SolidWorks 2003

What's New

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

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Introduction

About This Book

This book highlights and helps you learn the new functionality in the SolidWorks® 2003 software. It introduces concepts and provides step-by-step examples for many of the new functions.

This book does not cover every detail of the new functions in this software release. For complete coverage of the new functions in the SolidWorks 2003 software, refer to the *SolidWorks Online User's Guide* by clicking **Help**, **SolidWorks Help Topics**.

Intended Audience

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

Late Changes

Due to printing deadlines, this book may not include all of the enhancements in the SolidWorks 2003 software. For enhancements that are not in this book, refer to the *SolidWorks Release Notes* by clicking **Help**, **SolidWorks Release Notes**. Also, refer to the **Overview of New Functionality in SolidWorks 2003** in the *SolidWorks Online User's Guide*.

Using This Book

Use this book in conjunction with the part, assembly, and drawing files provided. Read through this book from beginning to end, and open the proper part, assembly, or drawing document for each example. You can also use the Table of Contents or Index to locate topics of special interest to you.

To use the example files:

1 Install the SolidWorks 2003 software.

> If you have edited document templates, sheet formats, or Feature Palette™ items from a previous release, you should make a backup of the files before you install SolidWorks 2003.

2 Be sure to select the option to install the **Example Files**.

The example files for this book are placed in the *installation directory***samples****what's new** folder. For example, if you need to open **knob.sldprt**, then the full path to that example file is *installation directory***samples****what's new\knob.sldprt**.

3 Open the example files from the folder when instructed to do so.

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

Conventions Used in this Book

This book uses the following conventions:

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

Moving to SolidWorks 2003

System Requirements

For the most recent information about system requirements, refer to the *SolidWorks Read This First* document, which is included in the box that contains the SolidWorks software CDs.

Backup Copies of SolidWorks Files

We recommend that you save backup copies of all SolidWorks documents (parts, assemblies, and drawings) before opening them in SolidWorks 2003. These documents are automatically converted to SolidWorks 2003 format when opened. Once converted and saved, the documents are not accessible in earlier releases of the SolidWorks software.

Converting Older SolidWorks Files to SolidWorks 2003

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

The SolidWorks Conversion Wizard provides a way for you to automatically convert all of your SolidWorks files from an earlier version to the SolidWorks 2003 format. Depending on how many files you have, the conversion process may take a while, but once complete, the files open faster.

To access the Conversion Wizard, click the Microsoft® **Start** button, select **Programs**, **SolidWorks 2003**, **SolidWorks Tools**. Click **Conversion Wizard**.

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

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

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

SolidWorks Service Packs

You can take advantage of SolidWorks service packs that are regularly posted on the SolidWorks Web site. These service packs contain software updates and enhancements to the SolidWorks 2003 software. To check for a new service pack, click **Help, Service Packs**, and click **Check**. Select the check box if you want the software to automatically check the SolidWorks Web site for a new service pack once a week.

1

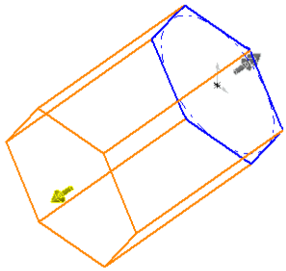
SolidWorks Fundamentals

This chapter describes enhancements to fundamental topics in the following areas:

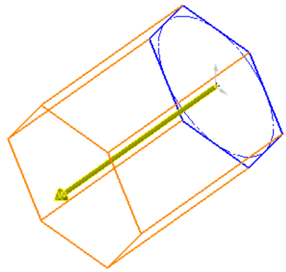
- Handles
- Shortcut menus
- Documentation
- FeatureManager design tree
- Parent/child relationships
- Performance
- Display
- Opening documents
- Options
- Reference triad
- Description labels
- Printing
- 3D Content Central
- Application Programming Interface
- Macros
- Configurations

Handles

When you create an extrude, cut-extrude, or base-flange, you must drag the *handle* to the desired extrude depth, and the handle spans that length.



SolidWorks 2001Plus:
Handle does not span the length of extrude



SolidWorks 2003:
Handle spans the length of extrude

Shortcut Menus

The shortcut menus have been reduced in size so it is easier to find the command you need. You can access the “more commands menu” by either selecting the double-down arrows in the shortcut menu, or by pausing the pointer over the double-down arrows. When you select the double-down arrows, the menu expands to offer more menu items.

Documentation

Design Portfolio

A new example is available in the Design Portfolio. The model in the example was the Model Mania part at the 2002 SolidWorks World Conference. This example showcases a layout sketch, Neutral Plane and Parting Line draft features, and two approaches to the creation of the pocket in the cross link.

Help for AutoCAD Users

Help for AutoCAD Users supports users in the transition from 2D AutoCAD® to 3D SolidWorks. It compares terms and concepts, explains SolidWorks approaches to design, and provides links into SolidWorks Help, tutorials, and other resources. You can find **Help for AutoCAD Users** on the **Help** menu in the SolidWorks software.

Online Tutorial

More Lessons

The *Online Tutorial* has been expanded to include over 20 lessons that cover SolidWorks topics and several add-in applications. These step-by-step lessons are available by clicking **Help**, **Online Tutorial**.

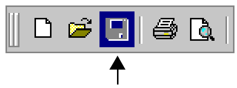
Automation

You can execute some SolidWorks commands directly from the *Online Tutorial*.

For example, if you need to open a sample part to complete a tutorial, you can click a link, and SolidWorks automatically opens the part for you. This way, you do not have to browse directories to find a part.

Toolbar Button Highlighting

When you click a toolbar button in the *Online Tutorial* window, the corresponding button highlights in the SolidWorks software. This helps you locate the toolbar button in the SolidWorks window.


Highlighted button

If the toolbar is not displayed in the SolidWorks window, SolidWorks activates the toolbar and highlights the toolbar button. If the toolbar is visible but the button is not, SolidWorks adds the button to the toolbar and highlights it.

You can customize the highlight color in Microsoft Windows® by clicking **Start**, **Settings**, **Control Panel**, **Display**, **Appearance**. The highlight color is the same color used in the **Active Window** setting.

Introducing SolidWorks

Introducing SolidWorks is a book intended for new SolidWorks users. The book introduces concepts and design processes in a high-level approach.

Introducing SolidWorks does not give step-by-step procedures on how to create a model. Instead, it guides you through the design process by illustrating how to plan models, make parts, build assemblies, then create drawings.

This book is shipped to new SolidWorks customers only, and replaces the *Getting Started* book.

FeatureManager Design Tree

In SolidWorks 2003, you can customize the display of items in the FeatureManager® design tree. For example, you can show feature descriptions, component descriptions, and so on. When you specify the appearance of the FeatureManager design tree, the settings are saved within the document; they are not system settings and do not apply to all documents.

Show Feature's Description

In part, assembly, or drawing documents, you can display feature descriptions in the FeatureManager design tree. In previous SolidWorks releases, only the feature name was displayed.

By default, feature names and feature descriptions are the same. You must specify new feature descriptions so they appear in the FeatureManager design tree.

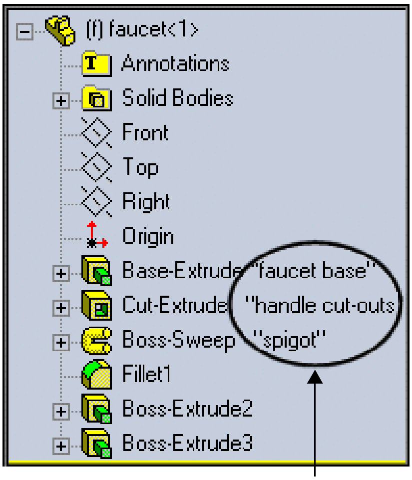
To change a feature description in a part or assembly, right-click the feature, select **Feature Properties**, and type a new **Description**.

To change a feature description in a drawing, you must change the feature description in the part or assembly file; then the description appears in the drawing.

To show feature descriptions:

1. Open **\faucet\faucet_assembly.sldasm**.
2. Click ⊞ to expand **faucet<1>**.
3. In the FeatureManager design tree, right-click **faucet_assembly**, and select **Tree Display**, **Show Feature's Description**.

   The FeatureManager design tree displays the feature descriptions.
4. Keep this assembly open for the next procedure.



Feature descriptions

You must select either **Show Feature's Name** or **Show Feature's Description**. Both options can be selected at the same time, but they cannot both be deselected.

Show Component's Description

In part, assembly, or drawing documents, you can display component descriptions in the FeatureManager design tree. Component descriptions can also be displayed in the ConfigurationManager of parts and assemblies.

By default, component names and component descriptions are the same. You must specify new component descriptions so they appear in the FeatureManager design tree.

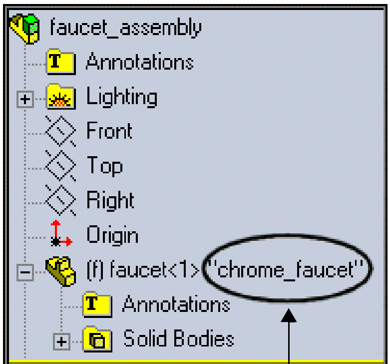
To change a component's description, you must open the component *part* file, then do the following:

1. Click **File**, **Properties**.
2. On the **Custom** tab in the **Summary Information** dialog box, select **Description** in the **Name** column.
3. Type a description name in the **Value** box.
4. Click **Modify**, then click **OK**.

To show component descriptions:

1. Open **\faucet\faucet_assembly.sldasm** if you do not have it open from the previous procedure.
2. In the FeatureManager design tree, right-click **faucet_assembly**, and select **Tree Display**, **Show Component's Description**.

   The FeatureManager design tree displays the component descriptions.
3. Keep this assembly open for the next procedure.



Component description

You must select either **Show Component's Name** or **Show Component's Description**. Both options can be selected at the same time, but they cannot both be deselected.

Show Component's Configuration Name

In part, assembly, or drawing documents, you can display component configuration names in the FeatureManager design tree.

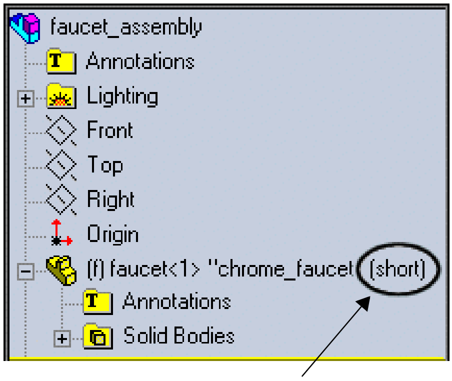
To show a component's configuration name:

1 Open **\faucet\faucet_assembly.sldasm** if you do not have it open from the previous procedure.

2 In the FeatureManager design tree, right-click **faucet_assembly**, and select **Tree Display**, **Show Component's Configuration Name**.

The FeatureManager design tree displays the component configuration name.

3 Keep this assembly open for the next procedure.



Component configuration name

Show Component's Configuration Description

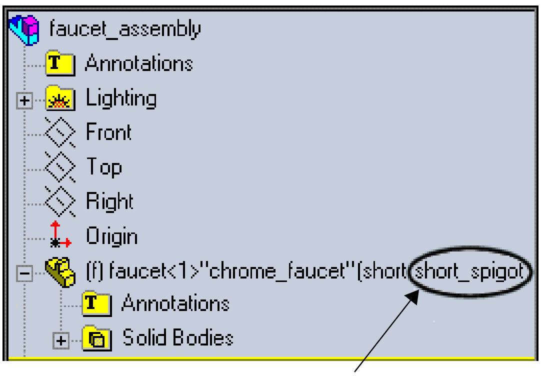
In part, assembly, or drawing documents, you can display component configuration descriptions in the FeatureManager design tree. Component configuration descriptions can also be displayed in the ConfigurationManager of parts and assemblies.

To show a component's configuration description:

1 Open **\faucet\faucet_assembly.sldasm** if you do not have it open from the previous procedure.

2 In the FeatureManager design tree, right-click **faucet_assembly**, and select **Tree Display**, **Show Component's Configuration Description**.

The FeatureManager design tree displays the component configuration descriptions.



Component configuration description

Folders

In part or assembly documents, you can add custom folders to the FeatureManager design tree. You can rename new folders and drag additional features into the new folders. This helps to reduce the length of the FeatureManager design tree.

Depending on how you create a folder, you can insert features automatically or manually.

You can place any set of *continuous* features or components into an individual folder. You cannot use **Ctrl** to select non-continuous features. This way, parent-child relationships are maintained.

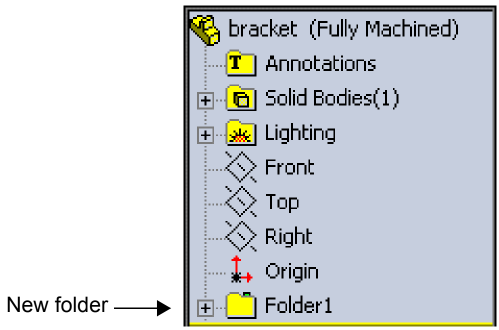
Folders cannot be added to existing folders.

To create a new folder and insert features automatically:

1. Open **bracket.sldprt**.
2. In the FeatureManager design tree, hold down the **Shift** key, then click **Base-Extrude** and **Boss-Extrude4**.

   This selects all of the features between **Base-Extrude** and **Boss-Extrude4**.
3. Right-click any of the selected features in the FeatureManager design tree and select **Add to New Folder**.

