# SolidWorks 2007 What's New



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## Introduction

## **About this Book**

This book highlights and helps you learn the new functionality in the SolidWorks<sup>®</sup> 2007 software. It introduces concepts and provides step-by-step examples for many of the new functions.

This book does not cover all details of the new functions in this software release. For complete coverage of the new functions in the SolidWorks 2007 software, refer to the *SolidWorks Help*.

#### **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 *Quick Start* guide, complete the *Online Tutorial* lessons, and 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 2007 What's New Highlights**. This book provides the highlights of the new functionality in the SolidWorks software. This book is available in print format for new and upgrading 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.

#### Late Changes

This book may not include all of the enhancements in the SolidWorks 2007 software. Late changes are documented in *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 2007 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.

#### **Interactive Features**

Many of the functionality descriptions in *What's New* include 3D images or video animations:

- 3D images include this annotation: Click the image for 3D viewing.
- Animations are shown with the icon (>) that you click to activate them.

#### **Conventions Used in this Book**

Convention	Meaning	Example
Bold	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 <i>Quick Start</i> .
*	Tip	When you create a 3D model, first make the 2D sketch, then create the extruded 3D feature.

## Moving to SolidWorks 2007

## **Converting Older SolidWorks Files to SolidWorks 2007**

Opening a SolidWorks document from an earlier release may take extra time. 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 2007 format. To access the Conversion Wizard, click Windows **Start**, then **All Programs**, **SolidWorks 2007**, **SolidWorks Tools**, **Conversion Wizard**.

Two report files are created in the conversion folder:

- 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 for the SolidWorks 2007 software. To check for a new service pack, click **Help**, **Check for Updates**.

1

## SolidWorks Fundamentals

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

- Add to Library
- Auto-recover, Backup, and Save Notification
- Background Images
- CommandManager
- Documentation
- Error Reporting
- FeatureManager Design Tree
- Keyboard Shortcuts
- Numeric Input
- Open and Save Dialog Boxes
- Pack and Go
- Performance Feedback
- Screen Capture
- Select Other
- Task Pane
- Triad
- Undo
- Units and Dimension Standards
- View Menu

## Add to Library

Add to Design Library has been changed to Add to Library. An Add to Library 谢 tool has been added to the Task Pane Design Library tab.

The **Add to Library** PropertyManager opens when you click the tool in the Task Pane or drag an item from a SolidWorks document into the Task Pane. In the PropertyManager, you specify items to publish, file names, folder names, and other options.

## Auto-recover, Backup, and Save Notification

In Tools, Options, System Options, Backup/Recover, you can specify:

- To be reminded to save documents and at what interval.
- The folder for saving autorecover files.



• The number of days to retain backups.

If you choose to be reminded to save documents, a transparent message box appears if the current document has not been saved within the specified interval (minutes or number of changes). The message box contains commands to save the current document or all documents; it fades after a few seconds.

If auto-recovery is initiated, the saved files are available on the Task Pane **Document Recovery** tab when you next open the SolidWorks application. You can open individual recovered files and your saved files, or click **Open All**.

## **Background Images**

You can use an image supplied with the software, or your own image, as background for the graphics area and FeatureManager design tree area.

Click Tools, Options, System Options, Colors. Choose a scheme under Current color scheme, or browse to an Image file under Background appearance.



## CommandManager

In the CommandManager shortcut menu, **Show Description** has been replaced by **Use Large Buttons with Text**. You can also select or clear **Use large buttons with text** in **Tools, Customize, Toolbars**.

You can drag complete toolbars from the graphics area or window border into the CommandManager control area.

#### Documentation

Two new documents are shipped with the software media kits: the *Quick Reference Guide* and the *Quick Start Guide*. The *Quick Reference* is also available from the Help menu.

The *What's New* PDF document now includes interactive graphics. You can view images in 3D and play video animations.

The *Installation and Administration* compiled help file (.chm) combines and replaces a number of documents and has been streamlined for easier access.

## **Error Reporting**

You can stop the build process for each error so you can fix feature failures one at a time. To choose the action the software takes when it encounters errors during model creation or rebuild, in **Tools**, **Options**, **System Options**, **General**, select **Stop**, **Continue**, or **Prompt** in **When rebuild error occurs**. In the FeatureManager design tree, hover the pointer over an item with an error to see the explanation in a tooltip.

If you drag the rollback bar to a specific position, the model is rebuilt to that position regardless of errors. The model is not rebuilt if:

- The document is not the top-level document.
- The feature with the error is directly above the rollback bar.
- The part does not have any more rebuild errors during the current rebuild than it did in the previous one.

## FeatureManager Design Tree

The FeatureManager design tree expands, collapses, and scrolls only on request. To collapse all items, right-click and select **Collapse Items**, or press **Shift+C**.

To toggle display of the left panel (FeatureManager design tree, PropertyManagers, etc.) click i in the center of the panel border, click **View**, **FeatureManager Tree Area**, or press **F9**.

You can save models with the rollback bar placed anywhere. When you open the document, you can use rollback commands and drag the bar from the saved position.

## **Keyboard Shortcuts**

The interface for managing keyboard shortcuts allows you to redefine (add, delete, or change) shortcuts for all commands. You can assign multiple shortcuts to commands.

Customize			? 🗙
Toolbars Comr	nands Menus Keyboard Options		
Category: 📕	Commands 😽	Print List Copy List	
Only show commands with shortcuts assigned Reset to Defaults			
Search for:		Remove Shortcut	
Category	Command	Shortcut(s)	•
File	New	Ctrl+N	-
File	👌 Open	Ctrl+O	
File	📁 Close		

In the dialog box, select a command and press a key or key combination for the shortcut.

A message informs you if the key combination is already assigned to another command. If you decide to use the shortcut for the new command, it is removed from the old command.

You can view the commands by category and show all commands or only commands with shortcuts assigned within the categories. Text strings in the **Search for** field filter the **Command** list within the selected category.

Macro icons with shortcuts appear in the **Tools** category.

**Print List** brings up the **Print Setup** dialog box so you can print out the list currently selected. **Copy List** copies the current list to the clipboard so you can paste it into documents such as Word or Excel.

To reset all shortcuts to the system defaults, click **Reset to Defaults**.

See Customize Keyboard in the help.

#### **Numeric Input**

Horizontal sliders and thumbwheels have been added to numeric controls such as those in the **Camera** and **Colors** PropertyManagers.

100mm	*

Thumbwheel for numeric input

45deg		÷.
<u>^</u>		

Slider for angular values

30	*

Slider for color selection

## **Open and Save Dialog Boxes**

The view style (**Thumbnails**, **Tiles**, etc.) that you choose from the **View Menu** in the **Open** and **Save** dialog boxes is remembered the next time you access the dialog boxes.

You can select multiple files with Ctrl and Shift in the Open dialog box.

#### **Pack and Go**

**Pack and Go** copies files (parts, assemblies, drawings, references, design tables, Design Binder files, COSMOS<sup>®</sup> results, and PhotoWorks content) to a specified folder or zip file. In SolidWorks, click **File**, **Pack and Go**. In Windows Explorer or the SolidWorks Task Pane **File Explorer**, right-click a SolidWorks document and select **SolidWorks**, **Pack and Go** . In SolidWorks Explorer, click **Pack and Go** .

In the **Pack and Go** dialog box, the selected document is listed, along with any references. You can choose to include drawings and PhotoWorks files and add a prefix or suffix to the file names. If you save the results to a zip file, you can send it with an email.

See Pack and Go in the help.

#### **Performance Feedback**

Performance feedback is sent via web service rather than email. In **Tools**, **Options**, **System Options**, **General**, the option **Enable performance email** is now **Enable performance feedback**.

## **Screen Capture**

Click **Screen Capture** (Standard toolbar) or **View**, **Screen Capture** to copy the contents of the active window or viewport to the clipboard. You can then paste the image into other applications (Microsoft Word, Microsoft Excel, etc.).

The image is captured without user interface menus.



## **Select Other**

The **Select Other** dialog box displays icons for the types of elements in the list. Hidden faces are not displayed. The list shows entities in the order pierced. Highlighting is displayed in multiple views. You can scroll through the list with the middle mouse button.



## **Task Pane**

The **SolidWorks Resources** tab has a link to **User Groups**  $\Re$ , a web site for finding and joining SolidWorks user groups, and a link to **News**.

The **View Palette** Itab contains drawing views. See **View Palette** on page 5-5.

The **Document Recovery** is tab appears with auto-recover documents if your system has crashed. See **Auto-recover, Backup, and Save Notification** on page 1-2.

## Triad

The triad has been enhanced so that it attaches more easily to geometry. Moving the center sphere drags the object rather than the triad.

The triad is larger and, when appropriate, displays rotation rings. The rotation motion can snap to angle. Right-click the ring and select:

- Snap while dragging
- Rotate 90°
- Rotate 180°

See Triad in the help.



## Undo

The Undo command is available in more areas of SolidWorks, including:

- Sketch actions even after exiting the sketch. See **Undo** on page 2-11.
- Move and suppression of components.
- Dimension and annotation creation and modification in context of drawing.

## **Units and Dimension Standards**

When SolidWorks creates new templates on startup, the **Units and Dimension Standard** dialog box asks you to choose default units (**IPS**, **MMGS**, etc.) and dimension standard (**ISO**, **ANSI**, etc.). You can change the units and standards for individual documents later in **Tools**, **Options**, **Document Properties**. This replaces selection during SolidWorks installation.

## **View Menu**

Items added to the View menu:

Screen Capture	Lights	Full Screen (F11)	FeatureManagerTree Area (F9)
Parting Lines	Cameras	Toolbars (F10)	

See Screen Capture on page 1-6.

In Full Screen mode, the SolidWorks borders (including the FeatureManager area, Task Pane, and status bar) and top level menus are hidden. You can access the menus by hovering the pointer over the top of the screen. The visibility of the toolbars, status bar, Task Pane, and FeatureManager area is stored separately for Normal Mode and Full Screen mode.



## Sketching

This chapter describes enhancements to sketching in the following areas:

- 3D sketching
- Blocks
- Copy dimensions and relations
- Define sketch patterns
- Fully define sketch
- Relations with midpoints
- Sketch orientation behavior
- Sketching options
- SketchXpert
- Splines

## **3D Sketching**

## **Equal Relations**

You can add equal relations to all applicable sketch entities in 3D sketching.

## **Trim Entities**

All trim options, **Power trim**, **Trim away inside**, etc. are available with any 2D sketch created on a 3D sketch plane. Previously, you could use the trim tool, but not select the type of trim.

## Tangent to Face

You can add either **Tangent** or **Equal Curvature** relations between adjacent faces and 2D or 3D splines. Select the face, edge, and spline, and then add the relation.



## **Blocks**

Enhancements to blocks facilitate:

- Orienting and aligning sketches
- Modeling pulleys and chain sprockets
- Modeling cam mechanisms

## Align Grid/Origin

Previously, blocks inherited their origin location from the parent sketch. Now, the origin of a block is aligned to the orientation of the sketch entity.



#### Origin Location

You can change the origin location and orientation for blocks and sketches. Align Grid/Origin replaces the Align Grid.

#### To Use Align Grid/Origin with blocks:

- 1 Create blocks from several sketch entities.
- 2 In Edit Block mode, click Tools, Sketch Tools, Align, Align Grid/Origin.
- 3 In the PropertyManager, under **Selections**:
  - Select a vertex or point to change the origin
    for Sketch Origin Location.
  - Click in X axis (horizontal) or Y axis (vertical), and select a line to change the orientation of the sketch origin.



You can select internal or external sketch entities as references to define both the orientation and the location along the X or Y axis.



New origin (vertex) and new orientation (X axis)

4 Click 🖌

When you change block origin or orientation:

- Geometry within the block uses the new coordinate system.
- A warning shows if existing relations conflict with the new coordinate system. Delete the relations to apply the new coordinate system.
- There is no association between the block and the references you select after you apply changes.

#### Using Align Grid/Origin in Sketches

Functionality with sketches is similar to blocks except that you can either relocate:

- Only the origin to any of the sketch entities.
- All the sketch entities in the model.



See Align Grid/Origin in the help.

## **Convert Blocks to Construction Geometry**

You can toggle between normal and construction geometry with the **Construction Geometry** tool (Sketch tools). The construction geometry blocks icon is displayed with a dashed border in the FeatureManager design tree.



### **Traction and Belts**

With new relations and sketch entities you can use layout sketches to create these mechanisms:

- Multiple gear sets
- Cable and belt pulleys
- Chain sprocket systems

#### **Traction Relation**

The new **Traction** Register relation allows you to create relative rotation constraints between blocks used to represent pulleys or sprockets. When you add a **Traction** Register relation, it adds a tangent relation between circles, or between circles and linear entities.

• **Circles**. Simulate gear mechanisms by creating equal rolling distances between two or more circular entities.



Gear mechanism can include:

- Blocks, such as two or more circles, each is defined as a block.
- Relations and dimensions between the centers of the circles prevent displacement during rotation.
- Traction relations between the two outer circles and the inner circle.



You can add a fix relation to the center of a circle, but use horizontal and vertical relations, dimensions, or construction geometry to locate the centers of additional circles. Adding multiple fix relations potentially restricts the degrees of freedom.

• **Circles and linear entities**. Simulate a rack and pinion mechanism with a traction relation between the two entities.



Rack and pinion-type mechanism can include:

- Circle with a fix relation at the centerpoint.
- Blocks, such as a circle and linear sketch defined as blocks.
- Traction relation between the circle and the horizontal line.

See Blocks in Parts and Assemblies in the help.

## **Belt/Chain**

Simulate a cable-pulley or chain mechanism by sketching:

- Circles or arcs to represent pulleys, cogs, or sprockets.
- Continuous tangent lines and arcs to represent the belt or chain path.



All sketch entities along the path are tangent, but you cannot select any single entity independently.

#### To create a mechanism using Belt/Chain:

1 Open \sketching\belt\_chain.sldprt.





2 Click Belt/Chain 🐼 (Blocks toolbar) or Tools, Sketch Entities, Belt/ Chain. 3 In the PropertyManager, under **Belt Members**, select the three circles, left to right, for **Pulley components**  $n^0$  to add the belt around each circle.



4 Click the arrow pointing up on the small circle to make the belt move under the circle.



See Belt Chain in the help.

#### Make Path

A path enables you to create a tangent relation between a chain of sketch entities and another sketch entity. For example, you can model cam profiles where the tangency relation between the cam and a follower automatically transitions as the cam rotates.

Once you have created a path, all the sketch entities in the path are selected simultaneously. To select an individual sketch entity in the path, right-click and choose **Select Other**.



You can make a path within the block, or you can make a path on the block entities.

#### To create a path:

- 1 Open \sketching\create\_path.sldprt.
- 2 Select the two arcs and the two lines in the cam, and click Make Path (Sketch toolbar) or Tools, Sketch Tools, Make Path, then click (r) to close the PropertyManager.
- 3 Select any arc on the cam and the bottom arc line on the follower.

