. Click and drag the move icon in the upper-left corner of the table.

Bills of Materials

New functionality for bills of materials (BOMs) allow you to:

- Change the **BOM Type** (**Top level only**, **Parts only**, or **Indented assemblies**) without losing manual edits to the table. When you change the **BOM Type**, the document's file size increases and may slow performance. It is recommended that you delete the original BOM, then insert a new BOM with the desired **BOM Type**.
- Restore the original value of a cell if you manually edited its data by right-clicking the cell and selecting **Restore Original Value**.
- Eliminate the label, **QTY**, from the configuration column.
- Expand or collapse sub-assemblies in indented BOMs.
- Show numbers for indented sub-assemblies.
- Set increment values for item numbers in the BOM.

To eliminate the QTY label from a BOM:

1 Open **drawings\press\press.slddrw**.

2 Click **Options** (Standard toolbar) or **Tools**, **Options**.

3 On the **Document Properties** tab, select **Tables**.

4 Select **Don't add "QTY" next to configuration name**.

You must select this option prior to inserting a BOM. If you select it after a BOM exists, the option has no effect.

5 Click **OK**.

6 Select the drawing view, then click **Bill of Materials** (Table toolbar) or **Insert**, **Tables**, **Bill of Materials**.

The configuration appears in the table without the **QTY** label.

To show numbers for indented sub-assemblies in a BOM:

1 Open **drawings\chain\chain.slddrw**.

The BOM lists indented sub-assemblies, but they are not numbered.

2 Click anywhere in the BOM, then click **Table Properties** in the PropertyManager.

3 In the PropertyManager:

a) Under **BOM Type**, select **Show numbering**.

Sub-assemblies are listed with numbers that correspond to the main assemblies.

b) Click **BOM Contents**.

4 In the dialog box, click **-** in the row for **Hyper shot**.

The symbol changes to **+**, indicating that the sub-assemblies for **Hyper shot** are not visible.

5 Click **OK**.

6 Click .

ITEM NO.	PART NUMBER
1	main body
2	Handle
3	Handle
3.1	Plunger body
3.2	Extractor pin
3.3	Extractor pin knurled nut
4	Hyper shot
5	Chain retainer

General Tables

You can create general tables to use in drawings. These tables allow similar functionality to other SolidWorks tables, such as split, merge, sort, and equations.

To insert a general table:

1 In a drawing document, click **General Table** (Table toolbar) or **Insert**, **Tables**, **General Table**.

2 In the PropertyManager, under **Table Size**, set **Columns** and **Rows** to **3**.

3 Click .

4 Click in the graphics area to place the table.

5 Click any cell in the table, then click **Table Format** in the PropertyManager.

6 In the **Table** PropertyManager, under **Header**:

a) Select **Display title**.

b) Type **WIRE RUN LIST** for **Title**.

7 Double-click cell A2.

8 Type **FROM 'D' CONNECTOR P1**.

9 Double-click each cell in the table, and type the text as shown:

	A	B	C
1	WIRE RUN LIST		
2	FROM 'D' CONNECTOR P1	WIRE COLOR	TO 9 PIN CONNECTOR P2
3	1	GRN	4
4	2	BLK	2

10 Right-click the table and select **Formatting**, **Entire Table**.

11 In the dialog box, set:

- **Column Width** to **70**.
- **Row Height** to **25**.

12 Click **OK**.

8

Sheet Metal

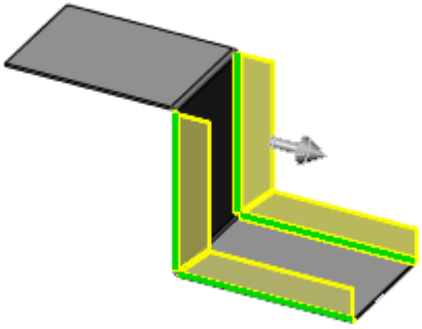
This chapter describes enhancements to sheet metal in the following areas:

- Edge flanges
- Closed corners
- Rips using sketch entities
- Sheet metal gauge tables
- Flat pattern color and font options
- Sheet metal bend notes
- Forming tools

Edge Flanges

Multiple Edge Selection

You can select multiple linear edges, contiguous or non-contiguous, and create edge flanges in a single operation. With contiguous edges, the software trims to 45°.



Closed Corners

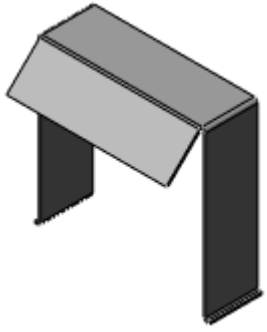
Enhancements:

- Close non-perpendicular corners
- Create closed corners on flanges whose bends do not equal 90°
- Change the overlap/underlap ratio
- Change the gap dimensions

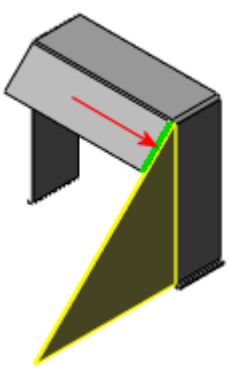
To add closed corners:

1 Open **SheetMetal\closed_corner.sldprt**.

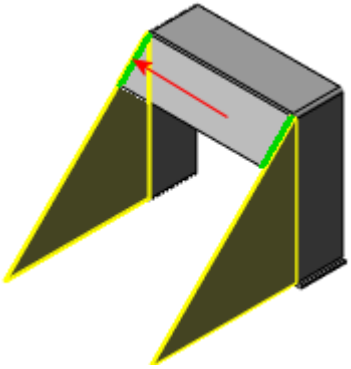
2 Click **Closed Corner** (Sheet Metal toolbar), or click **Insert**, **Sheet Metal**, **Closed Corner**.



3 Select the left and right faces of **Edge-Flange1**, as shown, for **Faces to Extend** .



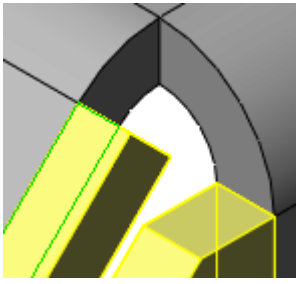
Right face of edge-flange



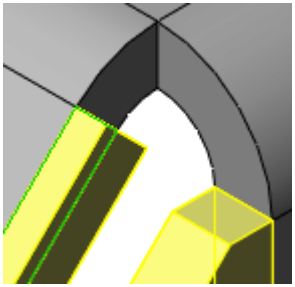
Left face of edge-flange

4 Select **Butt** for **Corner type**.

5 Increment the default value of the **Gap distance** by **1**.

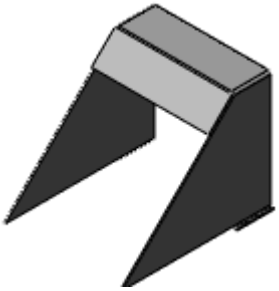


Default gap distance



Gap distance increments by 1 unit.

6 Click .



Rips Using Sketch Entities

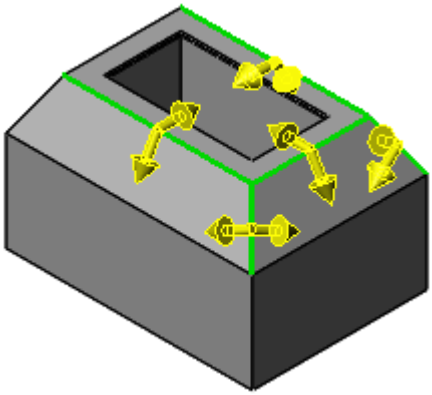
The rip functionality is now more robust. You can now use linear sketch entities as the driving geometry in sheet metal rips, combine sketch entities and existing edges, and use both internal and external edges to rip.

To use rip with linear sketch entities:

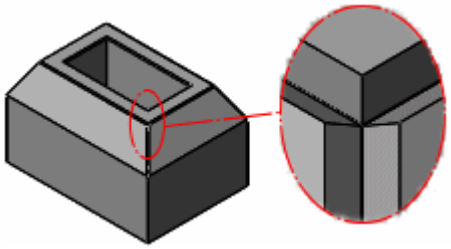
1 Open **sheetmetal**/**shm_rip.sldprt**.