   A new folder, **Folder1**, appears in the FeatureManager design tree, which includes all of the features you selected in step 2.
4. Keep **bracket.sldprt** open for the next procedure.



To create a new folder and insert features manually:

1. Open **bracket.sldprt** if you do not have it open from the previous procedure.
2. In the FeatureManager design tree, right-click **Hole1** and select **Create New Folder**.

   A new folder, **Folder2**, appears in the FeatureManager design tree. Notice that **Hole1** is not in the folder.
3. Drag **Hole1** onto the folder name, **Folder2**.

   The pointer changes to .
4. Release the mouse button.

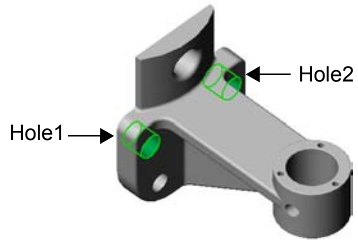
   The feature is inserted into the folder.
5. Drag **Hole2** onto the folder name, **Folder2**, but do not drop the feature.

   The folder expands, and the pointer changes to .

6 Release the mouse button.

The feature is inserted into the folder.

7 Keep **bracket.sldprt** open for the next procedure.



> When you click a folder name in the FeatureManager design tree, all of the features in the folder are highlighted in the model.

To remove features from a folder:

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

2 In **Folder2**, drag **Hole2** onto the folder icon , but do *not* release the mouse button.

The pointer changes to .

3 Release the mouse button.

The feature is removed from the folder, and appears just below the folder.

Rollback

Rollback Bar Options

When you use the **Rollback** function in the FeatureManager design tree, you can choose three new options: **Roll Forward**, **Roll to Previous**, and **Roll to End**.

To move the rollback bar with Roll Forward, Roll to Previous, and Roll to End:

1 Open **cog.sldprt**.

cog.sldprt

2 In the FeatureManager design tree, right-click **Boss-Extrude3** and select **Rollback**.

The model rolls back to its state prior to the boss-extrude feature.

Rollback

3 Repeat step 2, but select **Roll Forward**.

The model rolls forward one feature, to the boss-extrude.

Roll Forward

4 Right-click any feature name in the FeatureManager design tree, and select **Roll to Previous**.

The model rolls back to its state in step 2.

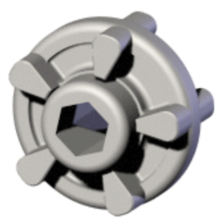
If a message appears that tells you **Sketch5** will be temporarily unabsorbed for editing purposes, click **OK**.

When you select **Roll to Previous**, the model rolls back to the previous state, not to the previous feature.

Roll to Previous

5 Repeat step 4, but select **Roll to End**.

The model returns to its original state.



Roll to End

Rolling Back Absorbed Features

You can use the **Rollback** function to roll back to features that are absorbed in their parent features. This is especially helpful if you want to add sketches to loft or sweep features.

To rollback to absorbed features:

1 Open **\faucet\faucet.sldprt**.

2 In the FeatureManager design tree, do the following:

a) Click ⊞ to expand **Boss-Sweep1**.

b) Right-click **Sketch3**, and select **Rollback**.

A message appears that tells you **Sketch3** and **Sketch4** will be temporarily unabsorbed for editing purposes.

3 Click **OK**.

4 Drag the rollback bar below **Sketch3**.

The sketch is unabsorbed and available to edit.

Rebuild

When you reorder an imported feature or any other feature with no parent or child, the part does not rebuild - only the FeatureManager design tree refreshes.

Curve Feature Icons

When you create curves with the **3D Curve** (**Curve Through Reference Points**) and **Curve Through Free Points** tools, feature-specific icons that match the Curve toolbar icons appear in the FeatureManager design tree. Previously, the **Curve Through Free Points** icon was used in the FeatureManager design tree for both of these curve types.

Parent/Child Relationships

The **Parent/Child Relationships** dialog box has been enhanced for ease of use. Right-click a feature in the graphics area or FeatureManager design tree, and select **Parent/Child**. In the **Parent/Child Relationships** dialog box, you can do the following:

- Right-click an item in the **Parents** or **Children** list, and select an option from the shortcut menu, such as **Edit Sketch** and **Edit Definition**.
- Resize the dialog box to see all of the items listed.

Performance

Parts

Feature interaction performance (rolling a model backward or forward, editing the definition of a feature, canceling an edit definition, editing a sketch, and so on) in parts is improved. Interaction now takes less time, depending on model complexity and your system configuration. The performance enhancement is noticeable on most feature edits; it is often faster on complex parts as well.

The performance improvement applies under the following conditions:

- The feature must be rebuilt at least once during the current SolidWorks session.
- No model changes have been made to the target feature or previous features in the FeatureManager design tree since the last rebuild.

Previously, interaction times varied much more from feature to feature and part to part. The interaction time was dependent on the complexity of the part and the complexity of the features in the region of the feature being edited.

This improvement affects any operation that involves part regeneration, such as reorder, rebuild, and suppress, as these operations involve an internal rollback.

Graphics Area

When you add a feature to complex SolidWorks models, the graphics area regenerates only on faces that are affected by the new feature. The entire graphics area does not regenerate, as in previous SolidWorks versions.

When SolidWorks regenerates the graphics area where faces are affected by a new feature, your model may have small gaps where faces intersect. Click **Rebuild** to regenerate the entire graphics area and eliminate the gaps.

Display

Hidden Lines Visible has replaced **Hidden In Gray** to more accurately describe the view mode.

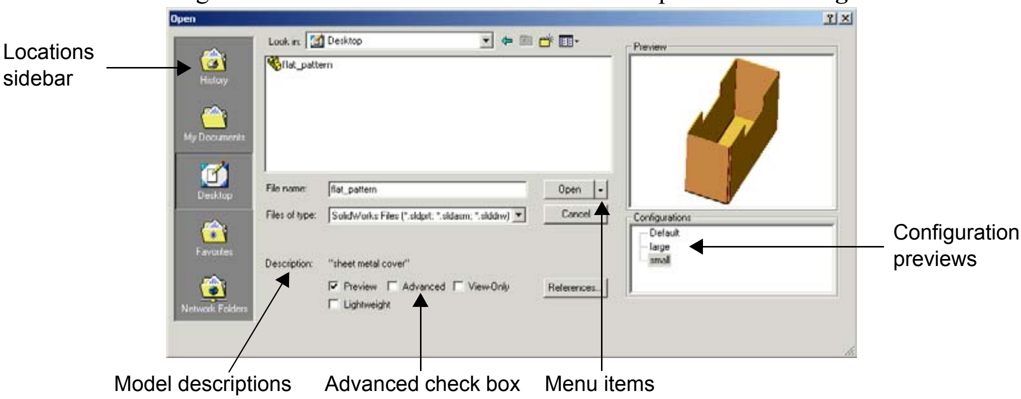
Opening Documents

When you click **File**, **Open**, the **Open** dialog box displays several new items:

- **Locations sidebar**. Helps you navigate to the location of your document. The sidebar also appears in the **Save As** dialog box.

  > The locations sidebar is available in Microsoft Windows 2000, XP, and Me operating systems only.

- **Configuration previews**. Displays a preview image of a configuration that you select. In the **Configurations** box, double-click a configuration to open the configuration. You no longer have to select the **Configure** check box to open a specified configuration of the model. See **Configuration Previews in the Open Dialog Box** on page 1-20 for more information.
- **Model descriptions**. Displays the document description. See **Custom Properties** on page 1-15 for more information.
- **Menu items**. Click next to the **Open** button to access additional menu items. The menu items include **Open as Read-Only** and **Add to Favorites** options. When you select **Add to Favorites**, a shortcut to the selected document is added to your **Favorites** folder. The **Add to Favorites** option is also available in the **Save As** dialog box.
- **Advanced**. Accesses the **Configure Document** dialog box after you click **Open**. In this dialog box, you can specify part configurations for assemblies and create new configurations for assemblies. This check box replaces the **Configure** check box.



Options

General

The **Edit design tables in a separate window** option in **Tools**, **Options**, **System Options**, **General**, has been removed. To edit a design table in a separate window, click **Edit**, **Design Table**, **Edit Table in New Window**.

If you inserted a design table in your model, you can also right-click **Design Table** in the FeatureManager design tree and select **Edit Table in New Window**.

Drawings

The **Automatic update of BOM** option has been moved from **Tools**, **Options**, **System Options**, **Drawings** to **Tools**, **Options**, **Document Properties**, **Detailing**. The option can now be applied on a per document basis.

File Locations

New File Types

You can set the default open and save locations for file types, including palette features, blocks, and bend tables. In addition, the following file types have been added to the **Tools**, **Options**, **System Options**, **File Locations** list:

- BOM Templates
- Custom Property File
- Dimension Favorites
- Hole Callout Format File
- Library Feature Files
- Macros
- Macro Feature Files
- Palette Assemblies
- SolidWorks Journal File

Adding New Folders

When you click **Add** to add a new file location in the **System Options-File Locations** dialog box, you can now create a new folder. In the **Browse For Folder** dialog box, click **New Folder** to create a new folder.

You can also add a new folder when you click **File**, **Find References**, **Copy files** in a parent document.

Backups

There is a new option to save backup files in the same location as the original file. Backup files use the naming convention: **Backup of** <*document name*>**.sld***.

To save backup files in the same location as the original file:

1. Click **Tools**, **Options**.
2. On the **System Options** tab, click **Backups**.
3. Select the **Save backup files in the same location as the original** check box.
4. Click **OK**.

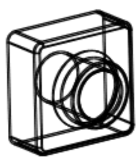
Display/Selection

Anti-Alias Edges

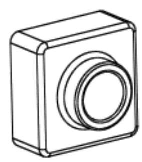
The **Anti-alias HLR edges in shaded and fast HLR/HLG modes** option has been renamed to **Anti-alias edges**. Anti-alias edges now apply to **Wireframe**, **Hidden Lines Visible**, and **Hidden Lines Removed** view modes. Previously, anti-alias edges applied to **Shaded** view mode only.

To turn on anti-alias edges:

1. Click **Tools**, **Options**.
2. On the **System Options** tab, click **Display/Selection**.
3. Select the **Anti-alias edges** check box.
4. Click **OK**.


Wireframe


Hidden Lines Visible


Hidden Lines Removed

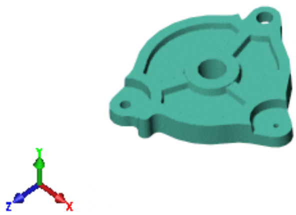
Repaint After Selection in HLR

The **Repaint after selection in HLR** option in **Tools**, **Options**, **System Options**, **Display/Selection**, has been removed.

High Quality Display of Interfering Bodies in HLR/HLG

The **High quality display of interfering bodies in HLR/HLG** option in **Tools**, **Options**, **System Options**, **Display/Selection**, has been removed. This option is always enabled now, and does not affect system performance.

Reference Triad



The graphics area now displays a reference triad that helps orient you when viewing models. The reference triad is turned on by default.

The reference triad is for display purposes only. You cannot select it or use it as an inference point.

To turn the reference triad off:

1 Click **Tools**, **Options**.

2 On the **System Options** tab, click **Display/Selection**.

3 Clear the **Display reference triad** check box.

4 Click **OK**.

You can change the colors of the reference triad in **Tools**, **Options**, **System Options**, **Colors**. Select any of the three axes to change:

- **X axis of Reference Triad**
- **Y axis of Reference Triad**
- **Z axis of Reference Triad**

Description Labels

Custom Properties

You can specify a description name for models in the **Summary Information** dialog box. When you specify a description, the name appears in the FeatureManager design tree (see **Show Component's Description** on page 1-5) and in the **Open** and **Save As** dialog boxes (see **Opening Documents** on page 1-12).

To specify a description name:

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

2 In the **Open** dialog box, click **lofted_bend.sldprt**, but do *not* open it.

Notice the label, **Description**, which is set to **<None>**.

3 Click **Open**.

4 Click **File, Properties**.

The **Summary Information** dialog box appears.

5 On the **Custom** tab, select **Description** from the **Name** list.

6 Type **sheet metal part** in the **Value** box.

7 Click **Modify**.

The new **Value** appears in the **Properties** box.

8 Click **OK**, then click **Save** .

9 Repeat steps 1 and 2, and notice that **Description** is set to **sheet metal part**.

10 Keep **lofted_bend.sldprt** open for the next procedure.

You can also specify a description name in the **Description** box of the **Save As** dialog box.

Custom Description Labels

You can define a custom description label.

For example, the **Open** dialog box has a **Description** label that displays the model description (see **Custom Properties** for more information). Instead of displaying the **Description** label, you can display a label that you specify.

To define a custom description label:

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

2 Click **File**, **Properties**.

The **Summary Information** dialog box appears.

3 On the **Custom** tab, do the following:

a) Select a **Name** from the list, or type a name that you want to use.

b) Type a **Value** in the box.

c) Click **Add**, then click **OK**.

4 Click **Tools**, **Options**.

5 On the **System Options** tab, click **General**.

6 In the **Custom property used as component description** box, click or type the **Name** you used in step 3a.

7 Click **OK**.

8 Save the part.

9 Click **Open**, and click the part name, but do *not* open it.

The **Open** dialog box uses the description name and value you specified.

In new parts, you do not have to complete steps 1 through 3. Instead, you can go directly to the **System Options-General** dialog box to define a custom description label.