5 Select the center of the small arc and rotate the cam.

- $\langle \cdot \rangle$

🤣 See Make Path in the help.

## **Copy Dimensions and Relations**

With the **Copy Entities** tool you can copy dimensions and relations, along with the sketch entities to which they belong, to a new location. This capability applies to:

- Blocks or other sketch entities.
- Relations and dimensions between sketch entities.



• Either From/To or X/Y under Parameters



Select sketch entities or blocks with dimensions and relations

Copy sketch entities or blocks to retain all dimensions and relations

See Copy Entities in the help.

## **Define Sketch Patterns**

#### **Linear Sketch Patterns**

- Select an edge from a part or an assembly to define Direction 1 for X-axis.
- Direction 2 for the Y-axis is active as soon as you select Direction 1.



If you do not select **Direction 1** first, you must manually select **Direction 2** to activate it.

## **Circular Sketch Patterns**

Select an edge from a part or an assembly to set the pattern direction.



## **Fully Define Sketch**

With the **Fully Define Sketch** is tool, the SolidWorks application calculates which dimensions and relations are required to fully define an under defined sketch. You can access **Fully Define Sketch** at any point with any combination of dimensions and relations already added.

#### To fully define a sketch;

1 In Edit Sketch mode, click Fully Define Sketch 🕑 (Dimensions/Relations toolbar) or Tools, Relations, Fully Define Sketch.

The **Fully Define Sketch** PropertyManager is displayed.

- 2 Under Entities to Fully Define select either:
  - All entities in sketch
  - Selected entities
- 3 Click **Relations** and choose **Select all**, **Deselect all**, or chose individual relations.



In some sketches only certain relations and dimensions can fully define the sketch. Limiting your selection may prevent the sketch from being fully defined.

- 4 Click **Dimensions** and select a dimension scheme and dimension placement.
- 5 Click Calculate under Entities to Fully Define, then click 🖌.

See Fully Define Sketch in the help.

## **Relations with Midpoints**

Enhancements to the midpoint relation in 2D sketches allow you to:

- Add coincident relations between the midpoint of an arc and another point.
- Add midpoint, horizontal or vertical relations between the midpoint of an edge and a point.



The point can be a sketch point, the center of a circle, or a point on a line.

• Use the shortcut menu and choose **Select Midpoint** for any sketch entity while the **Add Relations** PropertyManager is displayed.

## **Sketch Orientation Behavior**

Sketch orientation behavior is more consistent with the **Normal To** (Standard Views toolbar) tool. Using the **Normal To** command with sketches created on a plane or on a model face produces uniform orientation results. The amount of twist required for models to display normal to a selected plane or face is also minimized.

## **Sketching Options**

#### Anti-Alias

You can display sketch entities in anti-alias mode. Click **Options**, **System Options**, **Display/Selection**, and select **Anti-alias edges/sketches**.

#### Undo

You can undo sketch actions after you exit the sketch. Return to **Edit Sketch** mode and click **Undo** (Standard toolbar).

## **SketchXpert**

**SketchXpert**, formerly **Resolve Conflicts**, includes changes to the PropertyManager and visual solutions for over defined sketches. Color codes represent the sketch states:

- Yellow indicates that a relation or dimension is valid but in conflict.
- Red indicates that no solution can be found for the relation or dimension.



Fully defined sketch



Over defined sketch (yellow) with a dimension that has no valid solution (red)

Click **Over Defined** <u>A</u> on the status bar to diagnose or manually repair the sketch.

#### Diagnose

The **Diagnose** solution generates a number of potential solutions under **Results**. You can cycle through potential solutions using  $\leq$  and  $\geq$  under **Showing solution**.



Solution 3. Delete second 100 dimension

Solution 2. Delete equal dimension

The solution results appear:

- In the graphics area where the sketch is updated
- Under More Information/Options where the sketch entity is listed

#### **Manual Repair**

**Manual Repair** generates a list of dimensions and relations in the sketch. To repair the sketch, select one or more relations in **Conflicting Relations/Dimensions** and press **Delete** or **Suppress**.

See SketchXpert in the help.

## **Splines**

Enhancements to splines include:

- Correct drag behavior at the unconstrained ends of a spline, when dragging the control polygon.
- Options in the **Spline** and **Curvature Scale** PropertyManagers.
- Individual control for spline weight and direction.
- New polygon control handles.

## Curvature

In the **Spline** PropertyManager, under **Options**, you can select **Show Curvature** to display curvature combs.

In the **Curvature Scale** Property Manager, the **Density** scale adjusts the number of curvature combs you can display.

#### **Two-Point Splines**

With two-point splines that include curvature handles at both ends, under **Options**, you can raise and lower the degree of the spline.

#### Weight and Direction Handles

Spline handles have individual controls for weight and direction (vector).

Adjust both Tangent Weighting and Tangent Radial Direction (vector).



Graphic Control	Function	Result
	Drag either circle handle to control both tangency weight and direction (vector) asymmetrically.	
	Press <b>Alt</b> and drag either circle handle to control both tangency weight and direction (vector) symmetrically.	
	Drag either arrow head handle to control tangency weight <i>asymmetrically</i> .	
	Press <b>Alt</b> and drag either arrow head handle to control tangency weight symmetrically.	
	Drag either diamond handle to control tangency direction (vector). Tangency is applied symmetrically to the spline point.	- AC

#### Activation

Select the spline to display all non tangent driving handles.



#### Display

Select a spline point to display the handle.



#### Spline PropertyManager

When you press **Alt** and drag either arrow head handle, the weight is displayed for both **Tangent Weighting 1** is and **Tangent Weighting 2** inder **Parameters**. This indicates a symmetrical adjustment to the spline point. Although the weight of **Tangent Weighting 1** is and **Tangent Weighting 2** is does not have to be equal, each is being adjusted symmetrically.

## **Control Polygons**

New control polygon handles facilitate modifying the spline. Drag a control polygon to change the display from points to triangles. After you drag, a node is added to the polygon. The tangency arrow that is displayed matches the length or the segment of the polygon it applies to.



See Splines in the help.
3

# **Parts and Features**

This chapter describes enhancements to parts and features in the following areas:

- General
- Boundary Surfaces
- Feature, Fillet, and Draft Xperts
- Freeforms
- Hole Series
- Surface Fills

## General

#### SelectionManager

The SelectionManager, available in loft, sweep, and boundary surface features only, combines and replaces contour and smart selection, while offering enhanced selection capability.

- You can select edges and sketch entities, which was not possible in smart selection.
- You can select entities across multiple sketches as well as in combination with model edges.
- Open selection sets are now trimmable and extendable at both ends, regardless of how you create them.
- Parametric trim points snap to geometry, so if you modify the geometry, the trim is modified.

#### To use the SelectionManager:

- 1 Open Features\SelectionManager.sldprt.
- 2 Click Lofted Boss/Base 4 (Features toolbar) or Insert, Boss/Base, Loft.
- 3 Select the sketch line shown.

The SelectionManager appears. The **Select Group** tool is active.





4 In the SelectionManager, click the pushpin so the SelectionManager remains available.



You can also right-click in the graphics area and click **SelectionManager** to activate it.

5 Select the three connecting sketch entities, then click  $\checkmark$ .

**Closed Group**<1> appears under **Profiles**  $p^0$ .



- 6 In the SelectionManager:
  - a) Click Select Closed Loop 

    .
  - b) Select Auto-OK selections.
- 7 Select the two loops one-by-one in approximately the same area to add them to Profiles <sup>0</sup>/<sub>0</sub> in the PropertyManager as Closed Loop<1> and <2>.
- 8 In the PropertyManager, click in Guide Curves 💅.
- 9 Click **Select Group** and select the first section of the guide curve shown.

The callout appears indicating there is a tangent sketch segment.

10 Click the callout to select the tangent sketch segment automatically.

The callout changes color indicating the tangent sketch segment is selected.

11 Select the next segment on this guide curve, then click the callout to select the next tangent sketch segment. Click ✔.



12 Click Select Open Loop , then select the three remaining guide curves.
 One Open Group and three Open Loop selections are listed under Direction 2.

13 Click 🕢 in the PropertyManager.



See SelectionManager in the help.

#### **Selection Through Faces**

When creating or editing fillets and chamfers, click the **Select through faces** option in the PropertyManager to enable selection of edges through faces that hide the edges.



#### **Multibody Parts**

In multibody parts, the following items are configurable for each body:

- Hide/Show
- Color
- Texture

#### **Boundary Surfaces**

The boundary surface feature allows the creation of surfaces that can be tangent or curvature continuous in both directions (all sides of the surface). In most cases, this delivers a higher quality result than the loft tool.

#### To create a boundary surface feature:

- 1 Open Features\BoundarySurface.sldprt.
- 2 Click Boundary Surface (Surface toolbar) or Insert, Surface, Boundary Surface.
- 3 Under Direction 1:
  - a) Select the edge shown for Curves.

Edge <1> is listed under Direction 1.



b) Select the other edge shown.

**Edge<2>** is listed under **Direction 1**. A preview appears.



Select the edges at approximately the same point to prevent twisting. You can also right-click in the graphics area and select **Flip Connectors**.



c) Select Edge<1> and select Curvature to Face in Tangent Type as a constraint. Repeat for Edge<2>.

The edge callouts update to show the **Curvature** To Face curvature constraint. The boundary surface is now curvature continuous along both edges in **Direction 1**.

#### 4 Under **Direction 2**:

a) Select one of the 3D sketch curves for Curves.

The SelectionManager appears with the Select Open Loop  $\rightarrow$  tool active.

- b) Click the pushpin so the SelectionManager remains available.
- c) Click 🖌.

**Open Loop<1>** is listed under **Direction 2**.

- d) Select the other 3D sketch curve and click V.
- 5 Click 🕢 in the PropertyManager.



Direction 1 and Direction 2 are fully interchangeable. You could have selected the 3D sketch curves for **Direction 1** and the edges for Direction 2 and had the same results.

Click the

See Boundary Surface in the help.



## Feature, Fillet, and Draft Xperts

The FeatureXpert, powered by SolidWorks Intelligent Feature Technology (SWIFT<sup>™</sup>), manages fillet and draft features for you so you can concentrate on your design. The FeatureXpert is one of three interrelated Xperts:

- **FeatureXpert**. Works behind the scenes to resolve feature errors in constant radius fillets or neutral plane drafts.
- **FilletXpert**. Manages the creation and modification of all constant radius fillets.
- **DraftXpert**. Manages the creation and modification of all neutral plane drafts.



Use the FilletXpert and DraftXpert when you want the SolidWorks software to manage the structure of the underlying features. Use the manual **Fillet** and **Draft** PropertyManagers when you want to maintain control at the feature level.

## FeatureXpert

When you add or make changes to constant radius fillets and neutral plane drafts that cause rebuild errors, the FeatureXpert automatically fixes the errors. The FeatureXpert can change the feature order in the FeatureManager design tree or adjust the tangent properties so a part successfully rebuilds.

#### To add features using the FeatureXpert:

- 1 Open Features\Xperts\FeatureXpert01.sldprt.
- 2 Click Fillet 🙆 (Features toolbar) or Insert, Features, Fillet/Round.
- 3 In the PropertyManager, click Manual.
- 4 Under Items to Fillet:
  - a) Set Radius 🏹 to .25.
  - b) Select the two red faces for Edges, Faces, Features and Loops ?



If you have PhotoWorks active, make sure that **Display PhotoWorks Materials in SolidWorks (PhotoWorks, System Options)** is cleared so you see the model colors.



5 Click 🕢.

The **What's Wrong** dialog appears. The fillet error is highlighted in the dialog, which indicates you can try to repair the error using the FeatureXpert.

#### 6 Click FeatureXpert.

The FeatureXpert repairs the error by creating multiple fillets instead of one fillet that contains both faces. See the FeatureManager design tree.

#### To change features using the FeatureXpert:

1 Open Features\Xperts\FeatureXpert02.sldprt.

The entire part is filleted with a .1 inch radius fillet.

- 2 Double-click the **Fillet1** feature in the FeatureManager design tree to display the fillet dimension in the graphics area.
- 3 Double-click the dimension.
- 4 In the Modify dialog, change the dimension to .25, then click ✓
- 5 Click **Rebuild** 🚦 (Standard toolbar) or **Edit**, **Rebuild**.

The **What's Wrong** dialog appears. The fillet error is highlighted in the dialog.

6 Click FeatureXpert.

The FeatureXpert creates individual fillets as necessary, and places them in the proper order in the FeatureManager design tree to allow the model to solve.

See FeatureXpert in the help.

#### FilletXpert

The FilletXpert manages, organizes, and reorders fillets for you so you can concentrate on your design intent. The FilletXpert can:

- Create multiple fillets
- Automatically invoke the FeatureXpert
- Automatically reorder fillets when required

#### To create multiple fillets using the FilletXpert:

- 1 Open Features\Xperts\FilletXpert.sldprt.
- 2 Click Fillet 🙆
- 3 In the PropertyManager, click FilletXpert.
- 4 Select the cyan-colored cylindrical face, set the Radius → to .25, then click Apply.
- 5 Select the magenta-colored planar face and click **Apply**.

The FilletXpert added two fillet features without leaving the PropertyManager.





#### To automatically invoke the FeatureXpert while using the FilletXpert:

The FilletXpert automatically calls the FeatureXpert when it has trouble placing a fillet on the specified geometry.

1 Select the two brown-colored faces and click **Apply**.

The FilletXpert automatically calls the FeatureXpert to create the fillet. The FeatureXpert knows that it must create multiple fillets, and does so.

- 2 Click Hidden Lines Visible (View toolbar) or View, Display, Hidden Lines Visible.
- 3 Box select the entire model.

Box selection captures all the edges.

4 Click **Apply**.

The FilletXpert fillets the entire model with .25 inch radius fillets, automatically invoking the FeatureXpert to resolve the fillets.

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

## To automatically change or remove fillets using the FilletXpert:

- 1 Select the **Change** tab.
- 2 Hover over **Fillet1** in the flyout FeatureManager design tree.

Note that **Fillet1** is applied to multiple edges.

- 3 In the graphics area, select the lower filleted edge.
- 4 Set Radius 剂 to 1.0, then click Resize.

The FilletXpert resizes only the single selected edge by creating a new fillet for it. The fillets are listed by size under **Existing Fillets**.







- 5 Hover over Fillet2 in the FeatureManager design tree.Note that Fillet2 is applied to multiple edges.
- 6 Select the circular filleted edge.
- 7 Click **Remove**.

The FilletXpert removes the fillet from only the circular edge.



Click the image for 3D viewing



See FilletXpert in the help.

#### DraftXpert

The DraftXpert takes trial and error out of the draft process. You select the draft angle and references to draft, and the DraftXpert manages the rest.