2 Click **Rip** (Sheet Metal toolbar), or click **Insert**, **Sheet Metal**, **Rip**.

3 Select the edges shown for **Rip Parameters** .



4 Click .



Sheet Metal Gauge Tables

With the sheet metal gauge table, properties of a designated material are stored in tables. Tables are included with the SolidWorks application, and are located in:

<*install_dir*>**lang**\<*language*>**Sheetmetal Gauge Tables**\

Sheet Metal Bend Tables assign:

- Gauge thickness
- Allowable bend radii
- K-factor

To apply the sheet metal gauge table:

1 Create a base flange.

2 In the PropertyManager, under **Sheet Metal Gauges**, select **Use Gauge Table**.

3 Select a **Gauge Table** .

- or -

Click **Browse** to locate the folder with sheet metal gauge tables.

The table briefly appears and populates the **Sheet Metal Parameters**:

- species the **Thickness: Sheet Metal Gauge**.
- adjusts the **Bend Radius** .
- assigns the K-factor under **Bend Allowance**.

Flat Pattern Color Options

You can edit colors representing the following entities in flat patterns:

- **Bend Lines - Up Direction**
- **Bend Lines - Down Direction**
- **Form Feature**
- **Bend Lines - Hems**
- **Model Edges**

To set flat pattern colors:

1 With a drawing document open, click **Options** (Standard toolbar) or **Tools**, **Options**.

2 On the **Document Properties** tab, select **Sheet Metal**.

3 Select the flat pattern entity and click **Edit**.

4 Select a color from the **Color** palette.

Sheet Metal Bend Notes

Flat patterns inserted into drawing documents can display bend notes. Bend notes allow:

- **Editing content**. Edit text outside the note parameters (see **Modify Bend Note Format**).
- **Modifying default format**. Edit <*install_dir*>**lang**\<*language*>**bendnoteformat.txt**
- **Maintaining associative capabilities**. Change the angle or direction in the part, or move the bend line, and the notes update in the drawing.
- **Positioning notes**. Change the display position of bend notes.

To clear display of bend notes:

1 With a drawing document open, click **Options** (Standard toolbar) or **Tools**, **Options**.

2 On the **Document Properties** tab, select **Notes**.

3 Under **Bend notes**, clear **Display sheet metal bend notes**.

To display bend notes in a drawing when **Display sheet metal bend notes** is cleared, right-click on the drawing view, select **Properties**, and select **Display sheet metal bend notes**.

To add and position bend notes:

1 Open **SheetMetal\sheet_metal_cover.sldprt**.

2 Click **Make Drawing from Part/Assembly** (Standard toolbar).

3 In the **Sheet Format/Size** dialog box, click **OK**.

4 In the PropertyManager:

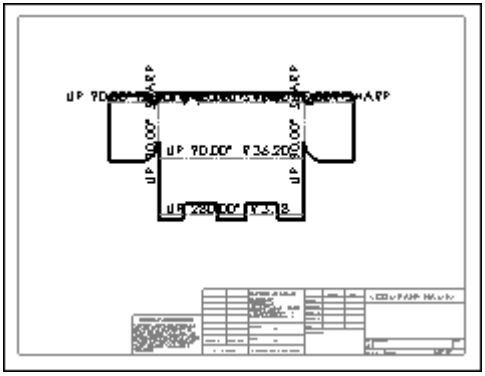
a) Under **Orientation**, in **More views** , select **Flat pattern**.

b) Under **Options**, clear **Auto-start projected view**.

c) Under **Scale**, select **Use custom scale**, and type **1:3**.

5 Click to place the view on the drawing sheet.

6 Click **Rotate** (View toolbar) and rotate the drawing 90°.

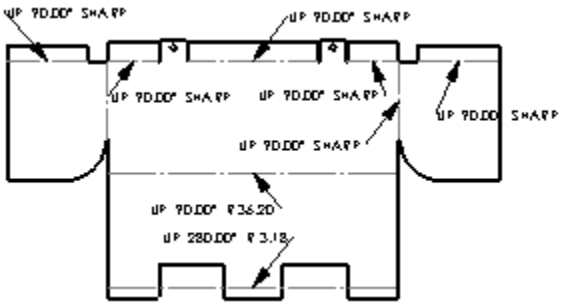


If the font is too large, block-select the bend notes, and in the PropertyManager, under **Text Format**, click **Font**, and resize.

7 Select bend notes individually to:

- Drag into a legible position
- Rotate the note
- Add leaders

You cannot individually delete bend notes, but you can right-click the bend note, and select **Hide**.



Sample repositioning of bend notes

Display Bend Note Position

You can display bend notes:

- **Above Bend Line**
- **Below Bend Line**
- **With Leader** (to all bend lines)

If you select above or below the bend lines, you can also add note leaders individually or to all bend notes simultaneously while in the drawing document.

To change the display position of bend notes:

1 With a drawing document open, click **Options** (Standard toolbar) or click **Tools**, **Options**.

2 On the **Document Properties** tab, select **Notes**.

3 Under **Bend Notes**, select an item in **Style**.

Modify Bend Note Format

To modify the content of bend notes:

1 Open **bendnoteformat.txt** located in <*install_dir*>**lang**\<l*anguage*> using a text editor.

```
**********************************************************************
* (c)2004 Solidworks Corp.
* Sheetmetal Bend Note Format Definition File
* Created: September 18, 2004
**
* Description:
**
**
**********************************************************************
**                                                    DISPLAY TYPE 1
<bend-direction> <bend-angle> <bend-radius>
<bend-direction-up>=UP
<bend-direction-down>=DOWN
<bend-radius-zero>=SHARP
```

Do not edit values inside the brackets.

You add values outside the brackets, or edit the default values appearing to the left of the = sign.

2 Add or edit existing values outside the brackets **<>**.

3 Save the edits.

The values inside the brackets are parameters that maintain associative capabilities. You can add an associative parameter by following the format inside the brackets. Text outside the brackets is not governed by format.

Forming Tools

The **Forming Tool** was added to the Sheet Metal toolbar.

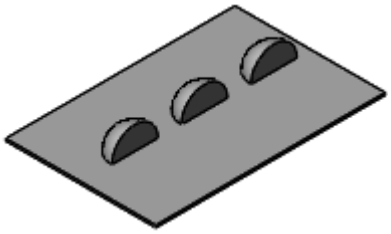
The steps to create form tools were streamlined. When you create a part, using the **Forming Tool** allows you to specify:

- **Stopping Face**
- **Faces to Remove**

The **Orientation Sketch** and the colors that identify the stopping faces are added automatically. Once you save the part to the forming tools folder in the **Design Library**, you can add the forming tool to a sheet metal part.

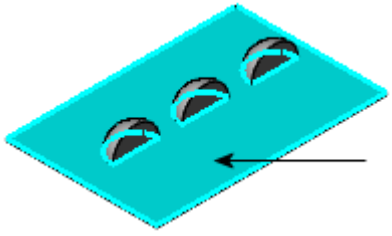
To create form tools:

1 Open Sheet Metal**halfshell3.sldprt**.

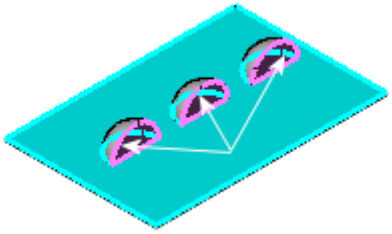


2 Click **Forming Tool** (Sheet Metal toolbar), or click **Insert**, **Sheet Metal**, **Forming Tool**.

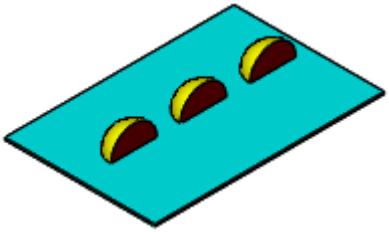
3 Select the face on which the three revolve features are located, as shown, for **Stopping Face**.



4 Select each of the faces, as shown, for **Faces to Remove**.



5 Click and save the part to <*install_dir*>**data****design library****forming tools**.



9

Other Functionality

This chapter describes enhancements to functionality in the following areas:

- Application Programming Interfaces
- Import/Export
- Installation
- Reference Geometry

Application Programming Interfaces

Click **Help**, **SolidWorks and Add-Ins API Help Topics** to access the SolidWorks, eDrawings, FeatureWorks, PDMWorks, PhotoWorks, SolidWorks Routing, SolidWorks Toolbox Browser, and SolidWorks Utilities Application Programming Interface (API) Help systems. The PhotoWorks and SolidWorks Routing APIs are new in this release.

Also included in this release is the SolidWorks for OEM partners. Contact SolidWorks API Support at **apisupport@solidworks.com** for details.