Printing

There are new options available when you print a document. Click **File**, **Page Setup** to use the following options:

- **Use system settings**. Prints the document with system print settings, and allows you to change these settings as needed.
- **Use this document's settings**. Overrides system print settings with the settings saved in the current document. Any changes to the settings are applied to the current document only. The system settings remain unchanged.

In subsequent files that you print, the SolidWorks software remembers the setting that you last selected.

3D Content Central

Use **3D Content Central**, an online CAD resource, to view and download models from catalogs and other resources. You can also upload models to share with others. Click **Tools**, **3D Content Central** to register for free.

Application Programming Interface

There are several enhancements to the Application Programming Interface (API) in SolidWorks 2003. To find out more information about these enhancements, click **Help**, **SolidWorks API Help Topics**, and read the *SolidWorks 2003 API Release Notes*.

ActiveX

FeatureManager Design Tree

You can insert ActiveX controls in the FeatureManager design tree and in the model view window for tighter integration with add-ins.

PropertyManager

You can insert ActiveX controls in the PropertyManager.

Callouts and Selection Colors

You can create callouts in the graphics area and set the selection colors in the PropertyManager.

Macro Features

You can create macro features in the API. Macro features create user-defined features such as extrudes and assistant tools.

When you use a macro feature, SolidWorks inserts the feature in the FeatureManager design tree. If you rebuild the parent feature of the macro feature, SolidWorks also rebuilds the macro feature.

Multibody Parts

The API supports multibody parts. See Chapter 5, “Multibody Parts.” for more information.

Opening Multiple VBA Projects

Each time you open an additional Visual Basic for Applications (VBA) project, the project is added to the Project Explorer window in VBA. This allows you to have multiple projects open at the same time.

Macros

New Macros

You can now create a new macro from the Macro toolbar or from a menu item. Previously, if you wanted to create a new macro, you had to record a macro, then modify the code in your macro editing application.

Creating a new macro is different from recording a macro. When you create a *new* macro, you program the macro directly from your macro editing application. When you *record* a macro, you create the macro from within the SolidWorks software.

To create a new macro:

1 Click **New Macro** on the Macro toolbar, or click **Tools**, **Macro**, **New**.

The **New Macro** dialog box appears.

2 Type a **File name**, and click **Save**.

Your macro editing application opens and you can create a new macro.

Running Specified Methods and Creating Custom Macro Buttons

You can now specify which method you want a macro to run. Previously, macros ran the last method recorded in a macro.

When you create a macro, you can now assign a bitmap to the macro button on the toolbar. The SolidWorks software includes sample bitmaps, or you can create your own bitmap.

If you create a bitmap to assign to a macro button, the bitmap must meet the following requirements:

- Dimension = 16 x 16 pixels
- Color = 16 colors
- Background color = white

To run a specified macro method and create a custom macro button:

1 In a new part document, click **Tools**, **Customize**.

The **Customize** dialog box appears.

2 On the **Commands** tab, select **Macro** from the **Categories** list.

3 Under **Buttons**, drag the **Macro** button to any toolbar in the SolidWorks window.

The **Customize Macro Button** dialog box appears.

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

- Click **Choose Image**.
- In the **Icon path** dialog box, select **bell.bmp**, then click **Open**.
- Type a **ToolTip** and **Prompt** message, if desired.

The **Prompt** message is the status bar information displayed in the bottom left corner of the SolidWorks window.

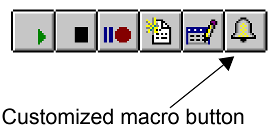
---

5 Under **Action**, do the following:

a) Click ... and open **Macro1.swp**.

b) Select **Macro11.Main** from the **Method** list. This runs a specified method contained within the macro.

6 Click **OK**.



7 Click **OK** again to close the **Customize** dialog box.

The macro is added to the toolbar with a customized button.

8 Click the button you added in step 7.

The selected macro method creates a new part with an extrude feature.

Configurations

Configuration Previews in the Open Dialog Box

Configuration previews of a part or assembly are displayed in the **Preview** box of the **Open** dialog box. The preview appears in the same view orientation as it does in the graphics area. In the **Configurations** box, double-click a configuration to open the configuration.

To specify a configuration preview:

1. Click **File**, **Open**.
2. In the **Open** dialog box, do the following:
   - Select the **Preview** check box.
   - Click **cog.sldprt**, but do *not* open it.
3. Under **Configurations**, click **Default**, then click **Simplified**.

   The **Preview** box updates with a preview of the configuration.

   If the **Preview** box does not display the configuration preview, you must open the document, then open each configuration, and save the document.

   The next time you open the document, the **Preview** box shows the selected preview.

4. Double-click **Simplified**.

   The part opens in the **Simplified** configuration.
5. Keep **cog.sldprt** open for the next procedure.

Configuration Previews in the PropertyManager

You can display configuration previews of a part or assembly in the PropertyManager. This way, you do not have to open a configuration to see it, which saves time in complex parts or assemblies.

To display a configuration preview in the PropertyManager:

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

2 In the ConfigurationManager, do the following:

a) Right-click **Default**, and select **Show Preview**.

The FeatureManager design tree automatically splits, and the **Default** configuration is displayed in the PropertyManager.

b) Click **Simplified**.

The **Simplified** configuration is displayed in the PropertyManager.

3 Click anywhere in the graphics area to hide the previews.

Show Configuration's Description

In part or assembly documents, you can display configuration descriptions in the ConfigurationManager.

By default, configuration names and configuration descriptions are the same. You must specify new configuration descriptions so they appear in the ConfigurationManager.

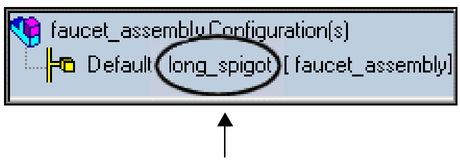
To change a configuration's description, right-click the configuration in the ConfigurationManager, select **Properties**, and type a new **Description**.

To show configuration descriptions:

1 Open **\faucet\faucet_assembly.sldasm**.

2 In the ConfigurationManager, right-click **faucet_assembly Configuration(s)**, and select **Tree Display**, **Show Configuration's Description**.

The ConfigurationManager displays the configuration descriptions.



Configuration description

Face Colors

You can now set individual face colors for each configuration.

2

Sketching

This chapter describes enhancements to sketching in the following areas:

- Contour selection
- Silhouettes
- Offset entities tool
- Autodimensions
- Splines
- Line Format Toolbar

Contour Selection

You can now select contours and apply features to them. This way, you can use a partial sketch to create features.

To select and extrude contours:

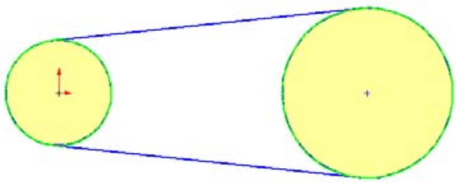
1 Open **autodim_sketch.sldprt**.

2 Edit **Sketch1**.

3 Right-click in the graphics area and select **Contour Select Tool**.

The pointer changes to .

4 Hold down **Ctrl** and select the two circles.



As you drag the pointer over the circles, the color of the contours changes to pink, then yellow when you select them.

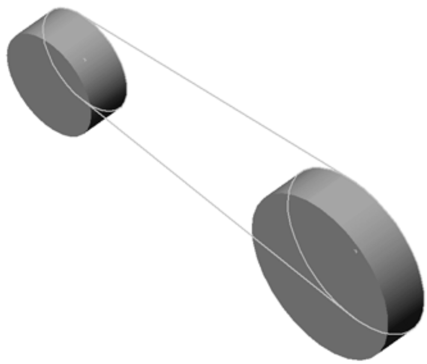
5 Click **Extruded Boss/Base** .

6 Set **Depth** to 20mm.

Notice the **Selected Contours** box. It lists the selected contours to extrude.

7 Click **OK** .

The selected contours extrude.



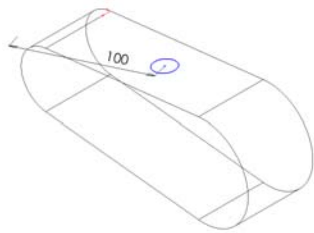
Silhouettes

You can now select and reference silhouette vertices. This is useful when you dimension sketches.

To dimension silhouette vertices:

1. Open **sketch_silhouette.sldprt**.
2. Edit the **Hole** sketch.
3. Click **Normal To** on the Standard Views toolbar.

4. Add a dimension between the center of the circle and silhouette vertex.



Offset Entities Tool

Offset Entities has been enhanced so that the preview of the offset entity does not follow the pointer when the mouse button is not down. Hold down the mouse button and drag the pointer to see a dynamic preview; the entity is created when you release the mouse button.

Autodimensions

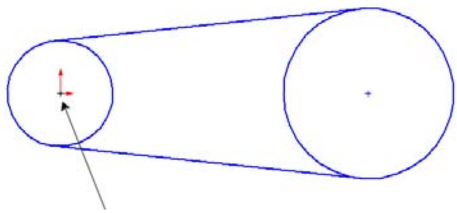
You can now dimension sketches automatically with the **Autodimension** tool. You can select among chain, baseline, centerline, and ordinate dimensions.

To autodimension a sketch:

1. Open **autodim_sketch.sldprt**.
2. Edit **Sketch1**.
3. Click **Autodimension Sketch** on the Sketch Relations toolbar, or click **Tools**, **Dimensions**, **Autodimension Sketch**.

   The **Autodimension** PropertyManager appears.
4. Under **Entities to dimension**, click **All entities in sketch**.
5. Under **Horizontal dimensions**, do the following:
   - Set **Horizontal Dimensioning Scheme** to **Baseline**.
   - In the **Point or Vertical Line on Baseline** box, select the point shown below.

   This sets the vertical point of origin of the dimensions.

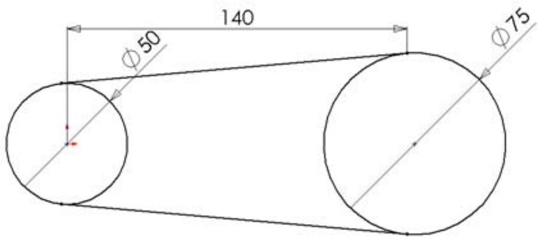


6. Under **Vertical dimensions**, do the following:
   - Set **Vertical Dimensioning Scheme** to **Baseline**.
   - In **Point or Horizontal Line on Baseline**, select the same point as in step 5.

   This sets the horizontal point of origin of the dimensions.
7. Click **OK**.

   The sketch is fully defined.

   You adjust the size, font, and distance of the dimension text in **Document Properties**, under **Detailing**, **Annotations Font**, **Dimension**.
8. Close the part.



Splines

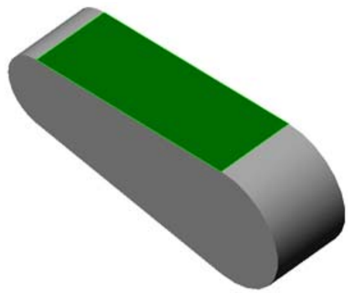
Splines have been enhanced so that you can convert sketch segments (lines and arcs) into a spline, or **Fit Spline**. There are two new diagnostic tools to support the use of splines: **Show Minimum Radius** and **Show Inflection Points**.

Fit Spline

You can now fit a spline to sketch segments to make a smooth edge. A spline creates a single curve from multiple sketch segments, and SolidWorks converts the sketch segments into a spline. After you perform the operation, SolidWorks displays a spline and construction geometry of the segments. This is useful when you use lofts and sweeps.

After you fit the sketch segments to a spline, altering the construction geometry segments does not update the spline. The spline and construction geometry are separate entities. You can select to delete the construction geometry on the **Fit Spline** PropertyManager.

To fit a spline to sketch segments:



1 Open **sketch_spline.sldprt**.

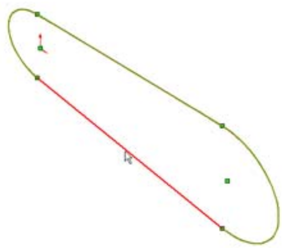
2 Click the top face of the extrusion, as shown.

Notice the face that highlights.

3 In the FeatureManager design tree, click ⊞ to expand **Extrude1**, then edit **Sketch1**.

4 Click **Fit Spline** on the Sketch Tools toolbar, or click **Tools**, **Sketch Tools**, **Fit Spline**.

The **Fit Spline** PropertyManager appears.



5 In the graphics area, select each sketch segment; there are two lines and two arcs.

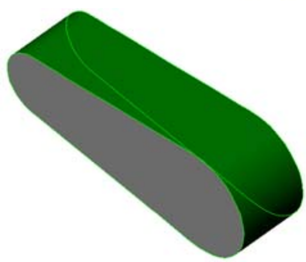
6 Click **OK**.

The segments are now one spline.

7 Close the sketch.

8 Click the top face of the extrusion.

The continuous curved face highlights.



Minimum Radius and Inflection Points Tools

There are two new diagnostic tools for use with splines: **Show Minimum Radius** and **Show Inflection Points**. The **Show Minimum Radius** tool displays the radial measurement of the curve with the smallest radius. The **Show Inflection Points** tool displays all points where the concavity of the curve changes.

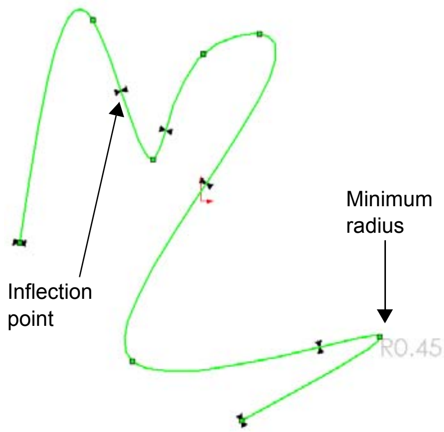
The data points for both tools are displayed as references, however, if you manipulate the spline with the spline points, the minimum radius and inflection points dynamically update.