The DraftXpert can:

- Create multiple drafts
- Do draft analysis
- Edit drafts

To create multiple drafts and do draft analysis using the DraftXpert:

- 1 Open Features\Xperts\DraftXpert.sldprt.
- 2 Click **Draft [Sectores** (Features toolbar) or click **Insert**, **Features**, **Draft**.

- 3 In the PropertyManager, click **DraftXpert**.
- 4 Under Items to Draft:
  - a) Set Draft Angle 📉 to 3.00deg.
  - b) Select the red-colored planar face on top of the cylinder for Neutral Plane. Make sure the Direction of Pull arrow points upward.

  - d) Click **Apply** to create the draft.

#### 5 Under **Draft Analysis**, select **Auto paint**.

The face you just drafted displays the **Draft Analysis** color for 3 degrees draft, while the inside of the cylinder is yellow, indicating no draft.

6 Move the pointer over the drafted face. Pointer feedback reports the draft angle of **3deg**.

- 7 Clear Auto paint.
- 8 Under **Items to Draft**, select the redcolored face on the square front of the model for **Neutral Plane**.

Make sure the **Direction of Pull** arrow points outward.

9 Select Auto paint.

The circular inside of the horizontal cylinder is yellow, indicating no draft.

- 10 Select the circular inside for Items to Draft 🕎.
- 11 Click **Apply** to create the draft.

The color of the circular inside updates to indicate **3deg** of draft. Pointer feedback verifies the draft angle.











#### To change draft using the DraftXpert:

- 1 Clear Auto paint.
- 2 Under Items to Draft, select the red-colored face shown for Neutral Plane. Make sure the Direction Of Pull arrow points upward.
- 3 Select the four orange-colored faces (two faces on each side of the horizontal cylinder) for Items to Draft ①.

Expand the FeatureManager design tree and note the order of the fillets.

4 Click Apply.

The DraftXpert creates the drafts and automatically invokes the FeatureXpert to resolve the model.

The FeatureXpert reorders the new draft features in the FeatureManager design tree. Compare the order of the fillets.

5 Select Auto paint.

The **Draft Analysis** colors indicate **3deg** draft on the four side faces.

- 6 Clear Auto paint.
- 7 Select the **Change** tab.
- 8 Under **Drafts to Change**, select the two orangecolored outer faces (one each on opposite sides of the horizontal cylinder) for **Faces to Draft (**.
- 9 Select Auto paint.
- 10 Set Draft Angle to 10.00deg and click Change.

Under **Draft Analysis**, the color range updates to

show **10.00** as the highest positive draft in the model. The color of the two faces you just drafted use the proper color.

11 Click 🕑

See DraftXpert in the help.

Click the image for 3D viewing









## Freeforms

The freeform feature deforms faces of surface or solid bodies. You can deform only one face at a time and the face must have four sides only. Designers have direct, interactive control of deformations by creating control curves and control points, then pushing and pulling the control points to deform the face. Use the triad to constrain the push or pull direction.

Freeform gives you more direct control compared to deform features. Freeform meets the needs of consumer product designers who create curvilinear designs.



Freeform features do not affect model topology because they do not create additional faces.

To create a freeform feature, you first add control curves to a face, then add control points to the control curves, then move the control points to deform the face.

#### To add control curves:

#### 1 Open Features\Freeform.sldprt.

Using freeform, you are going to create a grip handle matching the reference sketch curve, which is based on the sketch picture.



Reference sketch

- 2 Click **Shaded with Edges** (View toolbar) and select **IsoBottom** from the **View Orientation** list.
- 3 Click Freeform **b** (Features toolbar) or Insert, Features, Freeform.
- 4 In the PropertyManager, under Face Settings:
  - a) Select the face in the graphics area for **Face to deform**  $\Box$ .
  - b) Click Direction 1 Symmetry to use symmetrical mode, which creates the freeform symmetrically across the plane that appears.
- 5 Click the four **Continuity** callouts to set the boundary continuity to **Tangent**.



#### 6 Under Control Curves:

- a) Click **Through points** to use points placed on the control curve to deform the face.
- b) Click **Add Curves**, then move the pointer  $\searrow$  over the face until you reach the symmetry plane.



The **Add Curves** and **Add Points** buttons are modal. In add curve mode, you can add control curves only. Then click **Add Points** to enter point mode to add control points along control curves.

After adding control points, you accept them, automatically exit both modes, and drag the control points to deform the face.

A preview of the control curve appears on the face. When you reach the symmetry plane, it highlights in red.



If the control curve preview is not running length-wise, click **Flip Direction (Tab)**.

7 Click to snap the control curve to the plane, then right-click to accept the curve. Move the pointer over the curve to highlight it and verify its placement.

## To add control points to the control curve:

- 1 Click **Right** (Standard Views toolbar).
- 2 In the PropertyManager, under **Display**, drag the **Face transparency** slider so you can see the reference sketch and sketch picture.
- 3 Under Control Points, click:
  - Snap to geometry (Control points will snap to geometry when moved)
  - **Curve** (The X axis of the triad orients itself tangent to the control curve)
  - **Triad follows selection** (Constrains the triad to the selected control point)



4 Click **Add Points**, then click the control curve to place seven control points total approximately as shown.



5 Right-click to accept the control points.

Try to match the position of the control points to points on the reference sketch.



Use the grid lines to help match the points.

#### To move control points to create the freeform feature:

1 Drag the individual points to match the reference sketch curve and create the freeform feature. Use the triad arrows to re-position points along the curve.

The pointer changes to  $\mathbb{R}^{+}$  as you drag the points. The triad sphere highlights and snaps to reference points. Your model should resemble the image below.



2 Click 🖌



3 Hide the sketches and origin, then rotate the model to view the freeform feature on the bottom.



Io refine the grip, you could add another freeform feature with
 control curves over the three humps. You would then place
 control points on these control curves to define your design
 intent.

See Freeform in the help.

## **Hole Series**

The hole series feature has been enhanced:

- You can use an existing hole as the seed hole. Select **Use existing holes** in the **Hole Position** PropertyManager.
- You can specify the end of the hole series under **End Component** in the **Hole Series (Last Part)** PropertyManager.
- You have two new end conditions:
  - Up to Surface
  - Offset from Surface (Available only with Tap as the end hole)
- If you add new components between the start and end components after you create the hole series, you can choose to include the new components in the hole series. You must edit the hole series to update it.
- The depth of each hole within each component is measured from the entry face of the component to the end of the hole in the component. The depth displayed represents the actual depth of the hole in each component, resulting in accurate production drawings.



#### To create a hole series:

1 Open Features\HoleSeries\HoleSeries.sldasm.

The model has a counterbore hole in the **Top** component.

- 2 Click Hole Series 🙀 (Features toolbar) or Insert, Assembly Feature, Hole, Hole Series.
- 3 In the PropertyManager, select Use existing hole(s), then select the counterbore hole for Hole wizard feature/pattern 🗟.

Select the Hole Wizard hole 📸 from the **Top** component in the FeatureManager design tree.

- 4 Click () twice to accept the middle parts specifications.
- 5 In the PropertyManager, select Through All in End Condition.
- 6 Click 🕢.

The bottom component is included in the hole series.

7 In the FeatureManager design tree, right-click Middle<1> and select Set to Resolved.

The middle component appears but is not included in the hole series.

#### To include the middle component in the hole series:

1 In the FeatureManager design tree, right-click the hole series feature 🙀 and select Edit Feature.

Use existing hole(s) and the counterbore hole are selected.

- 2 Click ()
- 3 In the **Hole Series (Middle Parts)** PropertyManager, select and expand Solution New Component(s). Select the Middle component from the list.
- 4 Click ().
- 5 In the PropertyManager, make sure **Through All** is selected in End Condition, then click 📿.

The model updates to add the Middle component to the hole series.

See Hole Series in the help.













## **Surface Fills**

Surface fills have been enhanced:

② See Filled Surface in the help.

- Curvature continuity is supported at boundaries.
- Composite curves are supported. Previously only edges and sketches were supported.
- You can enable the **Fix up boundary** option in the PropertyManager to construct a valid boundary by automatically building missing pieces or trimming pieces that are too big.



Surface fill (red) is created despite the gap in the boundary. The gap between the blue and orange surfaces is bridged by extending the curve internally to the orange surface.



Surface fill (red) is created despite the green boundary being larger than the sides of the fill patch. To create the surface fill, the edges are split internally.



# Assemblies

This chapter describes enhancements to assemblies in the following areas:

- Belt/Chain
- Display and Selection
- Mates
- Smart Components
- General Enhancements

## **Belt/Chain**

Use the **Belt/Chain** assembly feature to model systems of belts and pulleys or chains and sprockets. This feature creates:

- Belt mates to constrain the relative rotation of the pulley components.
- A sketch containing a closed chain of arcs and lines describing the path of the belt.

The software calculates the length of the belt based on the positions of the pulleys. Optionally, specify the length of the belt and have the pulley positions adjust (at least one pulley must have an appropriate degree of freedom).

You can select to automatically create a new part containing the belt sketch and add the part to the assembly. In the part file, use the sketch as a sweep path to create a solid belt.

#### To create a belt feature:

- 1 Open Assemblies\pulleys.sldasm.
- 2 Click Belt/Chain 🐼 (Assembly toolbar) or Insert, Assembly Feature, Belt/Chain.
- 3 In the PropertyManager, under **Belt Members**, for **Pulley components**, select the outside cylindrical face of each pulley in the order shown.







4 Click the feature handle on the last pulley.



The belt flips to the other side of the pulley.



- 5 Under Properties:
  - a) Select Driving.
  - b) Type **32** for **Belt Length**.
  - c) Select Engage belt.
- 6 Click 🕢.

The top pulley moves to accommodate the specified belt length. In the FeatureManager design tree, the following appear:

- A Belt 淤 feature.
- A BeltMates 
   Solution for the second second

To see the belt curve when the PropertyManager is closed, select the **Belt** 🗞 feature in the FeatureManager design tree.

7 Drag the large pulley to rotate it.

All the pulleys rotate.



If you want to reposition a pulley without causing the other pulleys to rotate, edit the **Belt** 🗞 feature and clear **Engage belt**.

See Belt/Chain Assembly Feature in the help.

## **Display and Selection**

Commands have been added to facilitate editing large, complex assemblies.

#### Isolate

The **Isolate** command sets all components *except* the selected ones to be hidden, transparent, or wireframe, enabling you to focus on the selected components. Before you exit **Isolate**, you can save the display characteristics to a new display state. Otherwise, the display returns to its original state without any permanent changes.

#### To isolate components:

- 1 Open Assemblies \Power Supply Assembly.sldasm.
- 2 Select Voltage Switch and AC Connector.



#### 3 Click View, Display, Isolate.

All components except the selected ones are hidden. The isolate pop-up toolbar appears.



4 Select **Transparent** [Select Transparent] (Isolate pop-up toolbar).



5 Click Exit Isolate.

The model returns to its original display state.

#### **Cross Select**

When you box select from right to left (cross select), all components whose visible geometry intersects the selection box are selected.



## Mates

#### **Rack & Pinion Mates**

With rack and pinion mates, linear translation of one part (the rack) causes circular rotation in another part (the pinion), and vice versa.

#### To add a rack and pinion mate:

1 Open Assemblies\press.sldasm.

(The frame01 component is hidden to facilitate selection of mate entities.)



- 2 Click Mate 🔊 (Assembly toolbar) or Insert, Mate.
- 3 Under Advanced Mates, click Rack Pinion 🚇.
- 4 Under Mate Selections:
  - a) For Rack, select the linear edge of rack01.
  - b) For **Pinion/Gear**, select a circular edge of **spurgear**.



- 5 Under Advanced Mates:
  - a) Select Pinion pitch diameter.
  - b) Clear Reverse.
- 6 Click 🕢 twice to add the mate and close the PropertyManager.
- 7 Drag handle01 to see the rack move.

See Rack and Pinion Mates in the help.

#### **Multiple Mate Mode**

To mate multiple components to a common reference in a single operation, click **Multiple mate mode** in the **Mate** PropertyManager. If you select **Create multi-mate folder**, the resulting mates are placed in a **Multi-Mates** folder. You can change the common reference, mate type, or distance for all the mates in the folder in a single operation. In the example below, four components are mated concentrically to a common reference (the shaft).



#### **Gear Ratios**

For gear mates, the software assigns gear ratios based on the relative sizes of the cylindrical faces or circular edges you select. The values are parametric. You can override the automatic values.

#### Mate to Curve

You can create the following types of mates to single entity curves:

Mate Type	Reference 1	Reference 2
Coincident	point	curve
Distance	point	curve

This functionality can be used in SolidWorks Animator to animate a component's position along a curve. See **Mate to Curve** on page 9-34.

#### MateXpert

**MateXpert**, powered by SWIFT technology, replaces **Mate Diagnostics**. Enhancements help you to avoid, diagnose, and fix mate errors.

#### **View Mates**

- You can pin the **View Mates** PropertyManager to keep it visible.
- When multiple components are selected, the mates they have in common are sorted to the top of the list.
- In the graphics area, components involved in the mate system for the selected components are slightly transparent. Components not involved are hidden.
- Callouts appear in the graphics area to help you visualize mate systems. Each mate has one callout, with leaders pointing to the two mated entities. In the callout, you can perform several common mate functions, such as flipping the alignment or suppressing the mate. When you select a callout, the mate geometry highlights, and components not involved in the selected mate become more transparent. Error and warning icons are displayed in the callouts.



#### **Other Enhancements**

- During editing, the software changes the alignment of related mates automatically if necessary to prevent introducing mate errors.
- You can force a mate into position in spite of errors to help identify other mates that are causing the conflict.
- The mate status is reported in the status bar. You can click the status to run **View Mates** when mate warnings or errors exist.

In the following example, add a distance mate between two components, which over defines the assembly. Then use mate callouts to diagnose and correct the mate error.

#### To use mate callouts:

- 1 Open Assemblies\tableassembly.sldasm.
- 2 Click Mate (Assembly toolbar), or Insert, Mate.
- 3 Select the faces as shown.

A message points to the PropertyManager.

- 4 Click **Distance** 💦.
- 5 Set the distance to **10** and click *(*.

A dialog box asks if you want to force the mate to solve.



6 Click Yes.

The scale moves to the position defined in the distance mate. **Solution Found** appears in the status bar. The **What's Wrong** dialog box appears. **Coincident9** is flagged with **Serror**.

- 7 Close the What's Wrong dialog box.
- 8 Click 🕢 to close the PropertyManager.

Error and warning flags appear in the **Mates M** folder.

9 Click **ONO Solution Found** in the status bar.

The **View Mate Errors** PropertyManager appears at the bottom of the left panel. A mate callout for **Coincident9** appears in the graphics area. The callout is red, indicating that the mate is not satisfied.