The major enhancements and changes made to the SolidWorks API in SolidWorks 2006 are:

- **Task Pane**. You can add tabs and ActiveX controls to the Task Pane.
- **Library features**. Support for library features is included in this release.
- **Dimension tolerance**. New and previously existing dimension tolerance APIs now reside in the new DimensionTolerance interface, which makes it easier to find and use dimension tolerance functions.
- **OLE objects on the active document**. You can create and access OLE objects on the active document.
- **WizardHoleFeatureData2 objects and hole wizard features.** You can create a WizardHoleFeatureData2 object and use its user-interface settings to create a hole wizard feature. You can also use an existing hole wizard feature to create another hole wizard feature.
- **Objects that correspond to Dispatch objects in components and parts**. You can get any object that corresponds to a Dispatch object in a component or part (for example, a feature, annotation, or sketch object).
- **Patterned and mirrored cosmetic threads in models**. Support for patterned and mirrored cosmetic threads in models is included in this release.
- **New SelectionMgr APIs**. You can mark selected entities, which lets you work with only those selected entities with the specified marks.

- **New FeatureData objects**. These new FeatureData objects are included in this release: CoreFeatureData, LibraryFeatureData, SimulationGravityFeatureData, SimulationLinearSpringFeatureData, SimulationMotorFeatureData, WeldmentBeadFeatureData, and WeldmentTrimExtendFeatureData.
- **Lightweight enhancements**. Lightweight assemblies and lightweight drawings have been enhanced so that parts need not be resolved as often. These enhancements should not affect existing API applications that include lightweight-related functions; however, it is recommended that you test your applications using the enhanced lightweight functionality available in this release of the SolidWorks software.
- **RegenPostNotify AssemblyDoc event**. Applications that listen for the RegenPostNotify AssemblyDoc event as the last event fired when a component is added to an assembly might not work because of the performance enhancements made to the Solidworks software. You should test your applications that rely on this notification in this context and modify them as needed.
- **swdisp.cpp and swdisp.h**. These files are no longer included in the SolidWorks software: swdisp.cpp and swdisp.h. If you need these files, you can recreate them.
- **Old-style toolbar button images**. The old-style toolbar button images that are 16 colors and 15x16 pixels (small) or 22x22 pixels (large) are not supported in this and later releases of the SolidWorks software. Instead, you should create toolbar button images that are 256 colors and 16x16 pixels (small) or 24x24 pixels (large).

Import/Export

AutoCAD

You can import from and export to AutoCAD® R2006.

DXF/DWG

You can import 3D faces from DXF/DWG files.

You can export splines and ellipses.

Multi-sheet drawings now include the following export options:

- **Export active sheet only**
- **Export all sheets to separate files**
- **Export all sheets to one file**

DWGeditor

You can copy and paste entities and blocks from SolidWorks to DWGeditor™. Click **Edit**, **Copy to DWGeditor** to copy entities, including:

- 2D entities
- Blocks
- Drawing views
- Annotations

The standard commands for copying and pasting are not supported for copying to the DWGeditor because the entities must be translated to DWG format.

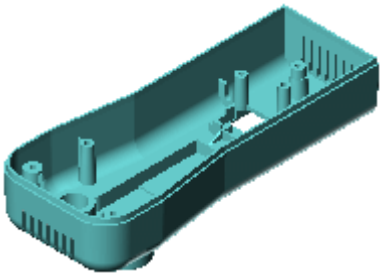
Import Diagnostics

Import Diagnostics has been added to the Tools toolbar and the **Tools** menu. When you import a file in a non-SolidWorks format, a message prompts you to run Import Diagnostics.

The **Import Diagnostics** PropertyManager has been redesigned as follows:

- Faulty faces and gaps are listed when the PropertyManager is opened.
- Messages inform you of the model status and results of actions.
- **Attempt to Heal All** might fix the model in one step.
- Additional manual steps are available on the shortcut menus.

To diagnose and repair an imported model:



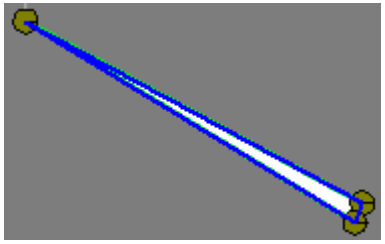
1 Open **other_functionality\diagnostics1.x_b**.

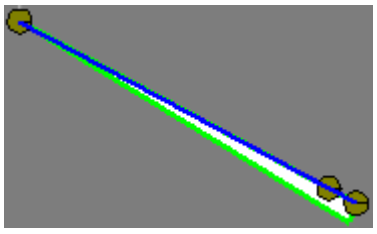
2 Click **Yes** to the message **Do you wish to run Import Diagnostics on this part?**.

3 In the PropertyManager, click **Attempt to Heal All**.

The listing of gaps under **Gaps between faces** disappears and a message states that no faulty faces or gaps remain.

To repair a gap with the gap closer:





1 Open **other_functionality\diagnostics2.x_b**.

2 Click **Yes** to the message **Do you wish to run Import Diagnostics on this part?**

3 In the PropertyManager, click **Attempt to Heal All**

Two gaps remain.

4 Right-click **Gap<2>** and select **Gap Closer**.

The gap is defined with lines and handles .

5 Drag the lowest handle to the upper line (not to the upper handle).

The lower line turns green, indicating that the gap can be closed.

6 Right-click **Gap<2>** and select **Finish Gap Closer**.

7 Select **Gap<1>**.

This gap is the boundary of the whole surface, which is normal.

Import Surface Repair

When imported IGES surface data is not mathematically smooth, SolidWorks tries to repair the surfaces instead of splitting them at discontinuities.

Solid Edge

You can now import Solid Edge sheet metal parts with the file extension .**psm**.

Adobe Illustrator

You can import Adobe® Illustrator® (***.ai**) files with the SolidWorks **Open** command.

Installation

You can customize your installation to include the **Export to PDF** option which exports SolidWorks documents in PDF format. During installation, in the **SolidWorks Option** dialog box, click **Customize** to access the **Export to PDF** option.

Reference Geometry

You can create variable pitch helixes by entering different pitches along the curve at distances you specify.

To create a variable pitch helix:

1 Open **Other_Functionality\WormGear.sldprt**.

2 Select **Sketch3**.

3 Click **Helix and Spiral** (Curves toolbar) or **Insert**, **Curve**, **Helix/Spiral**.

4 In the PropertyManager, under **Defined By**, select **Pitch and Revolution**.

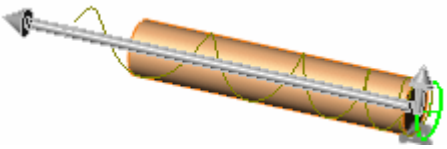
5 Under **Parameters**:

a) Select **Variable Pitch**.

b) In **Region Parameters**, the first two rows of the table are pre-populated. Click the appropriate cell in row three and type:

Rev	Dia	P
5.5	75	200

6 Click in the graphics area to see a preview. Select **Reverse direction** if the preview does not span the length of the cylinder.

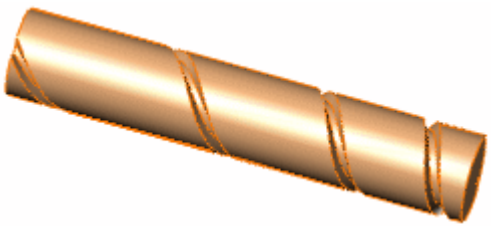


7 Click .

8 Click **Swept Cut** (Features toolbar) or **Insert**, **Cut**, **Sweep**.

9 In the PropertyManager, select:

- **Sketch4** for **Profile** .
- **Helix/Spiral1** for **Path** .



10 Click .

10

SolidWorks Office Add-Ins

This chapter describes enhancements to the SolidWorks Office Add-Ins in the following areas:

- eDrawings
- FeatureWorks
- PDMWorks
- PhotoWorks
- SolidWorks Animator
- SolidWorks Design Checker
- SolidWorks Routing
- SolidWorks Utilities

eDrawings

3D Annotations

The eDrawings software supports display of 3D annotations. When you rotate models in the eDrawings Viewer, 3D annotations remain visible.

COSMOSWorks

Assembly Structure

eDrawings now supports assemblies containing COSMOSWorks analysis files.

Multiple Plots in a Single eDrawings File

With COSMOSWorks, all plots are now output to a single eDrawings file.

To create a single eDrawings file of multiple plots, right-click **Study** in the COSMOSWorks Manager tree and select **Save all plots as eDrawings**.

eDrawings Administrative Install

Administrative install is supported by eDrawings 2006. Refer to Microsoft for Windows Installer information.

eDrawings Publisher for IntelliCAD

An eDrawings Publisher for IntelliCAD® is available.