To show the minimum radius and inflection points of a spline:

1. Open **spline_tools.sldprt**.
2. Edit **Sketch1**.
3. Select the spline.
4. Right-click and select **Show Minimum Radius**.

   The minimum radius point is shown with the radius.
5. Right-click and select **Show Inflection Points**.

   Inflections points are shown.

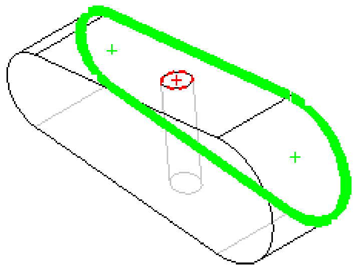


You can manipulate splines with the data provided by these tools to get the desired shape and attributes.

Line Format Toolbar

With a sketch selected, you can now select tools from the Line Format toolbar. You can use the **Line Color**, **Line Thickness**, and **Line Style** tools.

To use the Line Format tools in a sketch:



1. Open **sketch_silhouette.sldprt**.
2. Select the sketch for **Extrude1**, called **base**.
3. Click **Line Thickness** on the Line Format toolbar.
4. Select a thickness and click **OK**.
5. Select the sketch for **Cut-Extrude1**, called **Hole**.
6. Click **Line Style** on the Line Format toolbar.
7. Select a style from the list.
8. Close the part.

3

Features and Surfaces

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

- General enhancements
- Multibody-related features
- Fillet previews
- Full round fillets
- Sketch reuse in features
- Delete faces
- Cut with plane
- Multiple surfaces
- Untrim surface
- Surface fill options
- Deviation analysis

General Enhancements

General enhancements to features and surfaces include:

- Performance improvements for extrude and sweep features
- User interface options
- Changes to the mirror function

Extrude Enhancements

Improvements to the extrude feature include:

- Sketch vertices are now valid selections for **Up to Vertex** extrusions.
- If you double-click a surface, the extrusion changes to **Up to Surface** and selects that surface as the surface to extrude.
- For multibodies, you can select the **Merge results** check box (see **Bridging** on page 5-3) to bridge separate bodies into a single body.You can also select **Merge results** when you edit an extrude feature.
- For multibodies, you can extrude **Up to Body** (see **Local Operations** on page 5-5). The **Up to Body** option is also useful when extruding within an assembly or in mold parts.

Mirror Feature

As part of the multibody environment, the mirror feature includes the following changes:

- You can now mirror around a plane.
- The **Mirror All** option was removed from the menu. To access all mirror functions, click , or **Insert, Pattern/Mirror, Mirror.** Under **Mirror Face/Plane**, select the plane or planar face around which to mirror the model.
- Under **Options**, you can select to **Merge solids** or **Knit surfaces**.

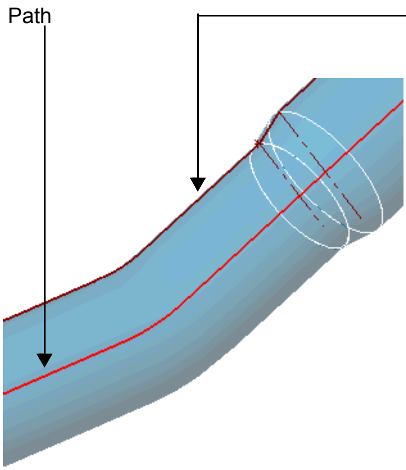
In the Mirror PropertyManager, select only **Bodies to Mirror** when you use the mirror feature to create a separate, unattached body. See **Symmetry Modeling** on page 5-7.

Sweeps with Guide Curves

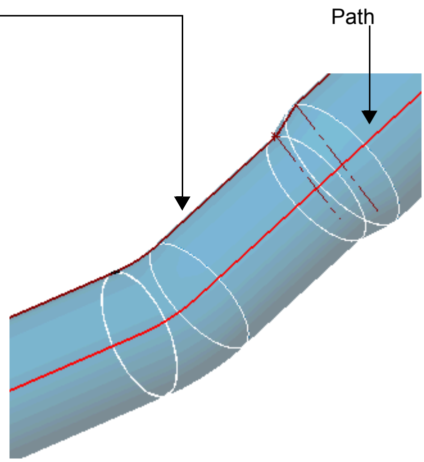
A new option allows you to clear the **Merge smooth faces** check box for sweeps with guide curves. Previously, there was no check box, and all sweeps merged smooth faces by default. Clearing the **Merge smooth faces** check box results in the following:

- The performance of sweeps with guide curves improves. Sweeps generate faster and merge between adjacent geometry and edges.
- The swept body is segmented at all points where the guide curve or the path is not curvature continuous (see the example below). Consequently, the lines and arcs in the guide curves are more accurately matched.

When you clear the **Merge smooth faces** check box, the potential exists that some features created later might fail due to changed geometry.



Merge smooth faces check box *checked*



Merge smooth faces check box *cleared*

Scale Surface

You can now scale surface bodies in the same way as you can scale solid bodies. The user interface and the options are the same as those used for scaling solid bodies.

User Interface

New buttons added to the Features toolbar include the following:

- **Combine**. Combine two or more solid bodies (see **Symmetry Modeling** on page 5-7).
- **Deletes Solid/Surface**. Delete a solid or surface.
- **Table Driven Pattern**. Create table driven patterns.
- **Sketch Driven Pattern**. Create sketch driven patterns.
- **Imported Geometry**. Insert a solid or a surface into an existing document (see **Insert Imported Geometry** on page 9-5).
- **Move/Copy Bodies**. Formerly **Move/Copy Surface** on the Surfaces toolbar.

New button added to the Surfaces toolbar includes the following:

- **Untrim Surface**. Extends edges and fills holes (see **Untrim Surface** on page 3-12).

Multibody-Related Features

The following feature enhancements result from the new multibody environment.

- Move/Copy
- Feature scope

See also **Extrude Enhancements** on page 3-2 for multibody-related extrude enhancements.

Move/Copy

You can use **Move/Copy** with all surface bodies and with solid bodies in a multibody environment. The user interface is unchanged. See **Surface Move/Copy** on page 5-13.

Feature Scope

You can set the scope of what bodies are affected. This allows you to define design intent in the modelling process, and to manage performance in parts with a large numbers of bodies:

- Extrude boss and cut (including thin features)
- Revolve boss and cut (including thin features)
- Sweep boss and cut (including thin features)
- Boss and cut thicken
- Surface cut
- Cavity

See **Feature Scope** on page 5-12.

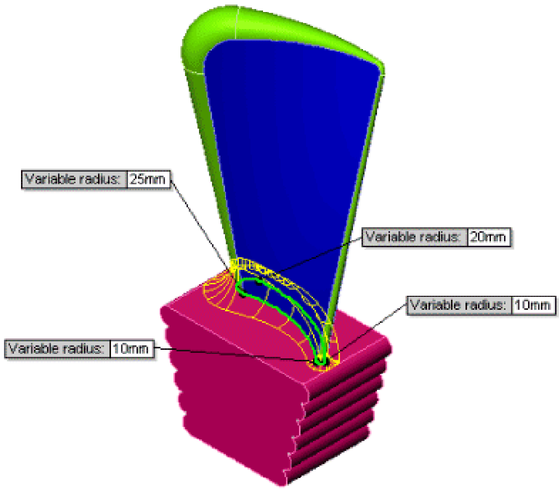
Patterns for Solid and Surface Bodies

Patterns are enhanced to support patterning of solid and surface bodies. In the FeatureManager design tree, the terms **Solid Bodies** and **Surface Bodies** are displayed.

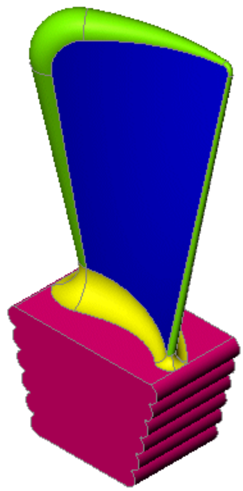
Fillet Previews

Fillets now include a preview in the graphics area. You can display a full or a partial preview. Partial previews are the default. Previews are available with all fillet types except face fillets, and the new full round fillets.

Fillet previews are particularly helpful with complex fillets such as a variable radius fillet as shown below.

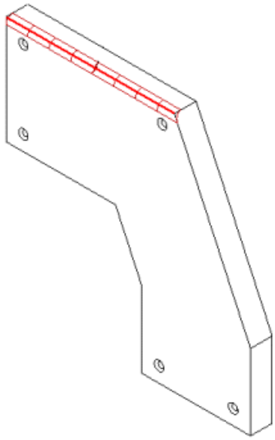


Variable radius full preview

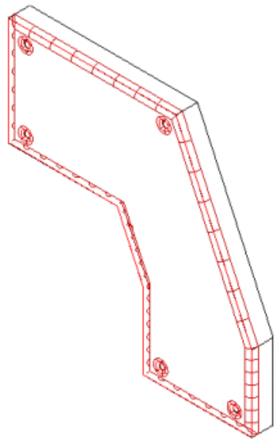


Variable radius applied

Under **Items to Fillet**, select **Partial preview** to display a preview of only the first edge in a series of edges. Select **Full preview** if you choose more than one edge to fillet, and you want *all* selected edges to display in the preview.



Constant radius partial preview
with one edge selected



Constant radius full preview
with face selected

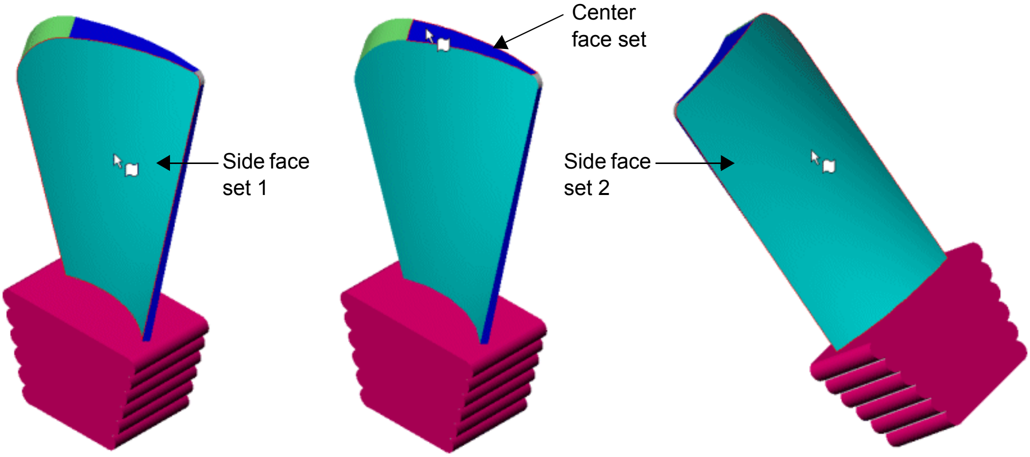
If you choose a face or more than one edge on the model and select **Partial preview**, you can cycle through all the filleted edges. To view each fillet in turn, press **A** (the default toggle key).

Full Round Fillets

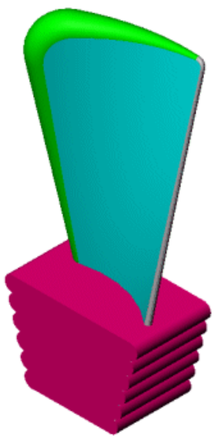
With full round fillets, you can select three adjacent face sets, and apply a fillet that is tangent to the three face sets. A face set includes one or more tangent faces.

To apply a full round fillet:

1. Open **full_round_fillet.sldprt**.
2. Click **Fillet** in the Features toolbar, or click **Insert, Features, Fillet/Round**.
3. Under **Fillet Type**, select **Full round fillet.**
4. Under **Items To Fillet**, do the following:
   - **a)** For **Side Face Set 1**, select the face, as shown.
   - **b)** For **Center Face Set**, select the top face of the model, as shown.
   - **c)** For **Side Face Set 2**, select the face opposite **Side Face Set 1**, as shown.