10 In the callout for **Coincident9**, click **Suppress** 🔊.

The mate errors and warnings disappear, because the assembly is no longer over defined.

11 Click an empty region of the graphics area to close the **View Mate Errors** PropertyManager.

## **Smart Components**

### **Selective Insertion**

When activating the Smart Feature of a Smart Component, you can select which individual associated features and components to insert. Previously, you had to select all or none of the features and all or none of the components.

## **Hide Components**

When making a component smart, you can click **Hide Components** under **Auto Size** to make it easier to select the concentric mate reference.

## **General Enhancements**

### **Component "in-use" Configuration Option**

In the **Component Properties** dialog box, the option to set the referenced configuration to **Use component's "in-use" or last saved configuration** has been eliminated. For legacy data, if this option was specified for a component, that component will be set to use the named configuration that corresponds to the "in-use" or last saved configuration.

## **Cosmetic Threads**

Tapped holes created as assembly features can display cosmetic threads across multiple parts or bodies as dictated by the thread depth. Previously, the thread was displayed only on the first part or body regardless of the depth of the thread.

#### **Flexible Sub-Assemblies**

The positions of nested flexible sub-assemblies are now maintained across sessions.

## **Open Part**

Now when you open a part from within an assembly, the part opens in the configuration referenced by the assembly. Previously, it opened in the last saved configuration.



# 5

# Drawings

This chapter describes enhancements to drawings in the following areas:

- Broken views
- Deleting views
- Drawing sheets
- Relative views
- Section views
- View Palette

## **Broken Views**

You can create multiple horizontal and vertical breaks in the same drawing view. When you insert the break lines, the breaks are immediately applied. Additionally, you specify horizontal or vertical breaks and the type of break lines in the PropertyManager.

In **Tools**, **Options**, **Document Properties**, **Dimensions**, the option **Show dimensions as broken in broken views** displays dimension lines with a zig-zag.



In views with multiple breaks, the break line style (straight cut, curve cut, etc.) must be the same.



See Broken Views in the help.

## **Deleting Views**

When you delete a detail or section view, the **Confirm Delete** dialog box has an option to **Delete detail sketch** or **Delete section sketch** so the sketch used to create the view is also deleted. This same dialog box appears if you delete a sketch used for a detail or section view.

## **Drawing Sheets**

## Navigating

In drawings with multiple sheets that span beyond the length of the screen, use the navigation buttons at the bottom of the drawing window to jump to the first or last sheet.



If you hold down the next or previous sheet buttons, the tabs scroll.

## Renaming

You can rename a drawing sheet by right-clicking a tab at the bottom of the window and selecting **Rename**.

#### Reordering

When you reorder drawing sheets with the tabs at the bottom of the window, the pointer changes to  $\frac{1}{2}$  (single sheet) or  $\frac{1}{2}$  (multiple sheets).

## **Relative Views**

You can select reference planes to specify the view orientation of relative views. This is helpful when you create relative views of cylindrical structural members. You can select reference planes for an entire part or for selected bodies.



See Relative to Model View in the help.

## **Section Views**

When you create a section view in a part or assembly, you can save the section view to include in the **View Palette**.

#### To save a section view to include in the View Palette:

- 1 Open Drawings\BodyAssembly.sldasm.
- 2 Click Section View 🙀 (View toolbar), or View, Display, Section View.



You can also right-click a plane in the FeatureManager design tree and select **Section View**.

- 3 In the PropertyManager, under **Section 1**, click **Right /**.
- 4 Click Save.
- 5 In the **Save As** dialog box, select **Drawing annotation view**, then click **Save**.



View orientation replaces the former **Save View** option. Cancel replaces the former **Reset** option.

- 6 For **Excluded components**, select the following to exclude from the section cut:
  - headcover
  - snapring <1> and <2>
  - collar
- 7 Click OK.
- 8 Click Make Drawing from Part/ Assembly 🗐, or File, Make Drawing from Assembly.





SolidWorks 2007 What's New

- 9 Select a sheet size, then click **OK**.
- 10 In the **View Palette** (located in the Task Pane), drag the following views into the graphics area:
  - a) \*Front
  - b) Section View A-A

#### **View Palette**

Use the **View Palette**, located in the Task Pane, to insert drawing views. It contains images of standard views, annotation views, section views, and flat patterns (sheet metal parts) of the selected model. You can drag any of the views onto the drawing sheet to create a drawing view.

The View Palette populates when you:

- Click Make Drawing from Part/Assembly
- Browse to a document from the View Palette.
- Select from a list of open documents in the View Palette.



6

# Detailing

This chapter describes enhancements to detailing in the following areas:

- Annotation views
- Arrows
- Balloons
- Blocks
- Center marks
- Datum feature symbols
- Dimensioning standards
- Dimensions
- DimXpert
- Leaders
- Moving and copying annotations
- Notes
- Tables
- Text alignment
- Undo
- View labels

## **Annotation Views**

You can change the horizontal orientation of annotation views. You might want to change the orientation because:

- The default view orientation is not acceptable for a drawing view.
- An imported part was created in the context of an assembly, and the part does not align with standard views.

In the FeatureManager design tree, right-click an annotation view and select **Edit Annotation View**, then adjust the following under **Horizontal Direction**:

- **Direction Reference**. Select a sketch, edge, or face to change the horizontal direction.
- Invert horizontal direction. Flips the annotation view 180°.
- Manual controls. Type a value or move the slider to change the annotation view.





Original annotation view from imported model

Annotation view after changing the horizontal direction

See Move to Annotation View in the help.

#### Arrows

You can set the arrow head size for dimensions and section views to scale automatically to the dimension height or section view letter height, if the arrows do not follow the document font. In **Tools, Options, Document Properties, Arrows**, select **Scale with dimension height** or **Scale with section view arrow letter height**.

See Arrows Options in the help.
#### **Balloons**

Using custom properties in balloons is streamlined for ease of use. You can define the custom properties individually in the PropertyManager or for the entire document in **Tools, Options, Document Properties, Balloons**.

#### To use custom properties in balloons:

- 1 Open assemblies\pulleys.slddrw.
- 2 Click **Options** [E] (Standard toolbar), or **Tools**, **Options**.
- 3 On the **Document Properties** tab, select **Balloons**.
- 4 Under Single balloon, select:
  - a) Circular Split Line in Style.
  - b) Tight Fit in Size.
- 5 Under Balloon text, select:
  - a) Custom Properties in Upper.
  - b) **Description** for the custom property.
  - c) Text in Lower.
- 6 Click OK.
- 7 Click Balloon (Annotation toolbar), or Insert, Annotations, Balloon.

The options you set are preselected in the PropertyManager.

- 8 In the PropertyManager, under Balloon Settings, type revised under Custom Text.
- 9 Click to place the balloon in the graphics area as shown.

10 Click 🖌



## Blocks

In drawings, you can create blocks from sketch entities, just as you create blocks in parts and assemblies. Previously you created blocks in drawings by using block definitions and block instances. Now you create blocks from sketches where the blocks can be:

- Created and edited in the drawing, not in a separate window.
- Nested (blocks within blocks).
- Copied between drawings and sketches.

Some differences separate the drawing block functionality from blocks in parts and assemblies. Blocks in drawings:

- Have dimensions displayed by default. You can control dimension scaling in the **Block** PropertyManager so dimensions scale with the block.
- Are displayed in the FeatureManager design tree in the **Blocks** in folder with one line per block (not per instance). In parts and assemblies, each block instance is listed.

When a block is on a layer, the layer's display attributes are applied to the entire block, regardless of the layer attributes of any entities in the block that are on a different layer. This does not apply to blocks created in previous releases.

See Blocks in the help.

#### **Center Marks**

In **Tools**, **Options**, **Document Properties**, **Detailing**, you can specify **Center marks - holes** or **Center marks - fillets** so that center marks are inserted based on the feature type.

See Detailing Options in the help.

## **Datum Feature Symbols**

Datum feature symbols are displayed to support various international standards. You can drag a datum feature symbol onto a geometric tolerance symbol or onto a dimension, or you can pre-select the entity to which to attach the datum feature symbol.





You cannot add datum feature symbols to ordinate dimensions, chamfer dimensions, angle dimensions (unless the symbols are displayed in the 1982 standard), or arc lengths.

See Datum Feature Symbols in the help.

## **Dimensioning Standards**

The following enhancements support the GB standard.

#### **Detail View Letters**

Detail view letters appear above the end of the leader.



#### **Detail View Labels**

Detail views omit the words DETAIL and SCALE. A line separates the label and scale.



SolidWorks 2006



## **Hole Callouts**

Hole callouts are displayed with text above and below the hole callout line.



# Dimensions

## Dragging

When you drag a dimension, the text remains visible.

## **Extension Lines**

When you dimension to a circle containing a center mark, but use the circle instead of the center mark as the reference, a gap exists between the center mark and the extension line.



## **Fractional Dimensions**

Fractional dimensions created with the **Dimension** tool are displayed in a stacked orientation. You set the orientation (horizontal stack, diagonal stack, etc.) and the stack size percentage in **Tools**, **Options**, **Document Properties**, **Dimensions** under **Fractional display**.

For stacked fractional dimensions, set the following in **Tools**, **Options**,

- Document Properties, Units:
  - Set Length units to feet & inches.
  - Select Fractions.
    Select Round to nearest fraction.



See Dimensions Options in the help.

#### Locking

The pointer  $\mathbb{R}_{\mathbf{R}}$  indicates that you can lock a dimension's orientation.

See Lock Dimensions in the help.

## Snapping

When you place radial, diameter, or chamfer dimensions, they snap at 15° increments. Linear dimensions snap at 0.40 from the offset, and 0.25 thereafter. You can specify the snap distances for linear dimensions in **Tools**, **Options**, **Document Properties**, **Dimensions** under **Offset distances**.



To override the snapping behavior, hold down **ALT** while dragging a dimension.

## Undo

If you make a change to a dimension (for example, if you insert a symbol into a dimension), you can undo the change.

## **DimXpert**

DimXpert applies dimensions in drawings so that manufacturing features (patterns, slots, pockets, etc.) are fully-defined. The DimXpert tool is accessible in the **Dimension** PropertyManager. You select a feature's edge to dimension, then DimXpert applies all associated dimensions in that drawing view for the feature.

#### To apply dimensions using DimXpert:

- 1 Open **Detailing\dimxpert.slddrw**.
- 2 Click Smart Dimension <a>[2] (Dimensions/Relations toolbar), or Tools, Dimensions, Smart.</a>
- 3 In the PropertyManager, under:
  - a) Dimension Assist Tools, click DimXpert 🖽.
  - b) **Pattern Scheme**, click **Polar dimensioning** 🕀.
  - c) Dimensioning Scheme, click Baseline 🖽.
- 4 In the graphics area, select the vertex and hole as shown.



The drawing is dimensioned based on the selected hole pattern.

5 Select the next hole as shown.



Again, the drawing is dimensioned based on the selected hole pattern.





6Χ Ø .13 THRU ALL -----



# Leaders

## **Dragging Leaders**

You can hold down **CTRL** and drag a leader from radial, diameter, fillet, and chamfer dimensions and hole callouts to create multiple leaders. The leaders are allowed on entities that are the same size as the original dimension or hole callout.

Placement of handles on arrowheads has changed. The square handle indicates that you can **CTRL**+drag to create multiple leaders. When you hold down **CTRL** and move the pointer over the square handle, the pointer changes to  $\searrow$  .



See Multiple Leaders in the help.

#### **Multi-Jog Leaders**

When you insert an annotation, you can create it with a multi-jog leader. All functionality remains identical to that of the **Multi-jog Leader** fool (Annotation toolbar). Click **Multi-jog Leader** from the PropertyManager to create the following annotations with a multi-jog leader:

- Notes
- Surface finish symbols
- Geometric tolerance symbols
- Balloons (accessed in the **Note** PropertyManager)
- Weld symbols (select Use multi-jog leader in the Properties dialog box)

To change an existing leader to a multi-jog leader, right-click the leader and select **Add Jog Point**.

See Multi-jog Leaders in the help.

## **Moving and Copying Annotations**

You can move and copy dowel pin symbols and area hatches or fills that are defined by sketch entities. You can use the commands **CTRL+C**, **CTRL+V**, **CTRL**+drag, **Move Entities**, or **Copy Entities**.

See Cutting, Copying, and Pasting in Sketches in the help.

#### Notes

When you insert a dimension into a note, any symbols or tolerances included in the dimension are included in the note. For SolidWorks documents from previous releases that you open in SolidWorks 2007, you can insert the symbols or tolerances from dimensions by selecting **Include prefix**, **suffix and tolerance of dimensions** in the **Note** PropertyManager.

#### Geardiameter: 1,50



SolidWorks 2006: Diameter symbol is omitted from note

Geardiameter: abla1.50



SolidWorks 2007: Diameter symbol is included in note

See Notes in the help.

## **Tables**

#### **General Enhancements**

#### Save as Microsoft Excel

When you save tables as Microsoft<sup>®</sup> Excel<sup>®</sup> files, the following is retained:

- GD&T symbols (SWGDT font only)
- Horizontal/vertical alignment
- Merged cells
- Wrapped text

See Annotation Tables in the help.

#### Selection

You can use the **CTRL** key to select multiple cells.

- Text values in cells (including custom property values)
- Text direction
- Font, font style, height, and effect *in cells* (not when applied to partial text)

#### Snapping

When you insert a table, the table snaps to the sheet format and other tables before you place the table. Previously you had to place the table first, then drag the table to have it snap to entities.

#### Templates

When you save a table as a template, column widths are also saved.

#### **General Tables**

When you type a note in a general table, you can link the note to a property.

#### **Hole Tables**

#### **Adding Holes**

To add holes to an existing hole table, right-click the hole table in the FeatureManager design tree and select **Edit Feature**. Previously you selected **Add Hole(s)** from the shortcut menu.

#### **Editing Datums**

To edit the datum origin of an existing hole table, right-click the hole table in the FeatureManager design tree and select **Edit Feature**. Previously you selected **Edit datum definition** from the shortcut menu.

#### **Far Side Holes**

If a model has holes on the opposite face from the one you select in the **Hole Table** PropertyManager, you can include the far side holes in the table by selecting them manually. Far side holes are labeled **FAR SIDE**.



TAG	SIZE	QTY
А	Ø3.30	4
В	Ø6 THRU ∟」Ø9⊽0.50	2
0	Ø3 THRU FAR SIDE	1
D	Ø3.40⊽4.30 ∐ Ø9⊽0.50	1
E	Ø3 FAR SIDE	1

#### **Hole Tolerances and Precision**

You can change the tolerances and precision of holes. When you select a cell in the **XLOC**, **YLOC**, or **SIZE** columns, you can change the tolerance and precision in the PropertyManagers.

See Hole Tables in the help.