Embedding Items

OLE Objects

- eDrawings supports OLE objects inserted in DWG files.
- You can embed eDrawings files in Word and PowerPoint files as OLE objects.

To embed eDrawings files in Word files:

1. In Microsoft Word, open a new document.
2. Click **Insert**, **Object**.
3. Select **eDrawings 2006 Control** for **Object type**, then click **OK**.
4. Right-click the **eDrawings Control** box and select **Properties**.
5. For **FileName**, click ..., browse to any eDrawings file to insert, then click **OK**.
   The path to the file appears.
6. Select **EnableFeatures**, then browse to the **Property Pages** dialog box.

7 Set the options, then click **OK**.

8 Close the **Properties** box, then exit **Design Mode** to view the eDrawings file.

For PowerPoint, view the slide show to display the eDrawings file.

Raster Images

eDrawings supports insertion of BMP, GIF, JPG, PNG, and TIFF images as raster images in DWG files. The image must accompany the DWG file. GIF, PNG, and TIFF images are supported on Windows XP only.

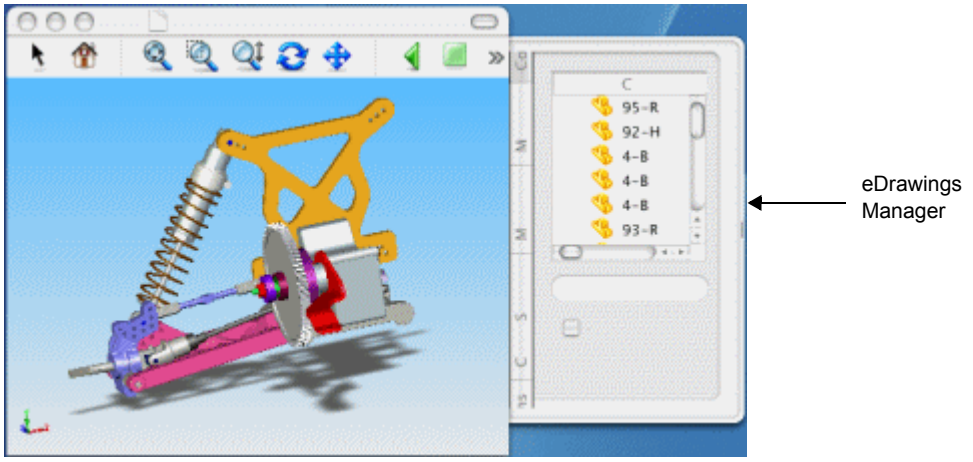
Macintosh

eDrawings is now supported on the Macintosh® operating system versions Panther 10.3.3 and higher. You can open DXF/DWG, SolidWorks, and eDrawings files (eDrawings 1.x files are not supported).

The complimentary eDrawings software for the Mac® operating system (eDrawings for Mac) contains most of the functionality of the Windows version including animation, measure, and print. Opening review-enabled files activates the eDrawings Professional tools (markup, move, cross-section, etc.). There is no eDrawings Professional for Mac.

The user interface is similar to the Windows version with these exceptions:

- Some tools that use buttons in Windows are only accessible via menu items.
- The eDrawings Manager opens outside the graphics window. Drag a button next to the graphics area to expand the eDrawings Manager.



- To customize the toolbars, right-click in the toolbar area and select **Customize Toolbars**.
- Unsupported items: UI Mode, Save/Send as exe/HTML/zip file

- Macintosh only:

  - Search for component in eDrawings Manager.

  - Multiple selection from eDrawings Manager.

  - Spotlight importer. Finds all eDrawings files that contain a specific component, configuration, or markup item (Tiger only).

  - Drag an eDrawings file into another document, such as a text file.

Mozilla Firefox and Netscape Browsers

You can use the Mozilla Firefox and Netscape browsers (Windows operating system only) to view eDrawings files embedded in HTML files. A plug-in is required for Mozilla Firefox. Netscape 7.1 or higher comes with the plug-in installed.

To check if the plug-in is installed:

In Mozilla Firefox or Netscape, type **about:plugins** as the URL.

You have the plug-in if the list shows:

- **Mozilla Active X control and plugin support**
- **File name: npmozax.dll**
- **Mozilla AcitveX control and plugin module**

If you have the plug-in, skip to **To enable the eDrawings control:**. Otherwise, install the plug-in.

To install the plug-in:

1 In your browser, type **http://www.iol.ie/~locka/mozilla/plugin.htm** as the URL.

2 Under **Download the precompiled Plug-in**, find your browser's version, then download the ***.xpi** file.

3 Click **File**, **Open File**, browse to the ***.xpi** file, then click **Open**.

This installs the plug-in automatically.

To enable the eDrawings control:

1 In the browser's installed directory, click **defaults**, **pref**, and edit the **activex.js** file.

2 Type this code at the bottom of the file as a single line:

pref("capability.policy.default.ClassID.CID22945A69-1191-4DCF-9E6F-409BDE94D101", "AllAccess");

3 Save the file.

4 In an HTML editor, open **SolidWorks_Office\eDrw_Browser.htm** and view the source.

5 In the **OBJECT** tag information, you must have the **CLASSID** information. If you manually insert an eDrawings file, you must enter the full path to the file for the **value**. In the sample file,
value="C:\ProgramFiles\SolidWorks\samples\whatsnew\SolidWorks_Office\ToolVise.EASM"
indicates the location of the source eDrawings file. The calling HTML file does not need to be in the same location.

6 Restart the computer.

To view the embedded eDrawings file using Mozilla Firefox or Netscape:

1 In your browser, click **File**, **Open File**, and browse to **SolidWorks_Office\eDrw_Browser.htm**.

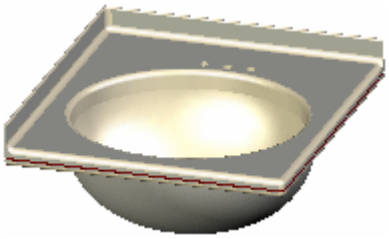
The HTML file displays the embedded eDrawings file **ToolVise.easm**.

2 Right-click in the graphics window to display the shortcut menu options such as **View**, **Animate Views**, and **Print**.

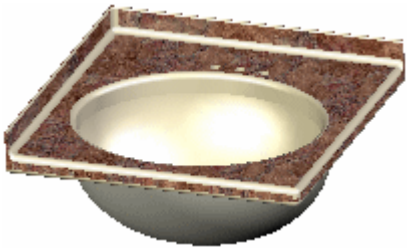
If the eDrawings file is not visible, make sure the path in the **eDrw_Browser.htm** file matches the installed location of **ToolVise.easm**.

SolidWorks Textures

eDrawings supports SolidWorks textures, materials, cosmetic weld textures, and cosmetic thread textures. Because eDrawings does not support RealView, turn it off in SolidWorks to see how the model will look in the eDrawings Viewer.



eDrawings 2005 - Textures not supported



eDrawings 2006 - Textures supported
Marble countertop shown

FeatureWorks

Hole Wizard

Additional Types Supported

FeatureWorks recognizes these holes:

- Countersink (ANSI Metric standard only)
- Tap (ANSI Metric standard only)
- Pipetap (ISO standard only)

Supported recognition methods:

- Automatic
- Interactive
- Re-recognition. You can re-recognize cut revolve features as countersink, tap, or pipetab features and vice versa.

Additional Standards Supported in Step-by-Step Recognition

FeatureWorks supports step-by-step recognition of all standards and all types of Hole Wizard holes contained in a model (not in an imported body). Previously FeatureWorks supported only the ANSI Metric standard.

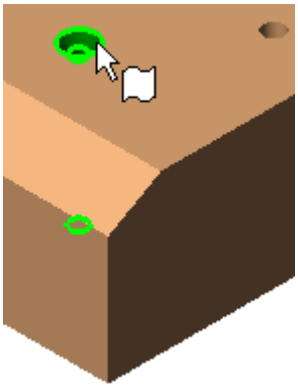
To view the enhanced FeatureWorks recognition of Hole Wizard holes:

1. Open **SolidWorks_Office\FWKS_HoleWiz.x_t**.
2. Click **No** if prompted to run Import Diagnostics on the part.
3. Click **No** if prompted to proceed with feature recognition.
4. Click **FeatureWorks Options** (FeatureWorks toolbar) or **FeatureWorks**, **Options**.
5. Select **Recognize holes as wizard holes**, then click **OK**.
6. Click **Recognize Features** (FeatureWorks toolbar) or **FeatureWorks**, **Recognize Features**.
7. In the PropertyManager:
   a) Under **Recognition Mode**, select **Interactive**.
   b) Under **Feature Group**, select **Standard features**.
   c) Under **Interactive Feature**, select **Hole** in **Feature type**.

d) Select the two faces of the lower-right hole shown for **Selected entities**.

e) Select **Recognize similar**.

f) Click **Recognize**.