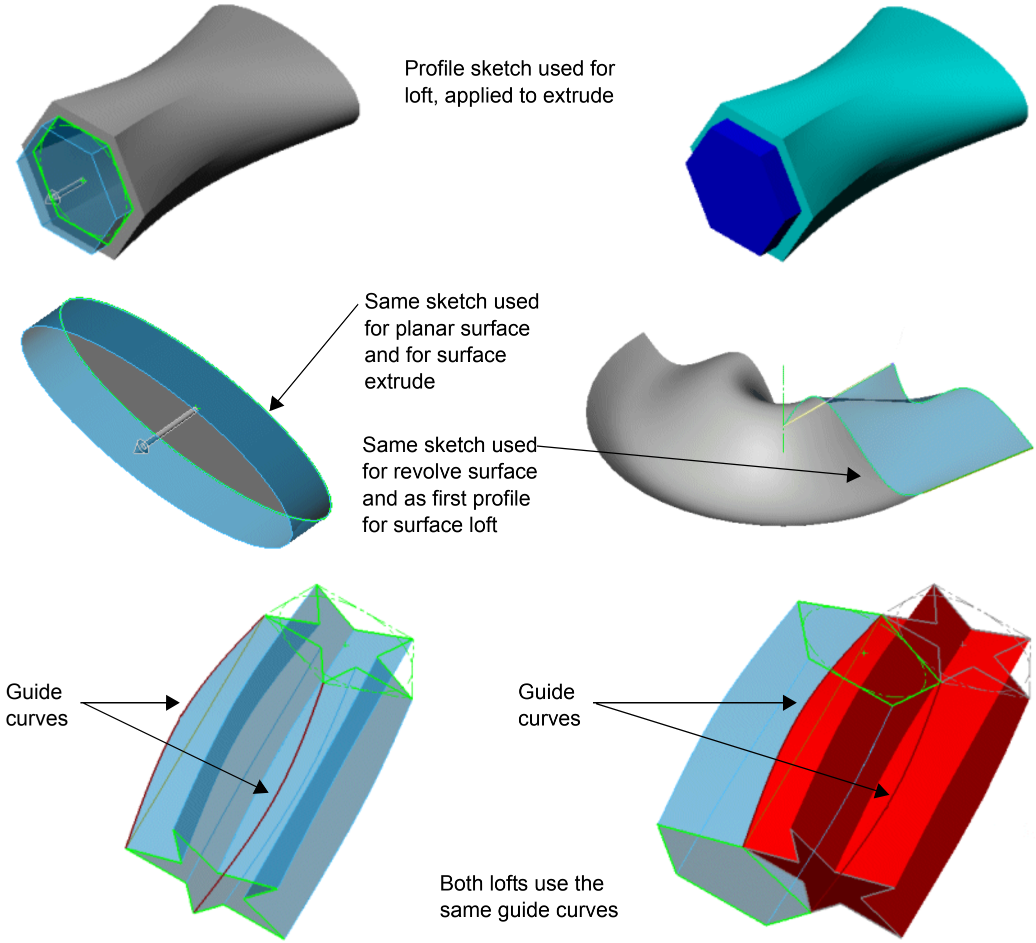
5. Make sure that **Tangent propagation** is selected and click **OK**.



Sketch Reuse in Features

You can use the same sketch entities multiple times within the same model. Shared sketches display the icon in the FeatureManager design tree with the following features:

- Extrudes
- Revolves
- Lofts
- Sweeps

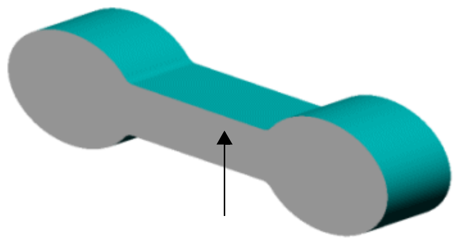


You can apply sketch reuse in conjunction with contour selection. See **Contour Selection** on page 2-2 for information on how to select multiple contours in sketching.

Delete Faces

You can select one or more faces on a solid body and delete the face sets to create one or more surface bodies.

To delete faces from a solid body:



1 Open **delete_faces.sldprt**.

2 Click **Delete Face** on the Surfaces toolbar, or **Insert, Face, Delete**.

The **Delete Face** PropertyManager appears.

3 Under **Faces to delete**, select the front, as shown, and the similar face on the back.

4 Under **Options**, click **Delete**.

5 Click **OK** to delete the front and back faces and create a surface body.

Cut with Plane

You can now use a plane in addition to any surface to cut solid bodies. The user interface is unchanged.

To cut a solid with a plane:

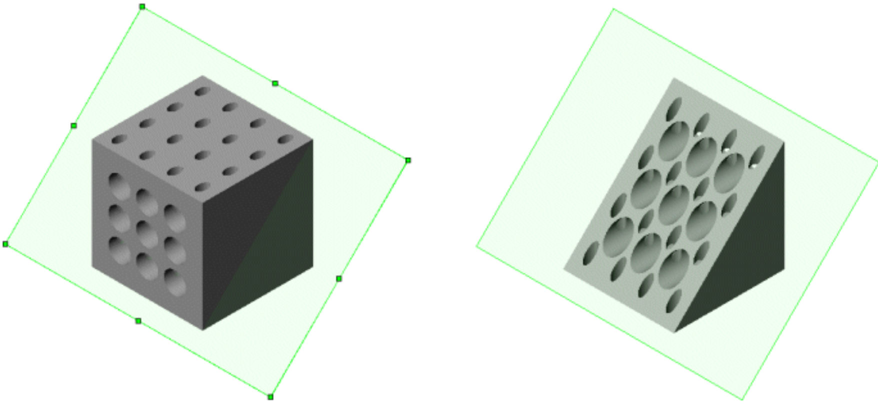
1 Open **cut_with_plane.sldprt**.

2 Click **Insert, Cut, Cut with Surface**.

3 Under **Surface Cut Parameters**, select the plane **1**.

4 If the section you want to keep is not displayed, click **Flip Cut** .

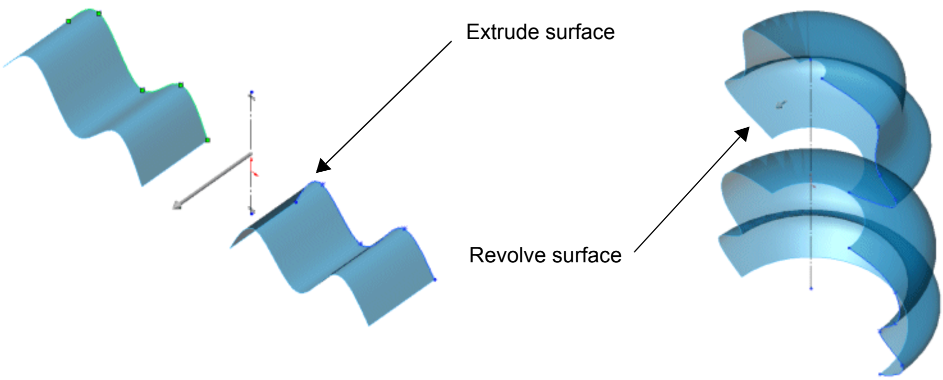
5 Click **OK** .



Multiple Surfaces

You can create multiple surfaces from a single feature in either open or closed contours with the following surface features:

- Planar surface
- Extrude surface
- Revolve surface
- Sweep surface
- Offset surface



Untrim Surface

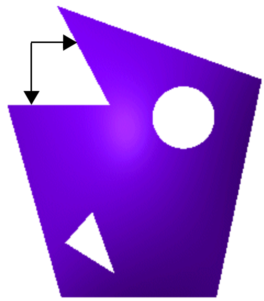
You can apply **Untrim Surface** to any imported surface or to one that you create. With this tool, you can untrim holes and external edges. When you select an edge and use **Untrim Surface**, the surface extends that edge to its natural boundaries.You can also extend the natural boundaries of the surface by a given percentage, or connect endpoints to fill the surface.

To use Untrim Surface:

1 Open **surface_untrim.sldprt**.

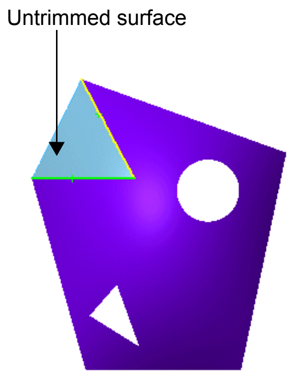
2 Click **Untrim Surface** on the Surfaces toolbar, or **Insert, Surface, Untrim**.

3 The **Untrim Surface** PropertyManager appears.



4 Under **Selections**, choose the two outside edges, as shown. Note the following:

- In the graphic preview, the surface extends, with the natural boundaries constrained by the two edges.
- The **Options** box expands. Under **Edge untrim type**, **Extend edges** is selected by default.
- The distance , shown as a percentage of the total selected surface, is also applicable only when you select two or more external edges.



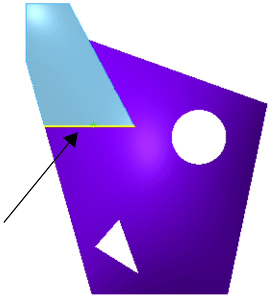
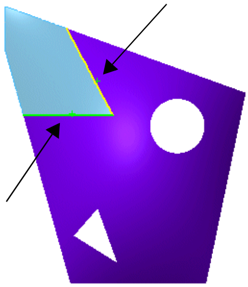
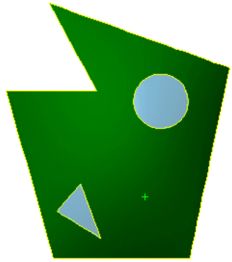
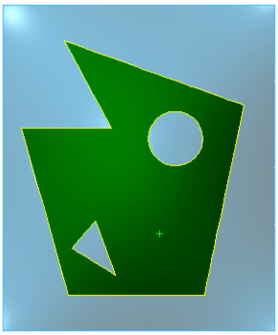
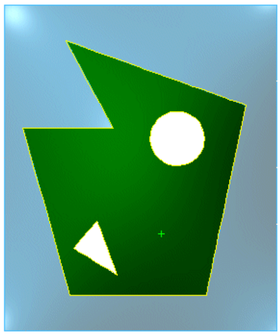
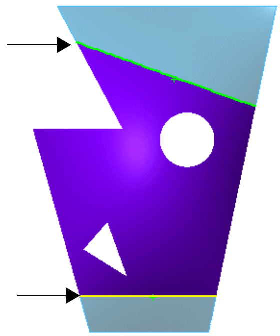
5 Under **Edge untrim type**, select **Connect endpoints**.

The endpoints now define the edge of surface extension.

6 Click **OK** .

The surface is untrimmed.

Additional ways to apply **Untrim Surface** include the following:

		
Select a single external edge, and the surface extends to its natural boundaries.	Select two adjacent external edges, and the surface extends, with the natural boundaries constrained by the two edges.	Select the face, and under **Options**, specify **Internal edges** as the **Face untrim type**. Only the *inside* holes are filled.
		
Select the face, and under **Options**, specify **All edges** as the **Face untrim type**. The inside holes are filled, and the external edges extend to their natural surface boundaries.	Select the face, and under **Options**, specify **External edges** as the **Face untrim type**. Only the *outside* edges are extended to their natural boundaries.	Select the top and bottom external edges, and the surfaces extend to their natural boundaries.

Surface Fill Options

Performance and interface improvements to the **Surface Fill** feature include the following:

- **Optimize surface**. Select the **Optimize surface** check box with a two, three, or four-sided surface, and the system applies a new simplified patch. The simplified patch, based on the Coons patch, is similar to a lofted surface. The advantages of the simplified patch are that the model builds faster, and that a simplified surface fill potentially offers greater stability when used with other features.

If the system cannot apply a simplified patch, the conventional patch is applied. With the conventional patch (**Optimize surface** check box cleared), you can use the **Resolution Control** slider.

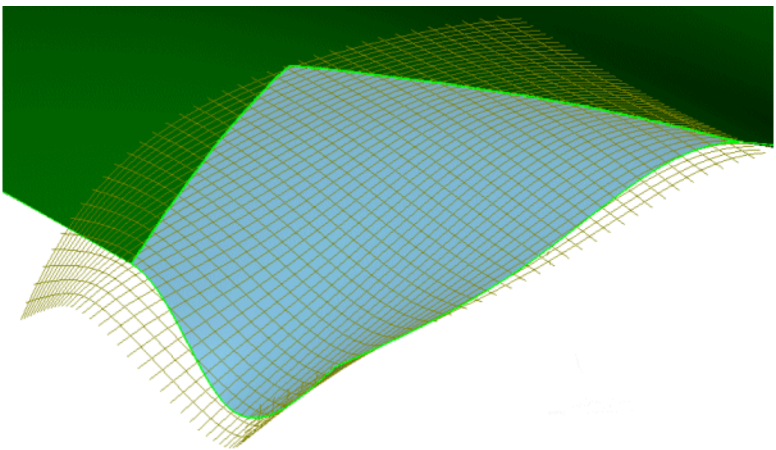
- **Show preview**. Display a shaded preview of the surface fill.
- **Mesh preview**. Display a grid on the patch to help you visualize the curvature.
- **Reverse Surface**. Change the direction of the surface patch.

The **Reverse Surface** button is dynamic, and only displays under specific conditions, such as when all the boundary curves are coplanar, there are no constraint points, and so on.

To change the surface fill patch type:

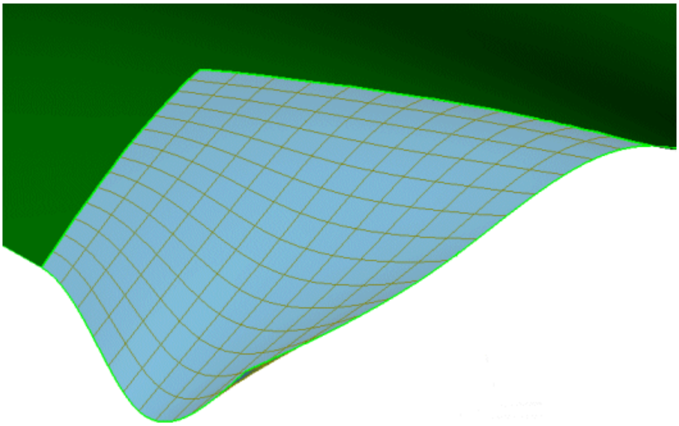
1. Open **fill_surface_change_patch.sldprt**.
2. In the FeatureManager design tree, right-click **Surface-Fill1** and select **Edit Definition**.

   The **Surface-Fill1** PropertyManager appears.
3. Under **Patch Boundary**, if necessary, select the **Show preview** and **Preview mesh** check boxes.