#### **Locating Dimensions**

Hole tables include tolerances for the X and Y locations of holes. If tolerance values were not defined in the model, you can still add tolerances by selecting a cell in the **X LOC** or **Y LOC** column and setting the tolerance in the **Cell** PropertyManager.

See Table Cell PropertyManager in the help.

#### **Locating Precision**

You can change the locating precision for each hole on a per-cell basis. Previously you could control the entire table in **Tools**, **Options**, **Document Properties**, **Tables**, but you could not control individual cells.When you select a cell in the **X LOC** or **Y LOC** column, you can set the precision in the **Cell** PropertyManager.

See Table Cell PropertyManager in the help.

#### **Multiple Views**

You can select more than one drawing view to define a hole table.

#### To define a hole table with multiple drawing views:

- 1 Open **Detailing**\HoleTable.slddrw.
- 2 Click Hole Table 🐻 (Table toolbar), or Insert, Tables, Hole Table.
- 3 In the graphics area, select the origin and face in **Drawing View1** as shown.



- 4 In the PropertyManager, click **Next View**.
- 5 In the graphics area, select the:
  - a) Origin in **Drawing View2** as shown.
  - b) Three faces that contain the holes.
- 6 Click 📿, then place the table.

All holes from both views are inserted in the table.

See Hole Tables in the help.



#### **Revision Tables**

Origin - Drawing View2

There are new options for revision tables when you have multiple sheets in a drawing. You can set them at **Tools**, **Options**, **Document Properties**, **Tables**, under **Multiple sheet style**:

- See Sheet 1. The revision table on the first sheet is the active table. On all other drawing sheets, the revision table is labeled See Sheet 1. This functionality is the same as previous releases.
- **Linked**. A copy of the revision table from sheet 1 is created on all sheets, and all revision tables update as one. For example, if you add a revision on one sheet, the revision tables on all other sheets include the revision.
- **Independent**. The revision table on each sheet is independent of any other revision table in the drawing. Updates to a revision table are not reflected in tables on other sheets.

## Text Alignment

Text alignment has been simplified. When you drag text, only the text displays; the text box does not appear because text is now aligned based on the inference lines only. When you align two lines of text, inference lines extend from one line of text to the moving text.

The following in **Tools**, **Options**, **System Options**, **Drawings** have been removed and are always enabled:

- Detail item snapping when dragging corner
- Detail item snapping when dragging center

When you drag existing text to align it with other text, the pointer changes to one of the following:

- 🕅 🔥 🕺 🕹 🕺 😽 😽
- 🛛 🗟 🗛 Center alignment
- Right alignment

GENERAL NOTE:

BREAK ALL S⊯ARP EDGES.

Example of notes aligned to the center

While you create annotations, you can snap to existing notes, balloons, or dimensions in the same drawing view. This makes it easier to align annotations before placing them. Previously you had to place an annotation, then drag it again to align it with other entities.



## Undo

When you edit an existing annotation, you can undo your changes to:

- Datum feature symbols
- Datum target symbols
- Dimensions
- Geometric tolerance symbols

## **View Labels**

#### Positioning

A new option, **Display label above view**, is in **Tools**, **Options**, **Document Properties**, **View Labels**. This option places view labels for detail, auxiliary, and section views above the view, and it applies to new drawing views only.

See View Labels Options in the help.

#### Font

New options in **Tools**, **Options**, **Document Properties**, **Annotations Font** allow you to set the font for **Detail View Label** and **Section View Label**. Previously, these labels were controlled by the font used for notes.

See Annotations Font Options in the help.

- Hole callouts
- Surface finish symbols
- Weld symbols



# **Sheet Metal**

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

- Corner Trim
- Edge flanges on curved planar edges
- Lofted bends with bend lines
- Transform sketches from flattened to bent state

#### **Corner Trim**

A **Corner Trim** tool was added to the Sheet Metal toolbar. The **Corner Trim** tool is available only with flattened parts.



## **Edge Flanges on Curved Planar Edges**

You can add curved edge flanges to edge flanges created off a planar base.

# Click the image for 3D viewing Finder curved edge Click the image for 3D viewing Finder curved edge

Click the image for 3D viewing



#### Requirements

- Curved edges must border a planar face.
- Multiple edges must be tangent but can include linear and curved segments.

## Characteristics

• Specify No Relief or add relief using Relief Type in the PropertyManager.





Applies to edge flanges and miter flanges when you can extend the bend to the next boundary.

- Default direction of the edge flange is normal to the base flange at the edge.
- Creating an edge flange along a non-planar edge adds a 3D sketch.

#### Limitations

- You cannot select additional edges that are not tangent to the original edges.
- Only **Bend Outside** is available for edge flanges created on the straight edge of lofted or cylindrical parts.

See Edge Flange in the help.

#### **Lofted Bends with Bend Lines**

Bend lines are displayed with lofted sheet metal flat patterns.



To control bend lines, you can do one of the following:

- Select Number of bend lines and enter a value.
- Select **Maximum deviation** and adjust the value. Decreasing the value, increases the number of bend lines.

Bend lines are displayed only if the lofted bend area is well-defined (see example below), and can be created using brake press manufacturing methods. To create well defined lofted bend areas, make each face separate. For example, you can make the number of straight and curved elements in each profile equal.



See Lofted Bends in the help.

## **Transform Sketches from Flattened to Bent State**

Transform any sketch and its locating dimensions from a folded state to a flattened state and back again. The sketch and the locating dimensions are retained.



#### To see the sketch dimensions:

- 1 Flatten the model.
- 2 Expand **Flat-Pattern**  $\square$  in the FeatureManager design tree.
- 3 Expand **Sketch Transformation** in and double-click the derived sketch.





8

# **Other Functionality**

This chapter describes the installation process.

It also describes enhancements to functionality of the software in the following areas:

- Installation
- Application Programming Interfaces
- COSMOSFloWorks
- Display
- DWGeditor
- Import/Export
- SolidWorks Explorer
- Weldments

# Installation

#### SolidWorks Installation Manager

The SolidWorks Installation Manager helps you manage SolidWorks individual or administrative image installations. Its purpose is to download and update your installation when Service Packs and major versions are available. The Installation Manager:

- Determines which Service Packs apply to your installation.
- Uses acceleration and resume technology to download the files needed to update your installation based on the installed product.
- Applies the Service Pack to your existing individual installation or administrative image.

In SolidWorks, click **Help**, **Check for Updates**. Click **Check** to initiate a check manually or select **Check for updates once per week**.

#### **Units and Dimensions**

You are no longer asked to choose default units and dimensioning standard during installation. The **Units and Dimension Standard** dialog box appears the first time you open a new SolidWorks document.

## **Application Programming Interfaces**

Click **Help, SolidWorks and Add-Ins API Help Topics** to access the SolidWorks, eDrawings<sup>®</sup>, FeatureWorks<sup>®</sup>, PDMWorks<sup>®</sup> Workgroup, PhotoWorks<sup>™</sup>, SolidWorks Document Manager, SolidWorks Routing, SolidWorks Toolbox Browser, and SolidWorks Utilities Application Programming Interface (API) Help systems. The SolidWorks Document Manager API is new in this release.

The major enhancements and changes made to the SolidWorks API in SolidWorks 2007 are:

• **Blocks**. The SketchBlockDefinition and SketchBlockInstance interfaces replace the now obsolete and unsupported BlockDefinition and BlockInstances interfaces. The new interfaces support blocks in drawings, parts, and assemblies.



If your application creates blocks using the old interfaces, you must migrate your code to use the new interfaces. Blocks created in earlier versions of the SolidWorks software using the now obsolete interfaces are automatically converted to the new blocks.

- **Custom properties**. For ease of use, the CustomPropertyManager interface now contains all of the methods and properties for custom properties that previously existed on other interfaces.
- **3D-printer interface**. Use SW3DPrinter for 3D printer applications and common Print dialogs. This interface is for developing drivers for rapid-prototyping devices only.

- Weldment cut-list tables. Use WeldmentCutListFeature and WeldmentCutListAnnotation for weldment cut-list features and weldment cut-list tables in drawings.
- **User-defined materials in parts**. Access user-defined materials for part documents using MaterialVisualPropertiesData.
- Multiple Undo command. Create a SolidWorks Undo command to undo multiple SolidWorks API operations using ModelDocExtension::StartRecordingUndoObject and ModelDocExtension::FinishRecordingUndoObject.
- **DXF/DWG files**. Load DXF/DWG files using SldWorks::LoadFile4 and the ImportDxfDwgData interface.
- **Multi-row callouts**. Create and edit multi-row callouts using SelectionMgr::CreateCallout2 and the new Callout methods and properties.
- Macro features. Create and edit multibody macro features using FeatureManager::InsertMacroFeature3, MacroFeatureData::EditBodies, MacroFeatureData::GetEditBodiesCount, MacroFeatureData::IGetEditBodies, and MacroFeatureData::ISetEditBodies. Embed the macro file in the document that contains the macro feature using MacroFeature::EmbedMacroFile.
- **Drag arrows**. Create drag arrows, which are called handles in the userinterface, using Manipulator::GetSpecificManipulator and the DragArrowManipulator interface.
- **Edges in drawings**. Get the edges associated with the display data in a drawing using View::GetPolylines6.
- **Annotations and spell check**. Spell check annotations using Annotation::CheckSpelling.
- **Splitter windows**. Display ActiveX controls in splitter windows using ModelViewManager::AddControl3.
- Triads. Change the labels of the SolidWorks triad using SIdWorks::SetUserPreferenceStringValue swReferenceTriadXLabel, swReferenceTriadYLabel, swReferenceTriadZLabel, and SIdWorks::SetUserPreferenceToggle swReferenceTriadUseAlternateLabels.
- **SolidWorks commands**. Activate a SolidWorks command and optionally show your title for the command on the PropertyManager page using ModelDocExtension::RunCommand.
- Alternate position views. Superimpose one drawing view on another using View::InsertAlternateView.
- **Library features**. Create a library feature by creating the LibraryFeatureData object.
- Section views in parts and assemblies. Create section views in parts and assemblies using ModelViewManager::CreateSectionViewData, ModelViewManager::CreateSectionView, and the SectionViewData interface.
- **Smart components**. Create smart components using AssemblyDoc::CreateSmartComponent.

- **Temporary bodies**. Display a temporary body in the context of a specified part or component using Body2::Display3.
- **Interference detection**. Detect interferences in assemblies using AssemblyDoc::InterferenceDetectionManager and the InterferenceDetectionManager and Interference interfaces.
- **Mouse**. Access the mouse using ModelView::GetMouse and the Mouse interface.
- **User progress bar**. Create and modify a progress indicator using SldWorks::GetUserProgressBar and the UserProgressBar interface.
- **Updated FeatureData objects**. Updated mold-related FeatureData objects are: ExtrudedFeatureData2, PartingLineFeatureData, SimpleFilletFeatureData2, SimpleHoleFeatureData2, and VariableFilletFeatureData2.
- **swHtmlInterface control for QuickTips**. If your application has QuickTips, then you must change swHtmlInterface.14 to swHtmlInterface.15 for SolidWorks 2007.

# COSMOSFloWorks

#### **Animation Enhancements**

Create very high quality result animations. Animation improvements include camera movements, sequential animations, and moving arrows in cut plots and cut plots with animated trajectories.



## **Cavitation Model**

COSMOSFloWorks<sup>™</sup> can indicate areas in the model where cavitation can possibly occur and allow you to explore design changes that prevent it.



# Fully Developed Flow for Rectangular Cross Section

Consider a fully developed flow at the inlet if the cross section of the inlet is rectangular. In earlier releases, this option was available for circular cross section inlets.



#### **Improved Mesh Control**

Specify the number of cells you want between 2 control planes. In addition, it is possible to define several control planes in a single dialog box.



#### **Orthotropic Material**

COSMOSFloWorks can now consider different thermal conductivity in different directions. You can input different values of conductivity in x, y and z directions.



# Peltier Thermo Electric Cooler (TEC)

Simulate a complex heat sink with a TEC using this new feature. The complex heat sink model is replaced with simple box geometry which takes into account all the properties of a TEC.



## **Relative Humidity Calculation**

COSMOSFloWorks can now calculate the relative humidity distribution in the flow model. You can track the moisture distribution in the entire flow domain.



## Thermostat

Model thermostats and relate the heat source to the temperature at any location in the model.



## Thin Wall Technology

COSMOSFloWorks now utilizes proprietary thin wall technology to solve problems involving thin geometry very efficiently and quickly.



# Display

## Lighting

When you create a new part or assembly, default lighting now includes one **Ambient** and three **Directional** lights.

#### **Rotate View**

**Rotate View** has been enhanced so that if you rotate the view while zoomed in on the model, the model does not fly off the screen. **Rotate About Screen Center** has been removed from the **View**, **Modify** menu. Previously saved macros and API calls that use the **Rotate About Screen Center** option are respected.



## **RealView Graphics**

RealView Graphics enhancements include the ability to:

- Change the color, diffusion, specularity, and shininess of materials through the **Color and Optics** PropertyManager.
- Assign a material to a part while editing the part from an assembly.
- See some changes in lighting in parts that use RealView materials.

## **Tangent Edge Display**

In Tools, Options, System Options, Display/Selection, the Part/Assembly tangent edge display option now also applies to models in Shaded With Edges mode.

## **DWGeditor**

You can save a file as a pdf document within DWGeditor. Click **File**, **PDF Out**, or **File**, **Export to File** and set **Save as type** to **PDF file (pdf)**. Multi-sheet drawings are exported to one pdf with multiple pages.

#### Import/Export

#### DXF 3D

You can import DXF 3D solid models into SolidWorks drawings. The models are displayed as solid entities and can be shaded, rotated, etc.

#### Colors

You can export colors to DXF/DWG files by layer and by block. Colors are mapped with True Colors for both import and export.



An option for **End Point Merging** has been added to the **DXF/DWG Export Options**. Select **Enable merging** and specify the tolerance within which gaps between line endpoints are eliminated.

#### **Import Options**

An option for **Import Dimensions** has been added to the DXF/DWG Import Wizard. When you import DXF files to new parts, on the **Document Settings** page, you can choose whether to import dimensions.

## SolidWorks Explorer

SolidWorks Explorer has a redesigned user interface, and now includes:

- Functionality previously found in PDMWorks Standalone client.
- Enhanced search capabilities with the new SolidWorks Search assistant.

See SolidWorks Explorer in the help.

## Weldments

For weldment cut lists, the settings in **Document Properties** now control the unit of measure and the trailing zeros in the **LENGTH** column.

9

# SolidWorks Office Components

This chapter describes enhancements to the SolidWorks Office components in the following areas:

- General enhancements
- COSMOSMotion
- COSMOSWorks
- eDrawings
- FeatureWorks
- PDMWorks Workgroup
- PhotoWorks
- ScanTo3D
- SolidWorks Animator
- SolidWorks Design Checker
- SolidWorks Routing
- SolidWorks Task Scheduler
- SolidWorks Toolbox
- SolidWorks Utilities

#### **General Enhancements**

SolidWorks add-in toolbars and menu items are more usable and consistent. You can customize toolbar buttons, add toolbars to the CommandManager, make toolbars as flyout toolbars, and toggle the display of toolbars in **View**, **Toolbars**.