8 Select **Show**.

9 In the Recognized features manager, right-click **Hole1** and select **Re-Recognize As**, **Revolve** to re-recognize this Hole Wizard feature as a cut revolve.

Re-recognition as cut extrude is not available for the newly supported Hole Wizard standards.

10 Click **Map Features** to display one Hole Wizard CounterSink hole and one cut revolve in the FeatureManager design tree.

11 Save the file as **HoleWiz.sldprt**.

To continue feature recognition using step-by-step and automatic recognition:

1 Click **Recognize Features**.

2 Select:

- **Automatic** recognition
- **Standard features**
- **Holes** and **Fillets/Chamfers**.

3 Click **Recognize**.

4 Click **Map Features**.

FeatureWorks recognizes two Hole Wizard tapped holes plus multiple fillets and chamfers. The **Imported** feature remains in the FeatureManager design tree because you have not yet recognized extrude features.

Miter on Multiple Edges

FeatureWorks supports interactive recognition of miter flanges on multiple edges instead of just single edges.

Miter recognition has limitations. See the online help for details.

To recognize miter flanges on multiple edges:

1 Open **SolidWorks_Office\FWKS_Miter.x_t**.

2 Click **No** if prompted to run Import Diagnostics on the part.

3 Click **Yes** if prompted to proceed with feature recognition. Otherwise, click **Recognize Features**.

4 In the PropertyManager:

a) Under **Recognition Mode**, select **Interactive**.

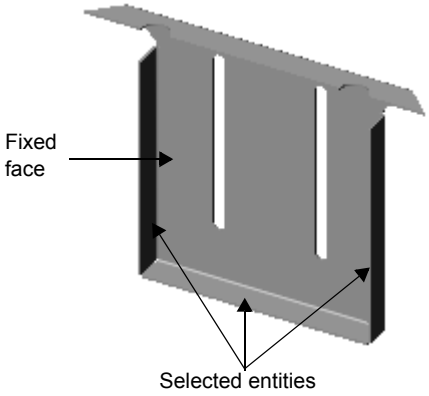
b) Under **Feature Group**, select **Sheet metal features**.

c) Under **Interactive Features**, select **Miter Flange** in **Feature type**.

d) Select the faces shown for **Selected entities** (three inner faces) and **Fixed face**.

The faces must be all inner or all outer. Mixed selection is not permitted.

5 Click **Recognize** then **Map**.



PDMWorks

Selection

Select multiple documents using **Ctrl** and **Shift** (as in Windows) in the vault view, in the local view, and in dialog box grid views.

You can select multiple documents:

- **Open/Check Out Document**
- **Delete Document**
- **Change Project**
- **Take/Release Ownership**
- **Add/Remove Attachment**

Grid View

The user interface is now a grid view that allows changing document properties for a single cell, a group, or a column in these dialog boxes:

- **Open**/**Check Out Document**
- **Reporting**
- **Search Results**

Clients

Attach Document

You can attach non-SolidWorks documents to a document in the Vault.

If you drag a document from the local view into a project in the vault view, PDMWorks offers to attach the document you are checking in to a Vault document.

In the **Check In Document** dialog box, you can click **Attach to another document?** to specify a document in the Vault.

Shortcut menu items are available in the vault view for **Add Attachment** and **Remove Attachment**.

Explain Document Access

Right-click a document in the Vault and select **Explain Access**. The dialog box lists read/write access and the vault settings regarding access. You can specify which revision to view. The vault administrator can restrict client access (**Open**, **Document Information**, **Search**, **Reporting**, and viewing and browsing in the Standalone and Web Portal clients) based on lifecycle status.

Lifecycle Status

If an attempt to change document lifecycle statuses fails, a message appears with possible explanations for the failure. In the **Change Lifecycle Status** dialog box, scope is restricted to the selected item unless the revision is the latest.

Conflict Icon

A new icon informs users of conflicts . The icon appears in the **Check In Document** dialog box to indicates a conflict in situations such as the following: User A checks out a document but does not have ownership. User B checks out the same document, changes it and checks it back in. If user A then changes the document and checks it back in, a conflict is noted. The icon also appears in the local view in SolidWorks 2006, but not in SolidWorks 2005.

Open/Check Out

The **Open Document** and **Check Out Document** dialog boxes are now combined into the **Open / Check Out Document** dialog box with a grid format and options for **Get Local Copy**, **Take ownership**, and **Open now**.

Collaboration

PDMWorks can check the Vault at a specified time interval to determine if any documents currently opened read-only have been modified. The vault administrator must have selected **Bind ownership to SolidWorks read-write / read-only access** on the VaultAdmin **Vault Settings** tab. Users are notified the same way as in SolidWorks and given the opportunity to reload or take ownership.

Document Information

In addition to the new grid layout and editing capabilities, the **Document Information** dialog box includes an **Attachments** tab and can remain open while you view the Vault.

Client Options

- Include COSMOS result files with check in, check out, and open operations for files created in SolidWorks 2006 and later.
- Specify ranges for displaying groups of documents within a project in the vault view. Ranges apply to the first level of documents in a project and help improve performance.

Usernames

Usernames are now displayed in the case entered by the vault administrator in VaultAdmin. Previously, they appeared in the case entered at login. This affects the display in the vault view and listings under **History**, for example.

Search Results

In the **Search Results** dialog box:

- Icons indicate the file type of documents.
- You can select multiple documents for editing, apply the changes in the dialog box, and save the changes to the documents in the Vault.
- Actions available for documents are restricted based on document ownership as specified by the vault administrator; for example, clients might not be allowed to edit the lifecycle status, project, owner, etc.

VaultAdmin

Vault Startup

You are notified when the vault startup is complete and the Vault is ready for use.

Vault Rebuild

The options on the **Vault Settings** tab for starting the Vault have changed. Under **On vault startup**, select or clear **Validate vault** and (if selected) choose either **Next startup** or **Every startup**. When **Next startup** is selected, **Validate vault** is cleared after the tree has been successfully rebuilt.

Regardless of the option setting, the Vault tree is rebuilt during the initial installation and upgrade installations.

Log File

Only one log file is created now, and it is created only on request. On the **Vault Settings** tab, click **Save** under **Vault Log** to save the current log file with a name you specify, or click **Clear** to start a new file. A row of asterisks appears in the log whenever the server starts up, along with information on the version and build of PDMWorks.

Options Removed

The **Allow Notifications** and **Allow Sync** options have been removed from the **Vault Settings** tab.

eDrawings

The vault administrator can grant or deny **Measure**, **Markup**, and **Save** functionality to all Web Portal and SolidWorks clients of eDrawings in the Vault regardless of whether the client has an eDrawings Professional license.

To see the changes in the VaultAdmin settings, clients must be restarted because the eDrawings control is created only once in a client session.

Revision Scheme

The default scheme on the **Revision Scheme** tab has changed so that the primary revision is now a range of **A** to **Z** instead of a list of **PRE**, **DEV**, and **REL**.

Lifecycle

Lifecycle status is now specific to a revision rather than to a document, which enables ISO compliance.

The options have been reorganized and expanded. When you select a status to specify rules, the name of the status is displayed in the rules section of the tab. You can allow:

- Revision level changes based on the status either in the vault before check in or selected in the check-in dialog box.
- Only administrators to change the status of previous revisions.
- Automatic changing of a document's lifecycle when checking in the document at the same lifecycle. For example, document at revision **B** with lifecycle status **Released** is in the Vault; check in a document with revision **C**, give it lifecycle status **Released**, and the rule changes the document with revision **B** to **Obsolete** lifecycle status.
- Validation on check in of the revision based on the revision rules of the selected lifecycle.

Standard Libraries

The **Toolbox** tab has been renamed **Standard Libraries**. When you add folders for standard libraries, you specify a **Display Name in Vault** in addition to a **Path**.

If a folder is not revision-managed, you can choose to view it in the vault view (under **Library Components**), although the documents reside on the specified path and not in the Vault.

Web Portal

eDrawings

When a Web Portal client views a document in the Vault in eDrawings, PDMWorks checks that the version of eDrawings is the same or a more recent version than PDMWorks. If not, the newest version of eDrawings is installed automatically.