Mesh preview using conventional surface fill patch

4 Under **Patch Boundary**, select the **Optimize surface** check box to apply the simplified patch.

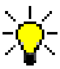


Mesh preview using the **Optimize surface** option

Note how the grid pattern is more uniform, and does not spread beyond the surface fill patch. Depending on your model, the visible differences are not always apparent. However, the gains in performance and behavior warrant using the **Optimize surface** option with a two, three and four-sided surface.

5 Click **OK** .

The surface fill patch is changed.



With **Surface Fill** (and loft, add loft section, and sweep features), you can view **Zebra Stripes**. With surface fill, place the pointer on the surface fill, and use the shortcut menu.

Deviation Analysis

The deviation analysis tool calculates the angle between faces adjacent to a selected edge. You can select the edges between faces on a surface, or any edges on a solid. After you select the edges, you base the analysis on the number of sample points along the edges.

When you apply deviation analysis between two adjacent faces resulting from a surface fill, analysis results are influenced by the following factors: the **Curvature Control** settings (**Contact** or **Tangent**), and the type of patch you applied. See **Surface Fill Options** on page 3-14.

In the first procedure, "To use deviation analysis," you apply the **Deviation Analysis** tool with a model that uses **Contact** as the **Curvature Control** setting. In the second procedure, "To view different deviation analysis results" (page 3-17), you edit the **Surface Fill** definition, and change the **Curvature Control** setting to **Tangent**. Then you apply the **Deviation Analysis** tool again.

To use deviation analysis:

1 Open **deviation_analysis.sldprt**.

2 Select the edge, as shown at right.

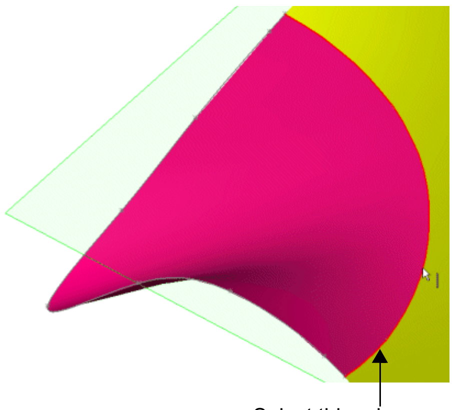
3 Click **Tools, Deviation Analysis**.

The **Deviation Analysis** PropertyManager appears.

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

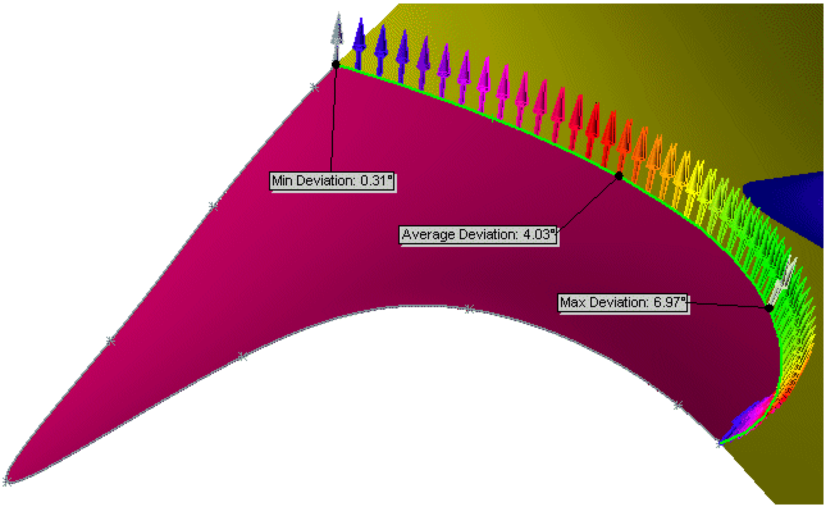
a) Use the slider to set the **Number of Sample Points** .

b) Click **Calculate**.



The SolidWorks application determines the number of sample points based on the position of the slider, the number of edges selected, and the size of the window.

Note the results in the graphics area for **Min Deviation** (minimum), **Max Deviation** (maximum), and **Avg Deviation** (average).



5 Click **OK** ✔.

If you move your pointer along the selected edges, the system displays the deviation value at the current position.

To view different deviation analysis results:

1 In the FeatureManager design tree of **deviation_analysis.sldprt**, select **Surface-Fill4** and click **Edit Definition**.

The **Surface-Fill4** PropertyManager appears.

2 Under **Patch Boundary**, do the following:

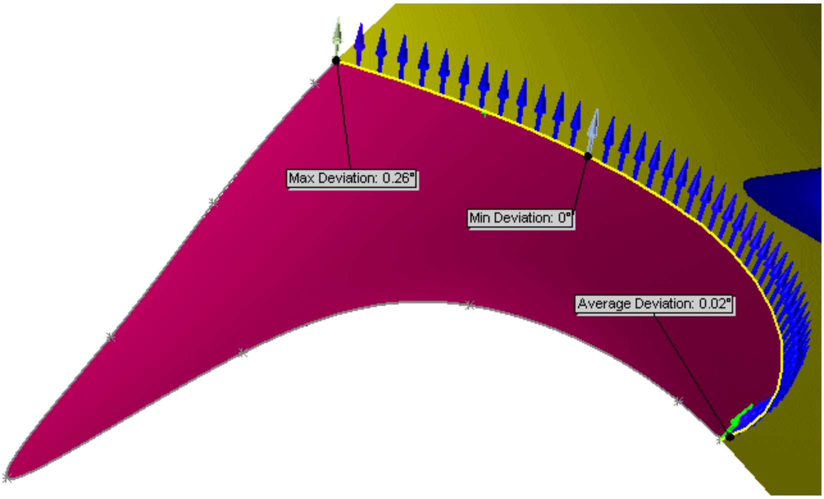
a) Select **Edge <1>-Contact** as the patch boundary.

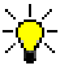
b) Change the **Curvature Control** to **Tangent**.

3 Click **OK** ✔.

4 Repeat steps 2, 3 and 4 from the proceeding procedure (see page 3-16) to reapply the deviation analysis.

Note the results in the graphics area for **Min Deviation** (minimum), **Max Deviation** (maximum), and **Avg Deviation** (average).





You can change the display colors for **Maximum Deviation** and for **Minimum Deviation**. To change the colors, click **Edit Color** in the **Deviation Analysis** PropertyManager to display the **Color** palettes, select a color for each deviation type, and click **OK** or click **Calculate** to apply the new colors.

4

Parts

This chapter describes enhancements to parts in the following areas:

- ❑ Derived parts
- ❑ Insert parts
- ❑ Interrupt regeneration
- ❑ Custom Properties
- ❑ Feature Statistics
- ❑ Measuring Tools
- ❑ Design Analysis Tool

Derived Parts

You can now transfer planes, axes, and cosmetic threads from the base part when you mirror and derive parts. Derived and mirrored parts optionally stay synchronized with multiple reference levels. To access this option, click **Tools**, **Options**. On the **System Options** tab, click **External References**. Under **Load Reference Documents**, select one of the following:

- **Prompt**. Prompts you to open referenced documents.
- **All**. Opens all referenced documents.
- **None**. Does not open any referenced documents.
- **Changed Only**. Opens only those referenced documents that have changed since the last time the current document was opened.

To open a derived part:.

1 Create a new part document from the Tutorial tab.

2 Click **Insert**, **Part**.

3 Browse to **\faucet\round_handle.sldprt**, and click **Open**.

The **Insert Part** PropertyManager appears.

4 Under **Transfer**, select the **Axis**, **Plane**, and **Cosmetic Thread** check boxes.

This transfers axes, planes, and cosmetic threads from the base part into the new part.

5 Click **OK** .

6 Expand **round_handle** in the FeatureManager design tree.

Note the planes. Each plane has been transferred and the name of the part added to the plane name.

Insert Parts

You can use **Insert Part** to insert one or more base parts multiple times into a part document. This creates a multibody part. For more information on multibody parts, see Chapter 5, "Multibody Parts."

When you insert more than one part, the **Locate Part** PropertyManager appears automatically so that the part is not placed on the origin over another part.

To insert multiple parts:

1 Create a new part document from the Tutorial tab.

2 Click **Insert**, **Part**.

3 Browse to **\faucet\round_handle.sldprt**, and click **Open**.

The **Insert Part** PropertyManager appears.

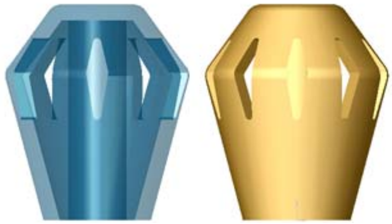
4 Under **Transfer**, select the **Axis**, **Plane**, and **Cosmetic Thread** check boxes.

This transfers axes, planes, and cosmetic threads from the base part into the new part.

5 Under **Locate Part**, select the **Launch Move Dialog** check box.

6 Click **OK**.

The **Locate Part** PropertyManager appears.

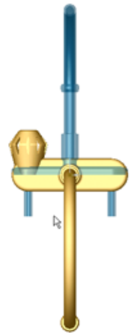


7 Under **Translate**, type **-50** for the **Delta X** value.

A dynamic preview of the part appears.

8 Click **OK**.

9 Repeat steps 2 and 3, opening **\faucet\parts_faucet.sldprt.**



Notice that the **Launch Move Dialog** check box is already selected.

10 Click **OK**.

The **Locate Part** PropertyManager appears.

11 Under **Rotate**, set **X Rotation Angle** to **270**.

12 Click **OK**.

Interrupt Regeneration

You can now press **Esc** to interrupt the regeneration of parts.

For example, suppose you just added a feature to a complex part, and you realize that you made an error. Instead of waiting for the part to regenerate with the incorrect feature, you can press **Esc** to interrupt the regeneration of the part. This also works with opening parts, rollbacks, and so on. The status of the rebuild is displayed in the status bar.

When you interrupt the regeneration of a part, the system completes regeneration of the current feature and then places the rollback bar after the feature.

Custom Properties

You can edit the list of custom properties found in **File**, **Properties**, **Custom** using the new **Edit List** button. You can add and delete custom properties, as well as change the order of the list with **Move Up** and **Move Down**.

This list is now stored in a text file. Since it is a text file, you can edit this file using any text editor. The file location is in **Tools**, **Options**, **System Options**, **File Locations**.

Feature Statistics

There is a new **Feature Statistics** tool on the **Tools** menu. **Feature Statistics** displays the feature name, percent time of rebuild, and the time for each feature to rebuild. Using this tool, you can optimize speed by suppressing features that take more time to rebuild. You can suppress, hide bodies, rollback, and so on using the shortcut menu.

To use feature statistics:

1. Open **two_bolt_flange.sldprt**.
2. Click **Feature Statistics** or click **Tools**, **Feature Statistics**.

   The **Feature Statistics** dialog box appears with the list of all features and their rebuild times in descending order.
3. Click **Feature Name**.

   This sorts the features to match the order of the FeatureManager design tree.
4. Right-click **Boss-Extrude1** and select **Suppress**.
5. Click **Refresh**.

   Note that **Boss-Extrude1** is now suppressed and its rebuild time is 0.00 sec. Note also that the **Features** are in descending rebuild time again.
6. Click **Close**.
7. Close the part without saving it.

Measuring Tools

You no longer need to select the entities that you want to calculate before you select the following tools: **Mass Properties** and **Measure** on the Tools toolbar, and **Section Properties**.

Mass Properties and **Section Properties** display a tri-colored, 3D triad, and a red 3D triad at the centroid of the calculated entities. A new check box, **Show output coordinate system in corner of window** displays the tri-colored 3D triad in the corner of the window. When the check box is not selected, the triad displays at the origin of the part.

Mass Properties

You can select a new check box, **Include Hidden Bodies/Components**, in the **Mass Properties** dialog box to indicate if you want to include hidden bodies and components in the calculation. For more information on **Mass Properties of Multibodies**, see **Mass Properties** on page 5-12.

You can select a higher accuracy level for calculations. On the **Mass Properties** dialog box, click **Options**. The **Measurement Options** dialog box appears. Under **Accuracy Level**, you can select one of the following:

- **Default mass/section property precision**. This is the calculation used in earlier versions of the SolidWorks software.
- **Maximum property precision (Slower)**. This provides greater accuracy of the calculation, but the computation is slower.

Measure

You can now use the **Measure** tool on the Tools toolbar, or **Tools**, **Measure** tool to measure the total surface area of multiple faces. If you select two or more faces, **Measure** displays the relationship between the faces.

Section Properties

You can now calculate section properties for multiple faces that lie in parallel planes. The faces must be planar and parallel in order for the **Section Properties** tool to perform the calculation.

Design Analysis Tool

You can now use **COSMOSXpress** to perform stress analyses on part documents. Click **Tools**, **COSMOSXpress** to start the **COSMOSXpress** wizard that guides you through a five step process. This wizard lets you specify units, materials, restraints, and loads.

COSMOSXpress uses the same design analysis technology that COSMOS/Works uses to perform stress analysis. More advanced analysis capabilities are available within the COSMOS/Works line of products.

Based on the analysis results, you can modify designs to strengthen unsafe or weak regions and remove material from overdesigned regions.

For more information and step-by-step examples, see the *Online Tutorial*. Click **Help**, **Online Tutorial**.