The settings for add-ins are transferred from one release to the next and can be saved and restored with the Copy Settings Wizard.

The following add-ins are supported:

FeatureWorks	SolidWorks Design Checker	
PDMWorks Workgroup	SolidWorks Routing	
PhotoWorks	SolidWorks Toolbox	
SolidWorks Animator	SolidWorks Utilities	

The **Add-Ins** dialog box (accessed by clicking **Tools**, **Add-Ins**) shows active add-ins by category: the SolidWorks Office product you have purchased, add-ins to the SolidWorks application, and other add-ins that you might have purchased.

Select add-ins for the current SolidWorks session (check boxes on the left) and to be included at the next SolidWorks start-up (check boxes on the right).

The SolidWorks add-in help files listed below are integrated into the main SolidWorks help file. The **Help**, *<add-in>* menu items have been removed.

2D Command Line Emulator	PhotoWorks
3D Instant Website	ScanTo3D
3D Meeting	SolidWorks Animator
COSMOSMotion	SolidWorks Design Checker
COSMOSWorks	SolidWorks Explorer
COSMOSXpress	SolidWorks Routing
DXF/DWG Import Wizard	SolidWorks Toolbox
FeatureWorks	SolidWorks Utilities
PDMWorks Workgroup	

# **COSMOSMotion**

#### Integration with SolidWorks Office Premium

COSMOSMotion<sup>™</sup> is fully integrated with SolidWorks Office Premium meaning that assembly geometry, mates, and drivers from Physical Simulation are used to run an analysis directly in COSMOSMotion to determine reaction forces.



#### Simulate with SolidWorks Animator and eDrawings

Add COSMOSMotion simulations to SolidWorks Animator as key frames using the Animator wizard. Use the camera functionality to visualize the motion results from the camera perspective. Also, save COSMOSMotion simulations as eDrawings files.



#### **Gear Teeth Forces**

Simulate the interaction between spur gear, bevel gear pairs and calculate gear teeth contact force using new Gear joint. This new joint uses less resources and time than 3D contact simulation from earlier releases.



## Multiple "What-if" Scenarios

Create multiple studies to compare and visualize different design alternatives.



## Load Bearing Faces

When creating mates in SolidWorks, you may now specify the load bearing faces to be used as a part of the COSMOSMotion analysis. COSMOSMotion uses this load bearing face information to accurately to transfer motion loads to COSMOSWorks.



# COSMOSWorks

This document contains a partial list of the COSMOSWorks<sup>®</sup> enhancements. See the COSMOSWorks online help for a complete list of enhancements with links to more details.

The following enhancements are available in COSMOSWorks Designer and above unless otherwise noted. Enhancements marked with a "P" are available in COSMOSWorks Professional and above. Enhancements marked with an "A" are only available in COSMOSWorks Advanced Professional.

## Adaptive Analysis for Assemblies

The h-adaptive method of mesh refinement now supports assemblies. In earlier releases, only parts were supported.



The bearing connector allows you to replace a physical bearing with a virtual component to duplicate the behavior of bearing. You can specify local stiffness as well as free rotations for this connector.

## **Bolt Connector Enhancements**

The software can now automatically project the bolt diameter and nut diameter area onto the faces of components held together by bolts. In earlier releases, you had to create a split line to define the faces.







# Custom Result Plots and Tree Structure

Specify the default plots and the result folder structure for each study type. As before, you can add additional plots after the analysis is complete.

# **Cyclic Symmetry**

Cyclic Symmetry is a Load/Restraint option that allows you to analyze a portion of a model when the geometry and boundary conditions repeat in a circular pattern.

# Envelope Plots <sup>P</sup>

Plot the maximum or minimum result value at a given location in a model across several results sets for transient thermal, drop test, and nonlinear analyses. Probe the envelope plot to determine the output value and the time step value.




### Follower Loads for Nonlinear Analysis <sup>A</sup>

Follower loads can update their orientation as the geometry they are attached to changes orientation.

This enhancement is available in COSMOSWorks Advanced Professional.



### **Free Body Forces**

Request free body forces between parts connected with bonded contact.



### **Mesh Quality Plots**

Create mesh quality plots such as aspect ratio and distortion of elements (Jacobian check). Plot tools such as section clipping, iso clipping and probe are also available for these plots to better isolate the areas with unacceptable distortion or aspect ratio.

### **New Animation Options**

Animate results in a forward-only, looping, or reciprocating pattern. You can save **.avi** files with this new sequence.

Additionally, use SolidWorks Animator to rotate, explode, or collapse assemblies with analysis results. Use the camera functionality to visualize the motion results from the camera perspective. You can save them as .**avi** files for use in presentations.



### **Remote Mass**

Represent certain purchased parts such as motors and actuators as a remote mass. Rather than create a CAD model of the parts, you consider the effect of these parts on rest of the assembly by specifying the mass and moments of inertia.



### **Section Clipping Handles**

Drag the section clipping planes by providing drag handles on the results plots.



### Sensors

Use sensors to retrieve result quantities such as stress and displacement on a repeatable basis. Sensors are defined at a location on the model or a specified x,y,z coordinate. You can save sensors per plot and display them when printing. Also, use sensors to query local reaction forces.

### Spring Connector Enhancements

Define tension only and compression only springs. In addition, springs can now be defined between cylindrical faces or 2 points or vertices.

# Values above Yield Stress in Specific Color

Plot vonMises stress above yield strength in a user-specified color to highlight areas of potential failure.

### Weldment Analysis

Analyze weldments efficiently using beam elements. A beam element is defined by two end points and an appropriately oriented uniform cross-section. The software automatically derives beam cross-sections from solid geometry, detects joints, and applies loads and restraints.









### eDrawings

### **Paper Color**

To define the paper color for drawings, click **Options** or **Tools**, **Options**, select **Override paper color**, and choose a color.

### Stamp Tool

To add stamp images to 2D and 3D documents, click **Insert Stamp** and or **Tools**, **Stamp**. A set of transparent stamps is provided in eDrawings. You can also specify the stamp folder and add images to the specified folder.

A 2D stamp behaves like a rubber stamp applied to a paper drawing. A 3D stamp remains fixed when the model is scaled, rotated, or dragged. You can drag and resize 2D stamps and drag 3D stamps.

Stamps become permanent when the file is saved.

### Images

To import images into documents, click **Image** or **Tools**, **Markup**, **Image**, browse to an image file (.bmp, .gif, .tif, .jpg), click **Open**, and place the image in the graphics area.

You can drag, resize, and delete images.

### **Check for Updates**

eDrawings checks periodically for service pack

updates and installs them. In **Tools**, **Options**, select or clear **Check for service pack updates every 30 days**.







### Measure Tool

**Measure** A displays measurement data in callouts that identify the value (**Length**, **dX**, **Center Dist**, etc.) and display units. Candidates for measurement turn red when the pointer hovers over them.

**Measure** an measure arcs and circles, both lines in 2D DXF/DWG drawings and real edges in 3D models. The arc's diameter and center coordinates are displayed.

### **User Interface**

The tabs for the left panel are identified by images:



- 🎨 Components
- 😫 Configurations
- 🎒 Layers
- 🥖 Markup
- 📰 Sheets

To hide the panel, click the arrows 0 or the bar between the arrows  $\blacksquare$ . This procedure replaces **View**, **UI Mode**, **Simple** or **Complete**.

Markup tools appear in the left panel border rather than the right window border.

### Printing

The **Print** dialog box now includes options:

- Select Color/grayscale or Black & white.
- For the Active sheet (drawings only), choose As displayed, Entire sheet, To scale (1 to 1), or Selected area.
- Click Line Weights and define Thin, Normal, Thick, etc. for both System settings and Document settings.

For files published from SolidWorks, the **Document settings** are inherited from the SolidWorks **Line Weights** in effect at publishing time.

### Layers

The **Layers** tab ists the layers displayed in the graphics area. You can select or clear layers to display or hide them. The initial state reflects how documents were saved in the original document. Resaving documents in eDrawings saves the layers in their initial state. Empty layers and hidden annotations and dimensions are included. Layer states apply to the whole document, not by sheet or view. Markups capture the state of the layers, so if a markup is retrieved, layers are displayed as they were when the markup was created. Shaded data for components in assemblies is published with the layer data.

### **Thickness Analysis Results**

Save the results of a SolidWorks Utilities thickness analysis in an eDrawings file. Select **Save results in eDrawings** in the **Thickness Analysis Save Report** dialog box. For multibody parts, only analyzed bodies are saved.

### Support for AutoCAD 3D DXF Files

Display and manipulate imported 3D DXF documents like SolidWorks models.



2006





### **Microsoft Office Integration**

You can embed eDrawings files in Microsoft Office documents such as Word, Excel, and Power Point. Drag the eDrawings files into the applications or insert them as objects. In Power Point, to be able to interact with eDrawings, you must insert the file as an object.

Double-click the eDrawings file name to launch the eDrawings Viewer.

### **FeatureWorks**

FeatureWorks recognizes the following geometry:



Ribs with negative draft



Rolled hems



Tear drop hems







Linear feature patterns

Circular feature patterns

Rectangular feature patterns

Rib features can be re-recognized as boss extrudes and vice-versa.

### **PDMWorks Workgroup**



PDMWorks has been renamed to PDMWorks Workgroup.

### **Checking in Drawings**

Performance has accelerated when you click **Find and include associated drawings** upon check in. Previously, each drawing found was opened during the find operation which slowed performance. Now, drawings are opened only after you click **Check In**.

See Check In Document in the help.

To further accelerate performance, the option **Skip drawings validation during check in** is available on the **Vault Settings** tab in the VaultAdmin. This allows you to check in drawings without prior validation in the SolidWorks software.

> In drawings where views or annotations are out-of-date or are about to change upon check in, the drawing in the vault will not be up-to-date. For example, title block text that is linked to a revision custom property will be incorrect.

See Vault Settings in the help.

### **Compatibility Between Service Packs**

You can run PDMWorks Workgroup clients with the PDMWorks Workgroup server from any service pack as long as the clients and server are within the same major version. For example, you can use a client from an earlier service pack with a server from a newer service pack as long as the client and server use PDMWorks Workgroup 2007. The following table illustrates when you need to update PDMWorks Workgroup clients:

Client	Action
Same release as vault	None
Newer than vault, same major release	None
Newer than vault, newer major release	Login is blocked
Older than vault, same major release	Optional update in the <b>About</b> SolidWorks Explorer dialog box
Older than vault, older major release	Mandatory update

### **Copy Settings Wizard**

PDMWorks Workgroup settings are now included when you use the Copy Settings Wizard.

### **Document Information**

The **Document Information** dialog box has been simplified. Information from the upper half of the dialog box has moved to the **Info** tab.

### **Multiple Time Zones**

PDMWorks Workgroup supports multiple time zones in a work environment. For example:

- If daylight savings time changes, you do not have to restart the vault.
- Documents in one time zone have proper time stamps when users work with a vault in another time zone.
- If a vault was created in one time zone, then it is relocated and restarted in a different time zone, the vault updates properly.

### My PDMWorks Cleanup

My PDMWorks Cleanup helps you manage your personal workspace. When you rightclick a folder in the SolidWorks software or in SolidWorks Explorer (with the PDMWorks Workgroup add-in) and select My PDMWorks Cleanup, it searches the folder and shows you all of the files based on selected search criteria. You can then clean up your workspace by deleting selected files.

🤣 See My PDMWorks Cleanup in the help.

### Online Help

The PDMWorks Workgroup online help is installed at *<install dir>\lang\english*. If the SolidWorks software is not installed on your computer, the PDMWorks Workgroup online help is installed at:

**Program Files**\SolidWorks\<*PDMWorks Workgroup product*>\lang\english

### **PDMWorks Advanced Server and PDMWorks Triggers**

PDMWorks Advanced Server no longer exists. As a result, PDMWorks Triggers are installed as part of the PDMWorks Workgroup Server. The Web Portal, (renamed to PDMWorks Workgroup Viewer), is installed as an independent product.

### **Project Permissions**

You can apply permissions from one project to others. On the **Projects** tab in the VaultAdmin, click Copy Project Permissions to propagate all permissions from one project to all selected projects.

### Searching

#### Search Results

When you search the vault, documents with more than one instance in the vault tree are listed only once.

### Indexing

The VaultAdmin includes a **Search** tab that allows you to build an Indexing Service catalog for the vault metadata. Indexing increases performance when you search the vault for documents. To use the indexing functionality, you must install the Microsoft Indexing Service.



🤣 See **Search** in the help.

### Select Labels

The **Select Labels** tool has been removed from the **File Explorer** tab in the Task Pane. You can access the functionality by right-clicking in the vault view and selecting **Display** Labels With, Select.

### Standalone Client

The PDMWorks Standalone client has moved into SolidWorks Explorer. If you have a PDMWorks Workgroup license, the vault view appears in SolidWorks Explorer and is referred to as PDMWorks Workgroup Contributor. See SolidWorks Explorer on page 8-9.

### Switching Between Vaults

If your company has more than one vault, you can quickly log out of the current vault and log in to a different one. Right-click in the vault view and select Switch Vault/User. The Login dialog box has a field that lists all installed vaults so you can switch vaults or log in as a different user.



You cannot switch between vaults in the Web Portal.

#### Web Portal



The Web Portal has been renamed to PDMWorks Workgroup Viewer.

A simplified version of the PDMWorks Workgroup Viewer is available. This version is primarily used for searching. To access the simplified version, in the PDMWorks Workgroup Viewer, click **Go to Search Portal** .



If you click the icon to open a document as an eDrawings file, you must use eDrawings 2007.



See PDMWorks Workgroup Viewer in the help.

### **PhotoWorks**

PhotoWorks includes enhancements to:

- Materials
- Materials display in options
- Materials library
- Scene Editor
- Preview Window

#### Materials

The **Materials** dialog box has moved to a PropertyManager. The multi-tab PropertyManager is fully interactive, allowing you to apply materials in the same manner as you apply decals.

The **Materials** in the Task Pane, enabling you to drag materials onto your model. The Task Pane integrates all file management capabilities. From the **Materials** is folder, you can browse to a material and then drag the material to:

- The graphics area, to apply the material to the model and open the PropertyManager.
- The graphics area while holding **Alt**, to apply the material without opening the PropertyManager.
- Onto faces to apply the material only to that face.
- Features and bodies in the FeatureManager design tree, to apply to those items.

You can also double-click the material in the Task Pane to display the **Materials** PropertyManager.

### **Surface Finish**

New surface finish types were added to support new materials (see **New Materials and Enhanced Materials** on page 9-21) such as improved textured plastics.