You can use **Measure**, **Markup**, and **Save** functionality regardless of whether you have an eDrawings Professional license if the vault administrator grants permission.

Access

Web Portal clients can view only revisions for which they have at least read-only access. Also, access can be restricted based on the document's lifecycle status.

API

New APIs:

- **Search**
- **Create/Delete Root Level Project**
- **Create/Delete Group**
- **Create/Delete User**

PhotoWorks

PhotoWorks Items

A new tab, **PhotoWorks Items**, has been added to the Task Pane. PhotoWorks must be added in for the tab to be available. Default folders include:

- **scenes**
- **lights**
- **decals**

When you drag a scene or decal from the **PhotoWorks Items** tab into the graphics area, the corresponding PropertyManager or dialog box appears. No PropertyManager appears for light schemes.

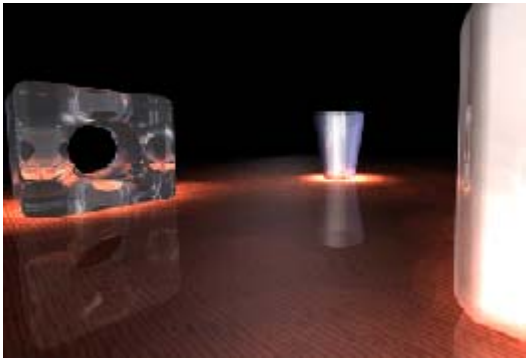
When you double-click scene or light files on the **PhotoWorks Items** tab, the scenes or lights are attached to the active document. When you double-click a decal, the **Decal** PropertyManager opens, but the decal is not inserted into the graphics area.

Lightweight Rendering

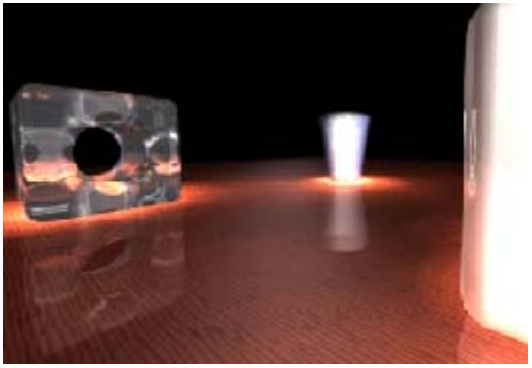
You can render lightweight models without resolving the assembly. The Render Manager tree is not available in lightweight mode.

Camera

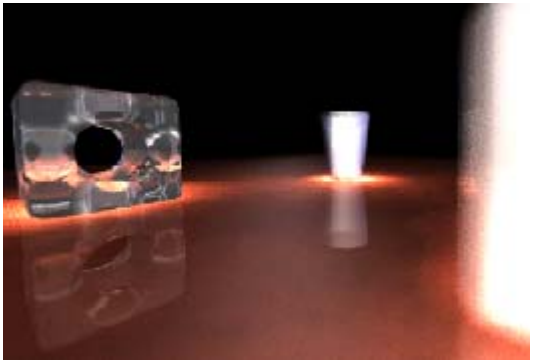
You can insert cameras and view the model from the cameras' perspectives. See **Cameras** on page 1-3. PhotoWorks renders all aspects of SolidWorks camera views; for example, differences in field of view.



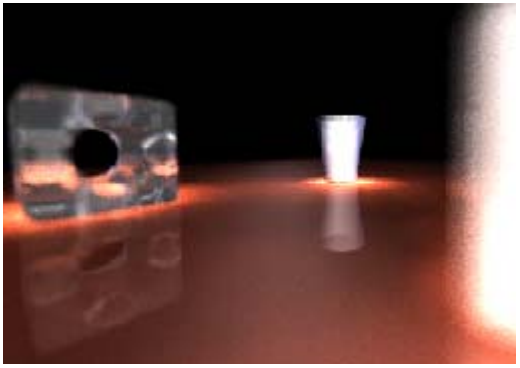
No field of view



Near field of view



Middle field of view



Far field of view

Decals

You can drag decals from the **PhotoWorks Items** tab in the Task Pane. You can size, move, and rotate decals with on-screen manipulators and specify properties in the **Decal** PropertyManager. You can now see decals in the SolidWorks graphics area in addition to the **Preview** window. The Decal Wizard has been removed.

Label (UV) mapping (horizontal and vertical lines of the underlying parameterization of a curve) maps decals onto the faces of models in a way that is analogous to placing an adhesive label on a real part. Label mapped decals are not stretched or shrunk when they are mapped to multiple non-planar surfaces.

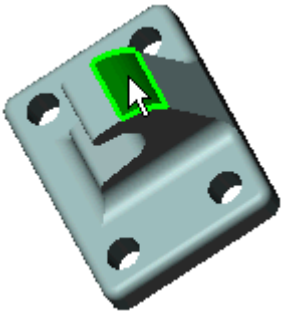
The **Decal** PropertyManager tabs include:

- **Image** (**Decal Preview**, **Mask Image**)
- **Mapping** (**Selected Geometry**, **Mapping**, options to render the image)
- **Illumination** (select material type and illumination properties)

To apply a decal:

1 Open **solidworks_office\anchor_plate.sldprt**.

2 On the Task Pane **PhotoWorks Items** tab, expand **decals**, select the **decals** folder, then drag **Barcode** from the lower pane onto the face shown.



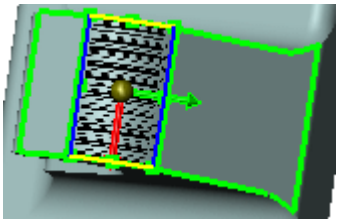
> To apply a decal to a whole part, drag the decal into an empty region of the graphics area (not onto the part).

3 In the PropertyManager, on the **Mapping** tab:

a) Clear **Fixed aspect ratio**.

a) Select **Fit width to selection**.

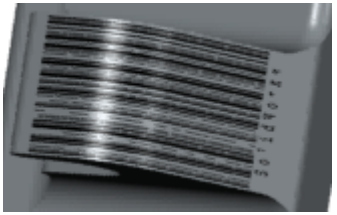
b) In the graphics area, select the two adjacent faces as shown. The faces are listed under **Selected Geometry**.



4 Drag the handles to stretch the decal over the three selected faces.

5 In the PropertyManager, click **Render Area**.

6 In the graphics area, drag the pointer to define the area to be rendered.

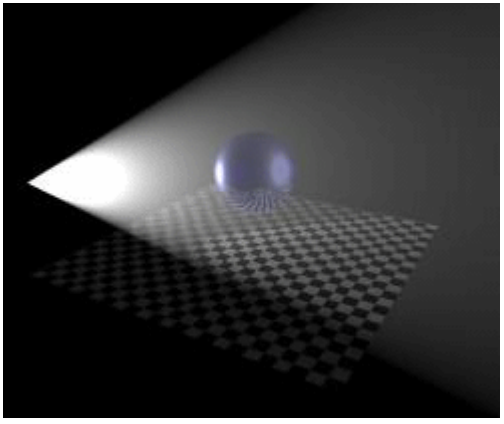


7 Click .

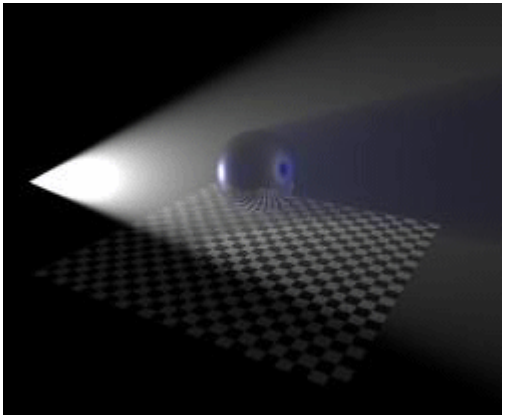
Fog Lights

When **Fog Quality** in a spot light or point light PropertyManager is set to **Low**, fog is evaluated in the default mode, which ignores objects blocking the light, so these objects do not cast shadows. This setting is fast and appropriate for many situations.

For any other quality setting, volumetric shadows are calculated by taking objects blocking the light into account. Higher quality settings increase rendering time because the calculations use numerical integration.