5

Multibody Parts

This chapter describes the new multibody features and functions in the following areas:

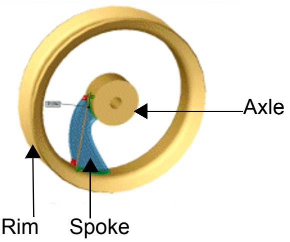
- Multibody parts
- Modeling techniques
- Multibody features

Multibody Parts

Multibody Overview

Part documents can now contain multiple solid bodies. A folder named **Solid Bodies** appears in the FeatureManager design tree when there are solid bodies in a single part document. The number of solid bodies in the part document displays in parentheses next to the **Solid Bodies** folder.

For example, when you design a spoked wheel, you know the requirements of the rim and the axle. However, you do not know how to design the spoke. With multibody parts, you can create the rim and axle, then create the spoke to connect the bodies.



You can manipulate multibody solids the same ways you manipulate single solid bodies. For example, you can add and modify features, and change the names and colors of each solid body.

You can hide and show solid bodies in the FeatureManager design tree. You can create multiple solid bodies from a single feature with the following commands:

- Extrude boss and cut (including thin features)
- Revolve boss and cut (including thin features)
- Sweep boss and cut (including thin features)
- Surface cut
- Boss and cut thicken
- Cavity

Multibody Parts Versus Assemblies

Multibody parts should not replace the use of assemblies. A general rule to follow is that one part (multibody or not) should represent one part number in a Bill of Materials. A multibody part consists of multiple solid bodies which are *not* dynamic. If you need to represent dynamic motion among bodies, use an assembly. Tools such as **Move Component**, **Dynamic Clearance**, **Mates**, and **Collision Detection** are available only with assembly documents.

You can save an assembly as a multibody part document. This enables you to save complex assemblies as smaller part documents to facilitate file sharing. For more information about saving assemblies as multibody part documents, see Save on page 5-16.

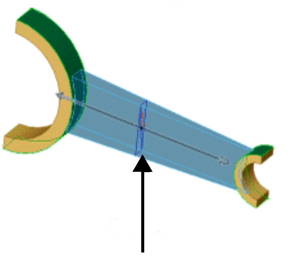
Modeling Techniques

There are many modeling techniques that you can use in a multibody environment. This section reviews the following techniques:

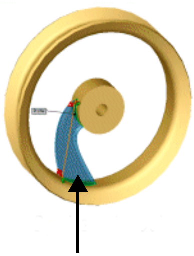
- Bridging
- Local Operations
- Symmetry Modeling
- Body Intersection
- Tool Modeling

Bridging

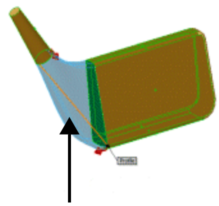
Bridging is a commonly used technique in a multbody environment. Bridging creates a solid that connects multiple solid bodies. This technique is useful when you create portions of the model first and create the connecting geometry afterwards.



Bridge



Bridge



Bridge

For example, you need to design a golf club. You know the specifics of the head and shaft design, but not necessarily how they are connected. You can design the head and the shaft first, then bridge the two bodies.

To design a part with the bridging technique:

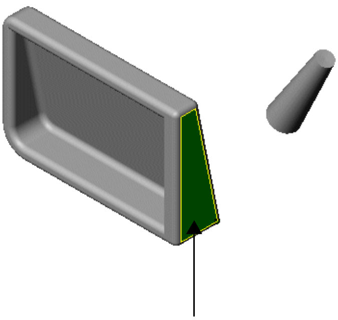
1. Open **multi_bridge.sldprt**.

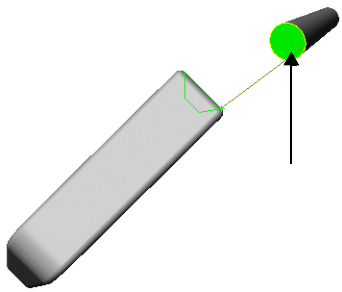
   Notice in the FeatureManager design tree that you can expand the **Solid Bodies** folder to see **Shaft** and **Club Head**.

2. Click **Loft** on the Features toolbar, or click **Insert**, **Boss**, **Loft**.

   The **Loft** PropertyManager appears.

3. Under **Profiles**, select the vertical face of the club head and the bottom face of the shaft, as shown.



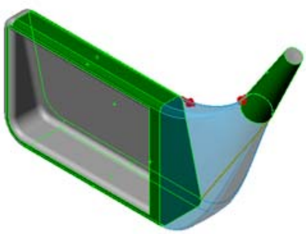


Use the arrow keys to rotate the model so that you can select the shaft face more easily.

4. Under **Start/End Tangency** do the following:
   - Set **Start tangency type** to **All Faces.**
   - Set **Start Tangent Length** to **1.00**.
   - Set **End tangency type** to **All Faces.**
   - Set **End Tangent Length** to **1.00**.

5. Under **Options**, clear the **Merge result** check box.

   This keeps all bodies separate. Otherwise, this check box merges the connecting bodies into a single body.

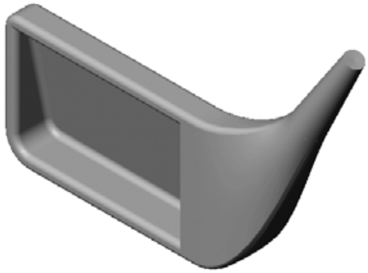


6 Click **OK** .

Loft1 appears in the FeatureManager design tree as a new feature and in the **Solid Bodies** folder as a new solid.

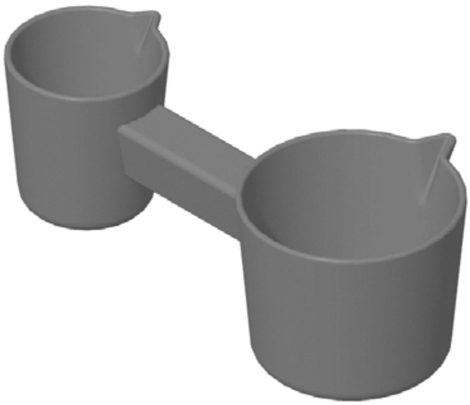
7 Select each solid body in the **Solid Bodies** folder.

Notice how each body highlights in the graphics area.



Local Operations

You use local operations when you want to perform operations on certain portions of the model, but not on others. For example, you design a double-ended measuring cup. You need to shell the two cups and fillet them. However, you do not want to shell the piece that connects the two cups. You can create the part and perform the feature operations on the separate bodies.



To design a part with the local operations technique:

1 Open **multi_local.sldprt**.

2 Click **Shell** on the Features toolbar, or click **Insert**, **Features**, **Shell**.

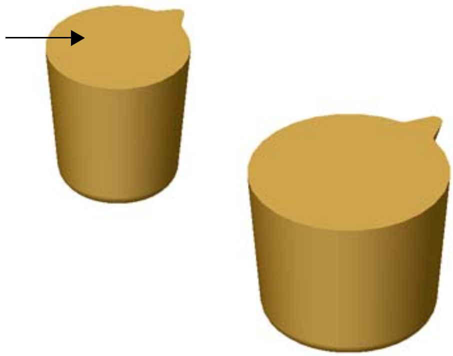
The **Shell1** PropertyManager appears.

The **Shell** feature is used separately on solid bodies; one shell feature applies to one body.

3 Select the top face of the smaller cup.

4 Under **Parameters**, set the **Thickness** to **2.00mm** and click **OK**.

5 Repeat steps 2 through 4, using the top face of the larger cup for **Shell2**.



To create the connecting piece:

1 Click **Sketch4** in the FeatureManager design tree.

2 Click **Extruded Boss/Base** on the Features toolbar, or click **Insert**, **Boss**, **Extrude**.

The **Extrude** PropertyManager appears.

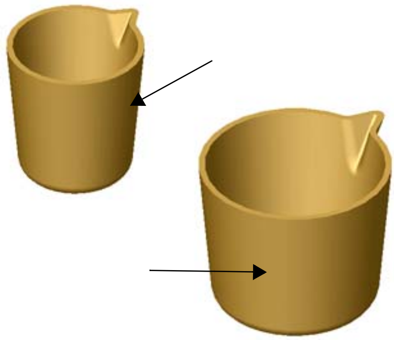
3 Under **Direction1**, do the following:

- Set **End Condition** to **Up To Body**.
- Select the smaller cup for **Solid/Surface Body**.
- Select the **Merge result** check box.

4 Under **Direction2**, do the following:

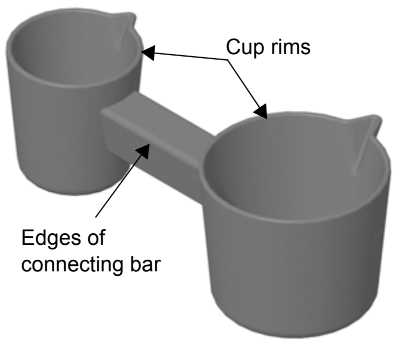
- Set **End Condition** to **Up To Body**.
- Select the larger cup for **Solid/Surface Body**.

5 Click **OK**.



6 To complete the part, apply the following fillets:

- 3mm constant radius fillet to the four edges of the connecting piece.
- 1mm constant radius face fillet to the cup rims.



Symmetry Modeling

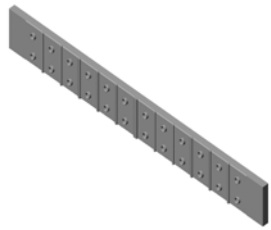
Symmetry modeling simplifies the creation of axis symmetric parts and also speeds performance for these types of parts. In this approach you make one symmetric body, pattern the bodies to obtain the remaining geometry, then use the **Combine** feature to “glue” all of the bodies together. You can use multiple pattern and combine features to create an entire model.

For instance, the example below shows the design progression of a symmetrical part. You begin by building the basic piece that you pattern later. Next, you add the end piece, keeping this body separate, but adjacent. Then you pattern the basic piece. Lastly, you mirror the entire part, including the end piece.

Basic piece

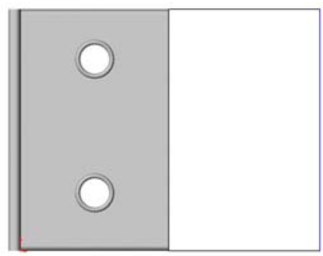
Add end piece

Pattern the basic piece and combine all bodies to make a single body



Mirror the body to create the complete part

To design a part with the symmetry modeling technique:



1. Open **multi_symm.sldprt**.
2. Edit **Sketch7**.
3. Click **Extruded Boss/Base** .

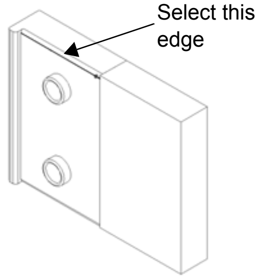
   The **Extrude** PropertyManager appears.
4. Clear the **Merge result** check box.

   This maintains the new extrusion as a separate body.
5. Under **Direction1** do the following:
   - Set **Depth** to **10mm**.
   - Set **End condition** to **Blind.**
6. Click **OK** .

To pattern the solid body:

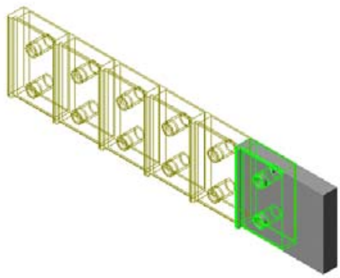
1 Click **Linear Pattern** on the Features toolbar, or click **Insert**, **Pattern/Mirror**, **Linear Pattern**.

The **Linear Pattern** PropertyManager appears.



2 Under **Direction 1** do the following:

- Select the top edge for **Pattern Direction** as shown.
- Click **Reverse Direction** if necessary.
- Set **Spacing** to **33mm**.
- Set **Number of Instances** to **6**.

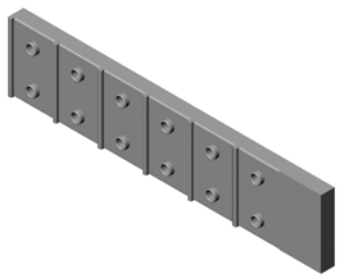


3 Under **Features to Pattern**, right-click and select **Clear Selections**.

4 Under **Bodies to Pattern**, **Solid/Surface Bodies to Pattern**, select **Fillet2**.

Use the flyout FeatureManager design tree to select **Fillet2** from the **Solid Bodies** folder.

Solid Body<1> appears in the **Solid/Surface Bodies to Pattern**.



5 Click **OK**.

In the **Solid Bodies** folder, the patterned bodies appear as separate solid bodies.

To use the Combine feature to make the multibody part a single body:

1. Click **Combine** on the Features toolbar, or click **Insert**, **Features**, **Combine**.

   The **Combine1** PropertyManager appears.

2. Under **Operation Type**, click **Add**.

   > You can use the **Add** option of the **Combine** feature only when the bodies are abutting.

3. Under **Bodies to combine**, select **Extrude4** and all **LPattern** bodies in the **Solid Bodies** folder in the FeaureManager design tree.

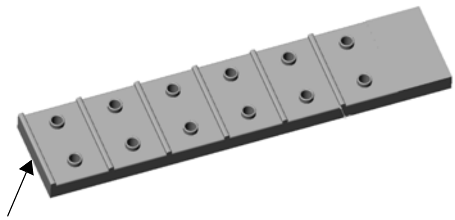
4. Click **OK**.

   The bodies are combined to form a single solid body in the **Solid Bodies** folder in the FeaureManager design tree.