New surface finishes include:



Rough and Smooth

New surface finish properties include **High Threshold** and **Low Threshold**. These controls mimic the effect of texturing a mold for a plastic part. They work in tandem to control the extent to which surface finish elements display on a material surface. The effects of high and low threshold are dependent on your amplitude settings.

See Surface Finishes in the help.

### Manipulators

Manipulators like those used with decals are now used with projection, spherical, and cylindrical mapped textures and for automatic mapping.



See Adding Materials and Textures in the help.

### **Materials Display in Options**

#### **System Options**

Under Material Display, the following options appear:

- **Display PhotoWorks Materials in SolidWorks**. Replaces **Overwrite SolidWorks Properties**. This option allows you to see a very accurate representation of the PhotoWorks material prior to rendering in SolidWorks OGL. The closest approximations occurs with single color, procedural materials.
- Display SolidWorks Colors in PhotoWorks. Replaces Use SolidWorks materials.



• Use SolidWorks part physical materials as default material. Replaces Use SolidWorks physical materials.

### **Document Properties**

**Store PhotoWorks data in model file. (Note: May increase file size.)**. Select to store PhotoWorks material with the model. This option stores the images used materials, decals and scenes. If increased file size is not an issue, this allows to share models that include all the required PhotoWorks data, instead of using **Pack and Go**.

### Illumination

The illumination produced by selecting both **Enable caustic** and **Enable global illumination** is more realistic. The **Diffuse** property of materials now accurately influences models when both options are selected. This results in more realistic renderings based on the setting.

### **Materials Library**

The reorganized materials library includes:

- Re-calibrated materials
- New materials
- The capability to share PhotoWorks materials, decals, and scenes with other users
- Greater efficiency in locating missing materials, decals, and scenes

#### **New Materials and Enhanced Materials**

Textured plastics Satin finish plastic Plastic meshes Chain link steel Sandblasted glass Translucent plastic Frosted plastic Brushed metals Sprayed paint Water Polypropylene High gloss white plastic Sandblasted metals Frosted glass Wood

#### Sharing Materials

The option **Store PhotoWorks data in model file** enables you to share PhotoWorks data so that other users can reproduce the same renderings (see **System Options** on page 9-20). You can also share materials, decals, and scenes using the streamlined **Pack and Go** interface.

You can select the following:

- The option **Include PhotoWorks files from default library**. This enables you to exclude default library items to reduce the size of **Pack and Go**.
- To view fields for Name, In Folder, Size, Type, and Date Modified.
- The option to add **Zip files**.
- To open **Pack and Go** through a central SolidWorks interface.

Materials in the system are reorganized by common usage rather than alphabetically to facilitate search.

#### **Searching for Missing Materials**

Search for missing materials, decals, or scenes activates only prior to rendering. Previously, this occurred when opening a file. In the Render Manager , materials, decals or scenes that have missing image are flagged. You have the option to ignore the missing image file warning.

The potential for missing materials is minimized now that you can:

- Store materials with files (see **Document Properties** on page 9-20).
- Update image file paths during copy and save operations (from the Color/ Image 🔜 tab in the **Materials** PropertyManager).
- Update image file paths using **Pack and Go**.

See Materials Library in the help.

### Scene Editor

Enhancements were added to several tabs of the **Scene Editor** dialog box. These enhancements were added to support High Dynamic Range Imaging (HDRI). Four HDRI scenes, accessed through PhotoWorks **Studios**, were added:

- Conference Room
- Industrial Space
- Interior Kitchen (default)
- Sunset

#### **Back/Foreground**

- Streamlined selection method for **Background**.
- Enabled preview image.
- Added the Background Image Brightness slider.



Controlling the Background Image Brightness

- Streamlined image path location when the selected **Background** type is **Image**. Click **Browse** to locate the image.
- Moved Environment to new tab.

### Environment

The **Environment** tab includes items previously located in the **Back/Foreground** tab.

- **No Environment**. Make the background selection from the **Manager** tab.
- Use Background Image. Make the background color selection from the Back/Foreground tab; click System Colors to restore the default.
- Select Environment Image. Browse to locate an image that is identified under Image File Path. Then select from the Environment Mapping options: Planar, Spherical, Or Cubic.

Under Image Contribution, there are two new options:

- Environment Reflections
- Diffuse Material Brightness

Use the **Environment Contributions** slider to control the environment's reflection on objects. At a low setting, you get a dull reflection; at a high setting, you get bright reflections. Materials such as polished chrome display this range very clearly. The **Environment Contributions** option manipulates:

- RGB (Red-Green-Blue) values
- HDR (High Dynamic Range) values

High dynamic range manipulates the contrast between bright and dark areas. Manipulating this range results in more accurate depictions of how the light affects objects, their shadows, and how those objects reflect the environment.



Default Setting: Environment Reflections: 1.00



Environment Reflections: 0.44



Environment Reflections: 1.70

The **Diffuse Material Brightness** slider controls the overall brightness of a scene (the objects and environment), without affecting reflections. The reflections below are uniform.





See Scene Editor in the help.

### **Preview Window**

While editing a material, decal, or scene, you can toggle the display of the **Preview** - [model name] (PhotoWorks toolbar).

#### Geometry

Choose Selected Reference to preview rendering on selected geometry.

#### Components

The following rendering and illumination properties were added to the **Preview**:

- Indirect Illumination
- Caustics
- Global Illumination
- Anti Aliasing (turns down the current anti-aliasing setting lower)
- **Surface Finish** (renamed)

Previously, you could not see the effects of indirect illumination, caustics, or global illumination in the preview window. These properties were toggled through **PhotoWorks**, **Options**.

### ScanTo3D

Using the SolidWorks software's ScanTo3D functionality, you can open scan data (mesh or point cloud files), prepare the data, then convert it into a surface or solid model.

ScanTo3D significantly reduces the time required to build complex 3D models from non-digital data. Designers can use ScanTo3D for various purposes:

- Medical designers Create anatomical objects for reference
- Consumer product designers Create quick representations of physical components made from clay, foam, etc.
- Machine designers Create quick references to OEM parts



ScanTo3D is available only in SolidWorks Office Premium.

Two methods convert scan data to a solid model:

- Semi-manual creation
  - Open the scan data
  - Prepare the mesh using the Mesh Prep Wizard
  - Sketch 2D and 3D curves on the mesh
  - Build surfaces using the curves
  - Trim and knit the surfaces
  - Create the solid model
- Semi-automated creation using wizards
  - Open the scan data
  - Prepare the mesh using the Mesh Prep Wizard
  - Use the Surface Wizard to create surfaces using automatic or guided creation
  - Trim and knit the surfaces
  - Create the solid model

The Mesh Prep and Surface Wizards guide you through the ScanTo3D process. If you already have clean mesh data, use just the Surface Wizard to create a solid model. Use the Surface Wizard's automatic creation for anatomical and organic shapes. Use guided creation for analytical faces.





**Overview of the ScanTo3D Process:** 

In the following example, you use the semi-automated wizard method, using the Surface Wizard's guided creation, to create a solid model.

#### To open mesh files:

- 1 Click **Open** (Standard toolbar) or **File**, **Open**.
- 2 Select Mesh Files (\*.xml, \*.nxm, \*.scn, \*.3ds, \*.obj, \*.stl, \*.wrl, \*.ply, \*.ply2) in Files of type.
- 3 Browse to ScanTo3D\BRIGHT-COLOUR.nxm and click Open.

The NextEngine mesh file takes a few minutes to open.

4 Zoom in on the model and examine it.



#### To use the Mesh Prep Wizard:

- 1 Click Tools, ScanTo3D, Mesh Prep Wizard.
- 2 Select the mesh.

The pointer changes to  $k_{\rm A}$ . The PropertyManager reports the number of faces.

- 3 Click ().
- 4 Under Orientation Method, select Numeric input and make sure Rotation Parameters are all set to 0.00deg.

Orientation aligns the mesh to the global axes and planes, which is important for accurate downstream manipulation of the model.

5 Click (•).

This mesh has no extraneous data, which typically comes from the fixture used to hold the part in place while you scan it.

- 6 Click (•).
- 7 Type **50** in **Reduction amount (%)** and press **Enter** to reduce the mesh size by 50 percent.

The PropertyManager reports the original (**258174**) and final mesh size (**129087**). Your size may vary.

For your models, adjust simplification to the mesh size that your computer can process.

8 Click 📀.

Now you smooth the model and fill holes.

 Under Global Smoothness, move the slider to approximately three-quarters as shown to smooth the entire model and remove most of the rough texture.



- 2 Click **Front** [6] (Standard Views toolbar), then zoom to the lower left corner of the mesh.
- 3 Under Local Smoothness, click Polygon Selection , then sketch a polygon to select the mesh approximately as shown.
- 4 Select Allow selection depth adjustment.

This 3D tool, aligned to the view orientation, lets you select the depth of the model to smooth.



- 5 Click **Rotate** (View toolbar) and rotate the model to the concave bottom area, then use the pointer area to drag the depth approximately as shown.
- 6 Rotate the mesh to the front view, then move the slider to approximately three-quarters to smooth the local area.
- 7 Click 🕟.

The PropertyManager lists the holes to fill, which are highlighted in the graphics area.



8 Click () to fill all holes and complete mesh preparation.

Under **Options**, note that **Launch Surface Wizard** is selected to continue directly to the Surface Wizard.

9 Click 🕟.

#### To use the Surface Wizard:

- 1 In the PropertyManager, click **Guided creation**.
- 2 Click ().

In this case, you do not split the mesh.

If you want to split the mesh symmetrically, you would select a plane of symmetry to split the model in half. You would surface the remaining half, then mirror the surface for symmetrical results. This procedure ensures the data is smooth across the mirror plane

3 Click 🕟.

You decompose the meshes into sub-meshes shown by colors. The goal is to try to eliminate radii from the part as much as possible so you can add them later using the SolidWorks application.

You would use paint tools to fill in sub-meshes that you fit with b-spline (non-analytic) surfaces because missing pieces would affect the shape of the fitted surface.

In this sample however, you will extract mostly analytic surfaces, so you do not need to paint the faces. However, you do need to make sure that the front and back of the model each have mostly one different color.

4 Move the sensitivity slider to the right to adjust the colors until the front and back of the model each have mostly one color. This is important to let you extract proper-sized analytical surfaces.

5 Click ().

6 Select the light blue sub-mesh on the front face.



Your system colors may be different.



#### 7 Under Face Settings, click Cylinder 😭.

A cylindrical surface is created by extracting the selected sub-mesh data.

8 Rotate the model to the back face and repeat the same procedure.



Front face



Back face

9 Select the top and bottom sub-meshes, click Spline based surfaces.

The model should resemble the image.

#### 10 Click 📀.

Surface preparation is complete. The surfaces are extracted to the **Surface Extracted1** folder in the FeatureManager design tree.





#### To finish the model, you would:

- 1 Use surfacing tools (such as **Untrim**, **Trim**, and **Extend**) to trim unwanted surfaces.
- 2 Knit the surfaces together.
- 3 Thicken the surfaces to create a solid model.
- 4 Apply fillets using the original mesh for reference.



Click the image for 3D viewing

### **SolidWorks Animator**

### **Camera Control**

You can control camera movement while looking through the camera:

- Back and forth along the target path
- Up, down, left, and right in the view plane
- Change the pitch and yaw
- Roll it about the target line

Camera movements use standard views. Mouse and keyboard combinations such as the following also control the camera view:

Keyboard/Mouse Action	Camera Action	Pointer
Middle mouse button	Orbit	
Ctrl+ Middle mouse button	Pan	- 
Shift+ Middle mouse button	Zoom in/out	Qt ∰∳
Alt+ Middle mouse button	Roll about the target line	G a.♦
Ctrl+Alt+ Middle mouse button	Rotate about the target point	

Camera movement control is possible when you are editing the **Camera** PropertyManager, and outside the PropertyManager if you clear **Lock camera position except when editing** under **Camera Type** in the PropertyManager.

### COSMOSMotion

If COSMOSMotion is installed and active and your model has a computed COSMOSMotion simulation, you can import the simulation using the Animation Wizard.

### **Enhanced Light and Camera Support**

You can define these camera properties at key points in an animation:

- Target point position
- Camera position
- Camera used to view the animation
- Depth of field
- Field of view
- Light properties (position, brightness, cone angle, etc.)

To create walk-through animations, you can define key points that change camera properties to create **Camera** key frames in the animation. Therefore, you no longer require a camera sled to animate the camera position.

You can also view an animation through multiple cameras, toggle the display of cameras and lights, and create animations that show the cameras and lights.

#### To see the enhanced light and camera support:

- 1 Open Animator\Speaker.Sldasm.
- 2 Click **View** and toggle the display of **Lights** and **Cameras** to show and then hide the light and camera manipulators in the graphics window.
- 3 Select the Animation1 tab and expand Lights and Cameras in the Animator FeatureManager design tree.

Two cameras exist. The beige bar in the **Camera1** key frame indicates the animation is viewed through **Camera1** from time **00:00:00** to **00:00:06**. An explode/ collapse animation also exists.

- 4 Place the timeline at 3 seconds.
- 5 Right-click **Orientation and Camera Views** in the Animator FeatureManager design tree and clear **Disable View Key Creation**.



When this option is cleared, orientation and camera view changes are placed as key frames in the animation.

- 6 In the Camera2 row in the Lights and Cameras folder:
  - a) Right-click the timeline and select **Place key**.
  - b) Right-click **Camera2** in the Animator FeatureManager design tree and select **Properties**.
- 7 In the PropertyManager, clear Lock camera position except when editing under Camera Type, then click 🖌.
- 8 In the lower-left corner of the graphics area, select **Camera2** from the View pop-up menu.
- 9 Click Pan (View toolbar) or Ctrl+ middle mouse button and center the model in the graphics window.

10 Drag the timeline to 6 seconds and place another key point in the **Camera2** row at 6 seconds.



This adds a view orientation key point that sets the view to be through **Camera2**.

- 11 Select **Camera2** from the View pop-up menu.
- 12 Click **Zoom In/Out** (View toolbar) or press **Shift+** middle mouse button and zoom the model in slightly, then right-click **Orientation and Camera Views** and select **Disable View Key Creation**.
- 13 Click Play from Start D.

The model explodes using the **Camera1** view, then collapses and zooms in looking through **Camera2**.



### Mate to Curve

You can mate a component to a single entity curve, then drag the component along the curve. See **Mate to Curve** on page 4-9. You can use this functionality to control a component's movement in an animation along a path defined by a curve. If you attach the camera position or target point to a single entity curve, you can animate the position along the curve, which is a convenient way to define camera walk-through animations.



### **View Orientation**

**View Orientation** has been renamed as **Orientation and Camera Views**. To unlock views, right-click **Orientation and Camera Views** and clear **Disable View Key Creation** (previously named **Lock**).