Fog Quality set to **Low** - no fog shadows



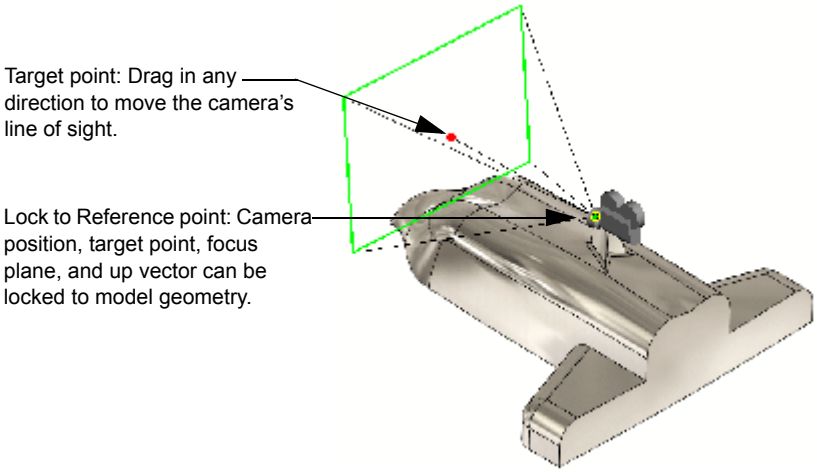
Fog Quality set to **High** - volumetric shadow displayed

Macros

Switching macros on and off is now handled in SolidWorks rather than PhotoWorks, so **Record actions to macro files** has been removed from the **General** tab in PhotoWorks Options.

SolidWorks Animator

Cameras can modify the view of static objects, and interface with SolidWorks Animator. You can apply camera attributes (see **Cameras** on page 1-3) to static or animated views.



You can play an animation while viewing the model through the camera. If a camera is attached to a component that moves during the animation, you view the animation through the camera, showing the effects of the component's movement.

For example, attach a camera to a spaceship. As you animate the spaceship, it enables you to see what the camera sees as it approaches and passes stationary objects along the motion path.

To simulate the camera view in an animation:

1 Open **Animator/camera_animation.sldasm**.

2 Select the **Animation 1** tab at the bottom of the graphics area.

3 Click **Camera View** (View toolbar) or **View**, **Display**, **Camera**.

4 Click **Play from start** (Animator toolbar).

To see how the camera was configured:

1 In the FeatureManager design tree, expand **Lights and Cameras**.

2 Right-click **Camera1** and select **Properties** to display the PropertyManager.

3 Note the selected parameters (see **Cameras** on page 1-3 for descriptions).

SolidWorks Design Checker

SolidWorks Design Checker is an add-in that verifies design elements such as dimensioning standards, fonts, materials, and sketches to ensure that a SolidWorks document meets pre-defined design criteria. You set the requirements by which the document will be evaluated, then SolidWorks Design Checker assesses the document.

To add SolidWorks Design Checker to SolidWorks:

1 Click **Tools**, **Add-Ins**.

2 Select **SolidWorks Design Checker**, then click **OK**.

To set the requirements for SolidWorks Design Checker:

1 Click **Build Checks** (Design Checker toolbar) or **Design Checker**, **Build Checks**.

2 In the **SolidWorks Design Checker** window, click **Open** or **File**, **Open**.

3 Browse to **Other_Functionality\requirements.swstd** and click **Open**.

The document shows several pre-selected design criteria, such as **BOM Balloon Check**, **Dimension's Units Setting Check**, etc.

4 On the **Document Checks** tab, click **Dimensioning Standard** or **Checks**, **Document**, **Dimensioning Standard**.

5 In the **Document's Dimensioning Standard Check** section (at the bottom of the window) select **ISO**.

6 On the **Annotation Checks** tab, click **GTOL Datum** or **Checks**, **Annotation**, **GTOL Datum**.

This requirement identifies any GTOLs with missing datums. There is nothing further to select.

7 On the **Drawing Document Checks** tab, click **Title Block** or **Checks**, **Drawing Document**, **Title Block**.

8 In row 1 of the **Title Block Check** section:

a) Type **Author** for **Name**.

b) Select **Text** in **Type**.

c) Select **List** in **Operator**.

d) Type **Bob** for **Value**.

9 Click **New**.

10 Repeat step 8 for the second row, but type **Company** for **Name** and **SolidWorks** for **Value**.

11 Click **Save** or **File**, **Save**.

To run SolidWorks Design Checker:

1. In the SolidWorks software, open **Other_Functionality\gear.slddrw**.
2. Click **Check Active Document** (Design Checker toolbar) or **Design Checker**, **Check Active Document**.
3. In the dialog box, browse to **requirements.swstd**, then click **Open**.
4. In the **Results** dialog box, expand **Passed Checks**.

   Items that passed the requirements checks are listed.
5. Expand **Failed Checks**.

   Items that failed the requirements checks are listed.
6. Expand **Document's Dimensioning Standard Check**, then select **gear - Sheet1**.

   The **Expected Value** was set to **ISO**, but the **Actual Value** in the drawing is **ANSI**, which is why this check failed.
7. Click **Save Report**.
8. In the dialog box, type a value for **Report Name**, then click **Save**.
9. Open the report, **SWDCReport.xml**, to view the **Design Check Analysis Report**.

   The report lists criteria used to evaluate the drawing, and shows the passed and failed checks.

SolidWorks Routing

System Options

(piping, tubing, and electrical cabling)

New options in **Tools**, **Options**, **System Options**, **Routing**:

- **Automatically route on drop of flanges**. Select to automatically create a route sub-assembly and start a route when a routing component such as a flange, tube fitting, or electrical connector is dropped into an assembly.
- **Enable insertion of empty route assemblies**. When selected, you can create an empty route assembly by clicking **File**, **New**, **Assembly** and selecting the **routeAssembly** template on the **Advanced** page of the **New SolidWorks Document** dialog box. Assemblies created from the **routeAssembly** template must be used *only for route modeling*.
- **Use automatic naming for route parts**. When selected, the software automatically assigns names for route parts. When cleared, the software asks you to enter a name whenever a route part or assembly is created.
- **Always use default document template for routes**. When selected, the software automatically uses the default template specified the in the box below. When cleared, the software asks you to specify a template whenever a route assembly is created.

Adding Components

(piping and tubing only)

Enhancements:

- You can rotate elbows, tees, and crosses during placement by holding down **Shift** and pressing the left and right arrow keys.
- The software extends the route sketch to continue the route. For elbows, the software adds an arc that follows the radius of the elbow and a line on the other end of the elbow. For tees and crosses, the software adds lines for each opening in the connector.
- You can add back-to-back fittings. For example, you can attach an elbow to an elbow or a tee to a tee without a pipe or tube in between. Flange-to-flange connections are not supported.

Additional Electrical Attributes

(electrical cabling only)

You can add electrical attributes to include custom data. To view or modify attributes in the **Edit Wires** PropertyManager, under **Additional Properties**:

- Type a **Name** and a **Value** for a new attribute, then click **Create/Edit**.
- Click **More Properties** to view a table of the attributes for the route.

Conduit

(electrical cabling only)

You can create electrical conduit designs.

To create a conduit design:

1. Create a route with components such as a junction box or a switch box. These components have a new connection point sub-type called **Conduit**.
2. Extend the route between the components with pipes and elbows.
3. Add electrical components such as plugs or ring terminals that mark the end of the cables or wires.
4. Use the **Auto Route** function to join the electrical components to the ends of the conduit route to indicate the path for the cables or wires through the conduit.
5. Add electrical properties to the cables or wires.

Connection Points

(piping, tubing, and electrical cabling)

To create a connection point on a cylindrical feature, select the flat face of the feature. You no longer need to create a sketch point first.

Coverings

(piping, tubing, and electrical cabling)

You can add coverings to any route. Coverings represent material such as insulation or split sleeves placed on routed items to protect them. Functionality includes:

- Add the covering during or after route creation.
- Display the covering in assemblies, drawings views, and Bills of Materials.
- Select a material for the covering.

Electrical Bill of Materials

(electrical cabling only)

You can add a bill of materials that displays items specific to electrical routing. The bill of materials is titled “Circuit Summary” and includes conductor-wire ID, color, length, and from/to information.

To create an electrical bill of materials:

1 Create a drawing of an electrical routing assembly.

2 Click **Bill of Materials** (Table toolbar) or **Insert**, **Tables**, **Bill of Materials**.

3 In the PropertyManager, under **Table Template**, browse for **bom-circuit-summary.sldbomtbt**.

4 Under **BOM Type**, select **Indented assemblies**.

5 Click to add the table to the drawing.

Exporting Data

You can export piping and tubing data as a **PCF** file, for use in other applications such as ISOGEN™. The **Create bend tables** command has been renamed to **Export pipe/tube data**.

Fixed Length Route

(tubing and electrical cabling only)

You can fix the length of a route segment. For example, if the connectors for a fixed length wire move closer to or farther apart from one another, the route does not change its length.