### SolidWorks Design Checker

The following are new in SolidWorks Design Checker:

- Auto correction
- Learn Checks Wizard
- Merge Standards
- New design checks
- Results in Task Pane
- Welcome to SolidWorks Design Checker dialog box

### **Auto Correction**

Use **Update Values** to specify and correct checks that failed. You can use **Update Values** for most design checks. For a list of exceptions, see **Auto Correction** in the help.

#### **To use Auto Correction:**

1 Click Design Checker, Check Active Document.

The list Failed Checks appears in the upper Design Checker Task Pane.

2 Select each check (for example **Document's Dimensioning Standard Check**).

The Parameter Name and Value is displayed below.

3 Expand the check and select the name of the model to list the **Expected** Value(s) and Actual Value(s):



4 Click **Update** to display the new values under **Passed Checks**.



Correction values must be in the same units as expected values.

See Auto Correction in the help.

### Learn Checks Wizard

Use the **Learn Checks Wizard** to retrieve design checks based on attributes from an existing SolidWorks part, assembly, or drawing document.

#### To use Learn Checks Wizard:

1 Open a SolidWorks document and click **Design Checker**, Learn Checks Wizard.

The Task Pane lists all the available check types in the document.

2 Expand each check type and select the checks to include.

As you select a check, the current **Parameter Name** and **Value** is displayed.

- 3 Click **Done** to display:
  - **Dialog View**. Lists your selections in each category.
  - Summary View. Lists your selections from each category by Check Name, Parameter Name, Operator, and Value.
- 4 Click Save 园.

Use the **.swstd** file as a template to check other SolidWorks part, assembly, or drawing documents.

See Learning Checks in the help.

#### Merge Standards

You can merge **.swstd** files through **File**, **Merge Standards**. You can also create multiple standards files and then merge them into one master file. In the **Merge Standards** dialog box, click **Add** and browse to **.swstd** files.

The merged file assumes the name of the first file on the list, but you can rename it. Once the merged file is created, the new file and all the original **.swstd** files are retained.



If you merge two files with conflicting checks, the system assigns priority according to the order in which they are listed in the **Merge Standards** dialog box.

Once saved, the new merged file is opened so you can view all the check types and their values.

- The left panel lists the types of checks such as **Arrow Style**, **Dimension Font**, etc.
- The right panel lists the details for each type of check such as Font Style Regular, Spell Checker, etc.

### **New Design Checks**

New checks were added to SolidWorks **Design Checker**.

### Spell Checker

Runs through **Annotations** to locate spelling mistakes. Any annotation that includes misspelled words fails.

#### To use Spell Checker:

- 1 Activate Spell Checker 💯 through Annotations Checks 🕺 .
- 2 Select the terms to spell check from the **Design Checker** Task Pane.

#### **Overridden Mass Check**

Verifies if **Mass Properties** were assigned to part or assembly documents. Any overridden mass property fails.

The Overridden Mass and the Initial Mass are displayed under Remarks.

#### **Non-Standard Holes**

Verifies if holes in the part or assembly document comply with the standard hole sizes created with **Simple Holes** or the **Hole Wizard**.

Activate **Standard Hole Sizes (i**) through **Feature Checks (i**). For example, the results list could display the following:

#### Parameter Name

• Standard Hole Sizes

#### Remarks

- Custom Throughhole
- Diameter: 0.3420
- Standard Throughhole
- Diameter: 0.24320

In this example, the standard diameter for throughholes is **0.24320**. However, the model that was checked included a custom throughole using a non-standard diameter of **0.3420**.



**Tapered Drill** and **Tapered** are not supported for **Legacy Hole** types.

### Welcome Page

From the SolidWorks software, click **Design Checker**, **Build Checks** to display the new **Welcome to SolidWorks Design Checker** dialog box. Select or clear **Show welcome page at startup**. Links include:

- Create a New Standards Files. Opens Design Checker to build a new document.
- Open an Existing Standards File. Browse to a saved .swstd document.
- **Design Checker Help**. Opens the help.
- Design Checker Online Tutorial. Opens the Design Checker tutorial.

### SolidWorks Routing

### Toolbars

Each route type now has its own toolbar. Click items on the Route toolbar to turn other toolbars on or off.



### Auto Route

The **Auto Route** command now filters out invalid selections. For example in **Re-route spline** mode, you can select only splines, not lines or arcs. Similarly, in **Merge guidelines to form routes**, only guideline splines are highlighted, not splines that are already in valid bundles.

### **Back-to-Back Flanges**

You can drag a flange from the design library and place it back-to-back with a flange already on the end of the route. The second flange resizes to match the first one.



### **Flexible Conduit**

Select **Use flexible** in the **Route Properties** PropertyManager to make a conduit route flexible.



☆

You can use the **Auto Route** command on flexible conduit in the same way as on flexible tubing.

### **Routing Quick Tips**

Routing Quick Tips are messages that give hints and options about basic Routing commands and workflow. Click **Routing Quick Tips** (Route toolbar) or **Routing**, **Routing Quick Tips** to turn them on or off.

### Stub Length

You can specify **Stub Length** in the **Connection Point** PropertyManager, and you can set the stub length in design tables using the parameter **StubLength@CPoint**<*n*>.

### **Change Route Diameter**

The **Change Route Diameter** command now uses the PropertyManager instead of a series of dialog boxes. The route elements that are being changed highlight in the graphics area.

#### To change the route diameter:

- 1 Open Routing\changeroute101.sldasm.
- 2 Turn on the Piping and Routing Tools toolbars by clicking **Piping** and **Routing Tools** [6] (Route toolbar).
- 3 Click Edit Existing Piping Route (Piping toolbar) or Routing, Piping, Edit Existing Piping Route.
- 4 Select the line segment near the flange, as shown.



#### 5 Click Change Route Diameter (Routing Tools toolbar) or Routing, Routing Tools, Change Route Diameter.

The first section of the route, including the flange and tee, changes color, indicating that this section of the route corresponds to the items in the PropertyManager. Because you selected near the flange, the flange appears under **First Fitting** in the PropertyManager.



## In the graphics area, **First Fitting** is green; **Second Fitting** is blue.



6 In the PropertyManager, under:

- a) Options, select:
  - Automatically select elbows & pipes
  - Automatically save new pipe file
- b) First Fitting, select Slip On Flange 150-NPS3.
- c) Second Fitting, select Tee Inch 3 Sch40.
7 Click (•).The next section of the route highlights.



- 8 Under Second Fitting select Slip On Flange 150-NPS3.
- 9 Click 🕟.

10 Under Second Fitting select Slip On Flange 150-NPS3.

- 11 Click 🕢.
- 12 Click **OK** or **Yes** for any messages that appear.
- 13 Click Save in the Save As dialog box.

The elements of the route change.



See Change Route Diameter in the help.

## Create Route on the fly

You can start a route from a component that has not yet been defined as a routing component. Previously, a component had to have at least one connection point (**CPoint**) before you could start a route from it.

### To start a route:

1 Open Routing\electrical.sldasm.

Component **plug001** does not have a connection point.

- 2 Click **Electrical** 剩 (Route toolbar).
- 3 Click Create Route on the fly 💱 (Electrical toolbar) or Routing, Electrical, Create Route on the fly.
- 4 In the dialog box, click **OK**.
- 5 In the PropertyManager:
  - a) Select the end face of the cylinder on the plug for Select plane or face and a point <sup>(1)</sup>.
  - b) Select **Electrical** for **Select route type 1**.
  - c) Type **6** for **Nominal diameter**
- 6 Click 🕢.

The **Route Properties** PropertyManager opens.

7 Click 🕢.

A route sketch opens, where you can define the route. **CPoint1** appears in the FeatureManager design tree, with an external reference to **plug001**.

8 Close the sketch.



You can create other types of routes on the fly by clicking **Create Route on the fly** (Piping toolbar) or **Create Route on the fly** (Flexible Tubing toolbar).



## **Routing Component Wizard**

The **Routing Component Wizard** guides you through the steps to transform a part into a routing component such as a piping flange, electrical connector, etc. Tasks include:

- Adding a connection point
- Adding a mate reference
- Adding the part to a components library file
- Creating a connector end view drawing
- Creating required geometry

### To create a routing component:

- 1 Open Routing\connector01.sldprt.
- 2 Click Routing, Routing Tools, Routing Component Wizard.
- 3 In the wizard, select:
  - a) Electrical routing, then click Next.
  - b) End Connector, then click Next.

The wizard indicates that one connection point is required.

- 4 Click Add connection point.
- 5 In the PropertyManager:
  - a) Select the end face of the cylinder as shown for Select plane or face and a point  $\widehat{\mathbf{Q}}$ .



- b) Select Electrical for Select route type **L**.
- c) Type **14** for **Nominal diameter**
- 6 Click 🕢.
- 7 In the wizard, click **Next** until the **Mate References** page appears.
- 8 Click Add Mate Reference.
- 9 In the PropertyManager, select the front face of the connector for Primary Reference Entity .



10 Click 🕢.

11 In the wizard, click **Next** until the **Save Component to Library** page appears.

**12** Select **Create end connector illustrations for harness**.

13 Click **Next**. (Click **No** if asked if you want to save to component library.)

14 In the PropertyManager, select:

- a) The front face of the connector for End Face 🐚 .
- b) The top face for **Up Direction**.



## 15 Click 🕢.

An end view drawing, **connector01.slddrw**, is saved in the same folder as the connector part file.

16 Click Finish.

## **Wizards for Libraries and From-To Lists**

The following wizards guide you through the process of creating and editing libraries and from-to lists used in Electrical Routing:

- Component Library Wizard
- Cable/Wire Library Wizard
- From-To List Wizard



If the Piping or Flexible Tube toolbars are active, you cannot access the wizards.

#### To create a wire library:

- 1 Open a new assembly.
- 2 Click Routing, Electrical, Cable/Wire Library Wizard.
- 3 In the wizard, select **Create a new library**, then click **Next**.
- 4 In the list in the upper-right corner, select **Wire Library**.
- 5 Double-click the first cell under Name.
- 6 Add the following data. Press **Tab** to move from one cell to the next, or double-click a cell to edit it.

Name	Part Number	Description	OD	Color	SWColor	Min Bend Radius	Size/Gauge
20g KP	9984	20g yellow	2mm	Y		2mm	20
20g KN	9985	20g red	2mm	R		2mm	20

### 7 Click Save.

8 In the Save As dialog box:

- a) Browse to <*install\_dir*>\**samples**\**whatsnew**\**routing**.
- b) Type **mywires** for **File name**.
- c) Select XML files (\*.xml) for Save as type.
- d) Click Save.
- 9 Click Finish.

# **Routing Options Setup**

A new wizard facilitates setting Routing options.

### **To set Routing options:**

- 1 Click Routing, Routing Tools, Routing Options Setup.
- 2 In the dialog box, move from one option to another by doing any one of the following:
  - Selecting options from the tree.
  - Pressing the up and down arrows.
  - Clicking Next.
- 3 Set options, then click **OK**.



You can still set options by clicking **Options** [E] (Standard toolbar) and selecting **Routing**.

## Harnessing

## Menus

The Harnessing menus are now located within the Electrical menus. Click **Routing**, **Electrical**, **Harness**.

## **Multiple Harnesses**

You can create multiple harnesses within one route assembly and have them all appear within one harness drawing. Previously only one harness appeared on the drawing.

## **Harnessing Path**

Harnessing now uses the path set in Library folder on the Routing page in System Options.

### **End View Drawings**

You can create end view drawings of connectors with a wizard. See **Routing Component Wizard** on page 9-45.

# SolidWorks Task Scheduler

## **Design Checker**

You can schedule tasks for **Design Checker**  $\swarrow$  and specify the folders where reports and documents are saved.

# **Update PDMWorks**

Converting documents with **Update PDMWorks** appens SolidWorks only once and converts the selected documents in a single session. The option to reopen after conversion has been removed. Documents are opened in a resolved state (not lightweight). Be sure documents can be opened with all references resolved on the computer running SolidWorks Task Scheduler.

# SolidWorks Toolbox

# **Adding Parts**

The **Add My Parts Wizard** has been replaced with right-click and drag-and-drop functionality. You can create new folders and add your own parts to SolidWorks Toolbox as you do with other folders in the Design Library.

# **Chinese Standard**

The **GB** (Guo Biao or Chinese) standard has been added to SolidWorks Toolbox, Hole Wizard, and Smart Fasteners.

# **Configure Data**

The new **Configure Data** dialog box consolidates and simplifies the interface for customizing SolidWorks Toolbox. You can:

- Enable standards, fasteners, and sizes, add custom properties, set default options, and configure Smart Fasteners all from one dialog box. (Previously you accessed some items from **Tools**, **Options**, **Data Options** and others from **Toolbox**, **Browser Configuration**.)
- Establish a complete set of fasteners in advance. Populate your SolidWorks Toolbox files by clicking Create Configurations or Create Parts on the All Configurations tab that appears when you select components on the Contents tab.

Additionally, for multi-user environments, you now set the access control options in the **Configure Data** dialog box. New options include **Allow editing of Favorites** and selections to apply settings either separately for each user or globally for all users.

Access the new dialog box by clicking **Toolbox**, **Configure** or **Tools**, **Options**, **Hole Wizard/Toolbox**, **Configure**.

See Configure Data in the help.

## **Inserting Fasteners**

When you drag a Toolbox fastener into an assembly, you now set values in the PropertyManager instead of a dialog box.

## **Recreate Fasteners**

Assembly files now contain data necessary to recreate SolidWorks Toolbox fasteners if another user opens the assembly file and the appropriate part or configuration is not found in that user's Toolbox.

# **Toolbox File Path**

You can modify the path to your SolidWorks Toolbox files from the **Hole Wizard/Toolbox** page in **System Options**.

# SolidWorks Utilities

## Find and Replace Text in Annotations

You can find and replace text in the following annotations for part, assembly, and drawing documents:

- Balloons
- Datum feature symbols
- Datum targets
- Dimensions
- Geometric tolerances
- Hole callouts
- Notes
- Surface finish symbols
- Weld symbols

Click **Find and Replace Annotation** (Utilities toolbar) or **Utilities**, **Find and Replace Annotation**, then use the PropertyManager.

# Find, Modify, and Suppress Features Parameters

The **Find Features**, **Modify Features**, and **Suppress Features** utilities support new parameters for chamfers, extrudes, simple and variable fillets, and simple holes.

# **Format Painter**

You can copy visual properties of dimensions and annotations from part, assembly, and drawing documents to the same document or to other documents. You can copy properties such as arrow style, font style, height, and strikeout property. Click **Format Painter (**Utilities toolbar) or **Utilities**, **Format Painter**, then follow the dialogs.

# Simplify

The **Simplify** tility can find simplified features based on their volume in parts and assemblies. In the **Simplify** dialog, select **Volume Based**. To find simplified features by feature type, select **Feature Parameter**.

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