To fix the length of a route:

1 Edit a route.

2 Right-click a spline and select **Route Properties**.

3 In the PropertyManager, select **Fix length** then set the parameters under **Fixed length**.

Graphics-only Cables

(electrical cabling only)

You can display cables without generating a part file for the cable. This improves performance when an assembly has many cables. You cannot select a graphics-only cable in the graphics area and the cable does not appear in the FeatureManager design tree. To activate, edit a route then right-click the route in the FeatureManager design tree and select **Toggle display mode**.

Minimum Bend Radius

(tubing and electrical cabling only)

The software now helps prevent violations of the minimum bend radius of the route. Right-click a spline in the graphics area or right-click a violating wire in the **Edit Wires** PropertyManager and select **Repair route**. A new route that does not violate the minimum bend radius is created.

Re-Route to Omit Clips

(tubing and electrical cabling only)

You can update a route if you delete a clip from the routing sub-assembly. The next time you edit the route, the software informs you that a clip has been deleted and asks if you want to update the route. If you re-route, the route updates to a more direct path ignoring the original clip.

If a clip is still in the assembly, but you no longer want the route to pass through it, you can choose to ignore the clip in the route. While editing the route, right-click the sketch segment under the clip and select **Unroute from clip**.

Re-Route Through Clips

(tubing and electrical cabling only)

You can update a route to travel through selected clips.

To re-route through clips:

1. Edit the route.
2. Click **Auto Route** (Electrical toolbar) or **Electrical**, **Auto Route**.
3. In the PropertyManager, under **Routing Mode**, select **Re-route spline**.
4. Select the spline and the axis of one or more clips, then click twice.

   The route updates to pass through the clips.

Rotate Route Components

(tubing and electrical cabling only)

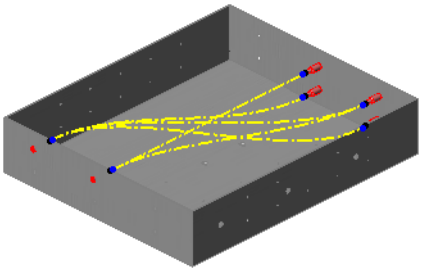
You can rotate clips in a routing sub-assembly. As you drag them into the sub-assembly, you can hold down **Shift** and press the left and right arrow keys. After they are in the sub-assembly, you can right-click the component and select **Add/Edit Mates (Rotate Clip)**. Shortcut menu items include:

- **Align Axis**. Aligns the component's axis with an edge or axis in the assembly. The component must have an axis named "Clip Axis."
- **Rotate Clip**. Sets the angle of rotation. Components must have an axis named "Axis of Rotation."

Route Guidelines

(electrical cabling only)

Previously, there was no way to preview the wires in a cabling route after import of the from/to data. Previews, called guidelines, now appear when the **Auto Route** PropertyManager is active. These guidelines show the connections of the wires that need to be bundled into finished routes. You have options to display the guidelines, to update them on changes, and to filter their visibility by length.



Additionally, you have options for converting guidelines into routes as follows:

Merge guidelines to form routes. Merges two or more guidelines to merge into a single route.

Join guidelines to existing routes. Joins a guideline into an existing route at a selected route point.

Convert guidelines to routes. Converts one or more guidelines into unique routes.

Merging or joining guidelines into routes works best if their paths are close together or approximately parallel to one another.

Split Routes

(piping, tubing, and electrical cabling)

You can split a route segment and apply different properties on each side of the split. To split a route while editing the route, use the **Split Entities** tool to split the sketch segment, then right-click the new point and select **Split Route**. A connection point appears in the FeatureManager design tree under the **Route** feature. The connection point can be renamed. **Split Route** is useful if, for example, you want to apply covering to only a portion of a segment. You split the segment into two segments using **Split Route** and apply covering to only one of the segments.

Standard Cables and Tubes

(tubing and electrical cabling only)

You can create a library of standard cables and tubes. The information for standard cables and tubes is stored in an Excel spreadsheet. Properties include length, diameter, end fittings, and configuration. Click **Electrical**, **Standard Cables** to access this functionality. A sample Excel spreadsheet is located at **<***install_dir***>\data\design library\routing\standard cables and tubes.xls**.

Virtual Clips

(tubing and electrical cabling only)

You can create a virtual clip to position a route during the initial stages of a design. The virtual clip does not require any solid geometry and you can exclude the clip from the Bill of Materials.

To create a virtual clip:

1. Create a new part.
2. Add one or two route points.
3. Add an axis to represent the path the route takes through the clip.
4. Optionally, add a mate reference to place the part when you add it to an assembly.

To route through a virtual clip:

1. Create a virtual clip with one or two route points and a clip axis.
2. Insert the virtual clip into the assembly.
3. Re-route through the virtual clip as described in **Re-Route Through Clips** on page 10-24.

Harnessing

Harness Board Drawing Formats

You can control SolidWorks user preferences (annotation details, title block details, etc.) in harness board drawings using standard SolidWorks drawing templates. Click **Wiring**, **Wiring Summary**, then in the **Wire Summaries** PropertyManager, browse to select a drawing template.

Splitting Wires in Harness Board Drawings

You can display individual wires emerging from an electrical cable or harness in harness board drawings. This is useful when only preparation of the wire end (no connectors or terminals) is needed. Click **Wiring**, **Edit**, **Parts**, then in the **Edit Parts** PropertyManager, select a part, then select **Display wire fan-out**.

Design Rule Checks

You can check electrical documentation for completeness against a set of design rules that you define. Click **Wiring**, **Design Rule Checks**, then in the **Design Rule Checks** PropertyManager, select the rules you want to check.

Partial Coverings

You can add coverings such as looms, shrink tubing, and tape, to a portion of a segment. Previously, they could be added only to the entire length.

Automatic Insertion of Wire Seals and Connector Plugs

You can set up the harnessing library to automatically add seal and plug components to the BOM with one button rather than having to add each individually.

SolidWorks Utilities

General Enhancements

The **Compare Features**, **Compare Geometry**, and **Compare Documents** utilities now compare two configurations of the same model.

To compare two configurations of the same model:

1 Click **Compare Documents** (Utilities toolbar) or **Utilities**, **Compare Documents**.

2 In the dialog box:

a) Under **Document1**, click **Browse**.

b) Open **SolidWorks_Office****KnobB.sldprt**.

c) Under **Document2**, click **Browse**.

d) Select **KnobB.sldprt**, but under **Configurations**, select **dome**, then click **Open**.

Under **Document2**, the part is renamed to **KnobB_dome.sldprt**.

e) Click **Compare**.

Two configurations of the same part are compared.

Compare Features

The results from the **Compare Features** utility are enhanced. The new **Compare Features: Results** dialog box has two trees, one for the reference part and one for the modified part. The trees contain two nodes: unique and modified features, and uncompared features.

To examine Compare Features results:

1 Open **SolidWorks_Office****KnobA.sldprt** and **KnobB.sldprt**.

2 Make sure **KnobB.sldprt** is in the **dome** configuration.

3 Click **Compare Features** (Utilities toolbar), or **Utilities**, **Compare Features**.

4 In the dialog box:

a) Select **KnobA.sldprt** in **Reference Part**.

b) Select **KnobB.sldprt** in **Modified Part**.

c) Click **Compare**.

The dialog box shows the comparison results in two trees, one for each part.

- Unique features are identified with red bullets and modified features with yellow bullets.
- The unique feature, **Fillet1**, in the modified part, is not listed under the reference part tree because it does not exist.

Power Select

Feature Types

The **Power Select** utility now supports the following features and entities:

- Area Fill Pattern
- Flex
- Indent
- Mate Reference
- Reference points
- Surfaces

Surface Types

The **Power Select** utility now supports a new filter: **Surface type**. The utility finds entities based on surface types such as planes, cylinders, cones, and so on.

Simplify

The **Simplify** utility now supports assembly documents. It simplifies all parts in the assembly by finding and suppressing small features, then creates a derived configuration. Features supported by this utility are:

- Chamfers.
- Extrudes. Boss, boss-thin, cut, cut-thin. (Base extrudes and extrudes that are not Blind or Mid-Plane are not found.)
- Fillets. Simple, multi-radius, face (without the hold line parameter), variable radius. (Full round fillets are not found.)
- Holes. Simple and Hole Wizard.

When you simplify an assembly, you can **Ignore features affecting assembly mates** so those features that would cause mate failures are not suppressed.

Assembly features are not found with the **Simplify** utility.

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