To mirror the bodies:

1. Click **Mirror Feature/Face/Surface** on the Features toolbar, or click **Insert**, **Pattern/Mirror**, **Mirror**.

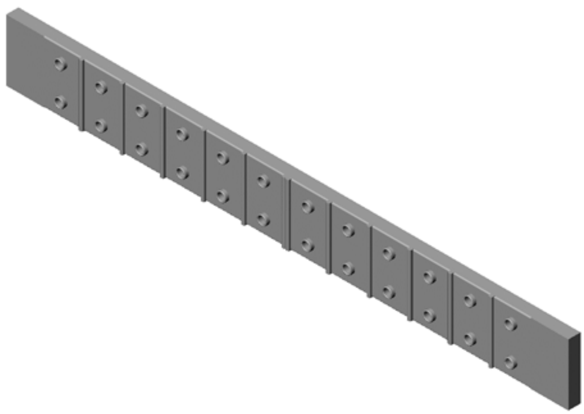
   The **Mirror** PropertyManager appears.



2. In **Mirror Face/Plane**, select the face shown.

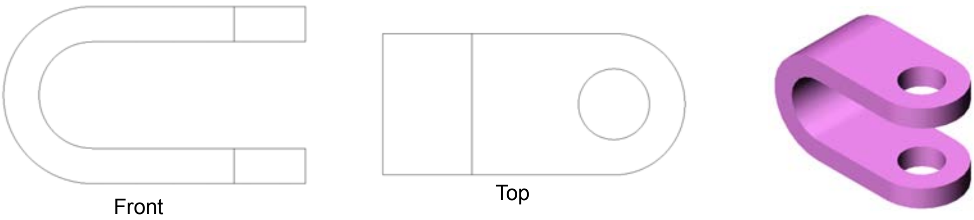
3. Under **Bodies to Mirror**, select the solid body.

4. Click **OK**.

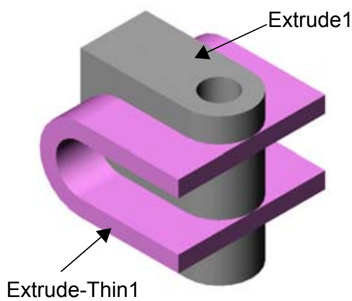


Body Intersection

For the body intersection technique, you use the **Combine** feature and its **Common** option. Body intersection is a quick way to create complex parts with very few operations, which can result in faster performance. The operation takes multiple solid bodies that overlap one another and results in only the intersecting volumes of the bodies. For most models that can be represented fully by two or three drawing views, this technique can be used by intersecting either two or three extruded solids. The extrusion sketches are the solid lines represented in the two or three views. The following example shows this technique with the intersection of two extrusions.



To design a part with the body intersection technique:



1 Open **multi_inter.sldprt**.
2 Click **Combine** on the Features toolbar, or click **Insert**, **Features**, **Combine**.

   The **Combine1** PropertyManager appears.
3 Under **Operation Type**, click **Common**.
4 Under **Bodies to combine**, select **Extrude-Thin1** and **Extrude1** from the FeatureManager design tree, or select them from the graphics area.

   Solid Body<1> and **Solid Body<2>** appear in **Solid Bodies**.
5 Click **OK**.

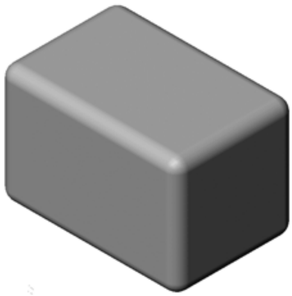
The solids that overlap combine and the excess is shed to reveal a single body.

In previous versions of SolidWorks, you use the following features to create this part: base extrude, cut extrude, shell, second cut extrude, and fillet. Multibody parts allow you to create the same part with three features.

Tool Body Modeling

Use tool body modeling to create complex tools to remove material from a solid body, or add complex shapes to geometry. You create common geometrically shaped bodies in separate part documents, then use the **Insert**, **Part** tool to create the multibody part document.

To design a part with the tool body modeling technique:



1 Open **multi_stamp_block.sldprt**.

2 Click **Insert**, **Part**.

3 Browse to **multi_stamp.sldprt** and click **Open**.

The **Insert Part** PropertyManager appears.

4 Make sure the **Launch Move Dialog** check box is selected, then click **OK**.

The **Locate Part** PropertyManager appears.

5 Under **Translate**, set the **Delta X** to **15** and **Delta Z** to **90**.

A preview of the inserted part appears in its new location.

6 Click **OK**.

Combine the two bodies:

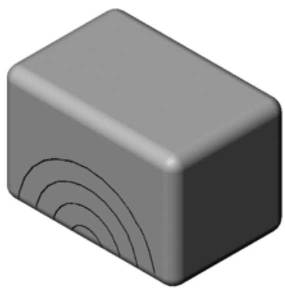
1 Click **Combine** on the Features toolbar, or click **Insert**, **Features**, **Combine**.

The **Combine1** PropertyManager appears.

2 Under **Operation Type**, select **Subtract**.

3 Under **Main Body**, select **Fillet1** from the **Solid Bodies** folder in the FeatureManager design tree.

Solid Body <1> appears under **Main Body**.



4 Under **Bodies to subtract**, select **multi_stamp** from the **Solid Bodies** folder in the FeatureManager design tree.

Solid Body <2> appears under **Bodies to subtract**.

5 Click **OK**.

The material from one solid body is taken away from the other.

Multibody Features

Features have been enhanced to include capabilities for multibody parts. A feature scope has been added so that you can select which bodies are affected when you apply features. The following tools have been enhanced to include multibodies:

- ❑ Mass Properties
- ❑ Selection Filter
- ❑ Surface Move/Copy
- ❑ Cut and Cavity
- ❑ Split Body
- ❑ Delete Body
- ❑ Save
- ❑ Translators

Feature Scope

You can set the scope of feature inclusion for the following features: boss and cut extrude, revolve, cavity, sweep, cut with surface, and thicken. In each feature's PropertyManager, there is a new option called **Feature Scope**. You can select **All bodies**, **Selected bodies**, or **Auto-select**.

- **All bodies** merges the feature with all possible solid bodies in the part document.
- **Selected bodies** uses only the bodies that you select.
- **Auto-select** applies the feature to any body that it can. **Auto-select** remembers the automatically selected bodies and regenerates the feature using only those bodies. This option offers improved performance compared to the **All bodies** because with the **All bodies** selection, every time the feature is regenerated, SolidWorks attempts to merge the feature with every body.

Mass Properties

The Mass properties tool allows selection of individual bodies of a multibody part for mass property calculations. There is a new check box to indicate whether or not to include hidden bodies and components. You do not need to pre-select features for calculation.

Selection Filter

The Selection Filter toolbar contains a filter for solid bodies. Click **Filter Solid Bodies**  to select solid bodies in multibody documents.

To calculate mass properties:

1 Open **\faucet\multibody_hidden.sldprt**.

> Depending on the **External References** option selected on the **System Options** tab, a message appears to ask if you want to open referenced documents. If a message appears, click **Don't open any referenced documents** and click **OK**.
>
> For more information and to learn how to change this option, see **Derived Parts** on page 4-2.

Note the faucet handles are hidden.

2 Click **Tools**, **Mass Properties**.

The **Mass Properties** dialog box appears with a default mass property calculation already performed for all unhidden bodies of the document.

3 Select the **Include Hidden Bodies/Components** check box and click **Recalculate**.

The mass properties are recalculated, with hidden bodies included.

4 Click **Close**.

Surface Move/Copy

The **Surface Move/Copy** feature has been enhanced to support moving and copying of solid bodies.

To move and copy solid bodies:

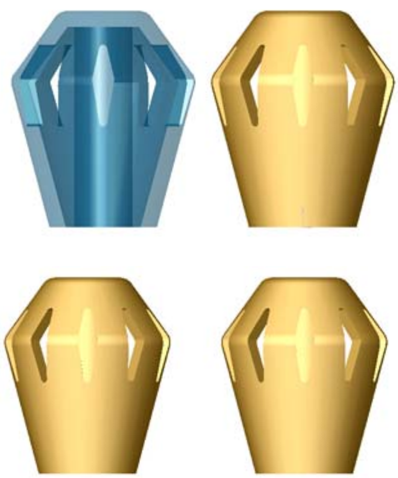
1 Open **\faucet\round_handle.sldprt**.

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

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

3 Under **Bodies to Move/Copy** , do the following:

- Select the faucet handle.
- Select the **Copy** check box.



4 Under **Translate**, set **Delta X** , to **60mm**.

A preview of the copy appears.

5 Click **OK** .

The faucet handle is copied.

6 Close the part without saving it.

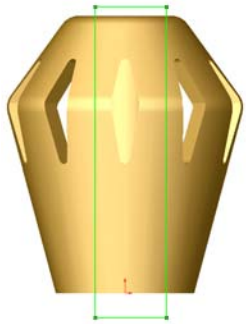
Cut and Cavity

The **Extrude Cut** on the Features toolbar and the **Cavity** tool on the Mold Tools toolbar now use a dialog box which allows you to select which bodies to keep.

Cut

When you make a cut in which you split a part into multiple solids, a dialog box appears that allows you to select which bodies to keep.

To cut and keep multibody solids:



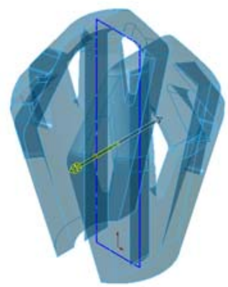
1 Open **\faucet\round_handle.sldprt**.

2 Open a new sketch on the **Front** plane.

3 Sketch a rectangle approximately as shown.

4 Click **Extruded Cut** on the Features toolbar, or click **Insert**, **Cut**, **Extrude**.

The **Cut-Extrude** PropertyManager appears.



5 Set the following parameters:

- Under **Direction 1**, select **Through All** from the **End Condition** list.
- Under **Direction 2**, select **Through All** from the **End Condition** list.

6 Click **OK**.

A preview of the cut and the **Bodies to Keep** dialog box appear.

7 Click **Selected bodies** and select **Body 1** and **Body 2**.

8 Click **OK**.

The handle is cut.

9 Close this part without saving it.

Cavity

In the context of an assembly, you can use the **Cavity** tool on the Mold Tools toolbar to make cuts. In the **Bodies to Keep** dialog box, you can select which bodies to keep in the assembly document.

Split Body

The **Split** feature now supports the multibody environment, which allows you to split bodies without exporting them to separate part files. You can select to do the following with the resultant bodies:

- **Show bodies**. Show all bodies.
- **Hide bodies**. Hide bodies selected in the **Resulting Bodies** box. These bodies remain a part of the part document.
- **Consume bodies**. Removes the split bodies from the current part document.

To split a solid into multiple bodies:

1 Open **\faucet\round_handle.sldprt**.

2 Click **Split** on the Features toolbar, or click **Insert**, **Features**, **Split**.

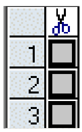
The **Split** PropertyManager appears.

3 Under **Trim tools**, select **mid-plane** from the FeatureManager design tree.

4 Click **Cut Part**.

The **Resulting Bodies** box updates to show that there are three solid bodies as a result of the cut.

> Under **Resulting Bodies**, you can select a file to save the cut part. Double-click **File**. The **Save As** dialog box appears.

5 Click the **1**, **2**, and **3** check boxes. 

When you select these check boxes, you keep the bodies in the same part document.

6 Select **Show bodies**, then click **OK**.

7 Click to expand the **Solid Bodies** folder in the FeatureManager design tree.

Note that there are now three solid bodies: **Split1[1]**, **Split1[2]**, and **Split1[3]**.

Delete Body

You can delete a body or bodies from a multibody part document.

To delete a body:

1. Open **\faucet\multibody_faucet.sldprt**.

   A message appears to ask if you want to open referenced documents.

2. Click **Don’t open any referenced documents** and click **OK**.

3. Click **Delete Solid/Surface** on the Features toolbar, or click **Insert**, **Features**, **Delete Body**.

   The **Delete Body** PropertyManager appears.

4. In **Bodies to Delete**, select one of the handles from either the FeatureManager design tree or the graphics area.

5. Click **OK**.

Save

You can now save an assembly as a multibody part document. This enables you to save complex assemblies as smaller part documents to facilitate sharing files. For example, you have a design of an intricate motor assembly and a potential customer wants to know if it fits in their frame. You can save the motor assembly as a part document and send the part file to potential customers without risking design integrity or transmitting a large document file.

To save an assembly as a multibody part document:

1. Open **\faucet\faucet_assembly.sldasm**.

2. Click **File**, **Save As**.

   The **Save As** dialog box appears.

3 Set the **Save as type** to **Part (*.prt, *.sldprt)**.

A set of options appears at the bottom of the dialog box.

Assembly geometry to save in part file:

- **Exterior Faces**. Save the exterior faces.
- **Exterior Components**. Save the exterior components.
- **All Components**. Save all components.

4 Select **All Components**, then click **Save**.

This saves all components of the assembly as solid bodies in a multibody part document.

5 Open the part file. Note that in the **Solid Bodies** folder, you have two solid bodies. Also note that there are no mates.

Translators

All SolidWorks translators now support multibody parts. For more information on importing and exporting files, see Chapter 9, “Import and Export.”

6

Assemblies

This chapter describes enhancements to assemblies in the following areas:

- General enhancements
- Mate references
- Mirror components
- Replacement components
- Replacement mate entities
- Physical Simulation

General Enhancements

Assembly Patterns

You can now pattern an assembly feature pattern. For example, you can create a circular pattern of a linear pattern of bolt holes. Also, you can now pattern a component pattern.

Feature Palette Window

You can add assemblies to the Feature Palette window and drag them into SolidWorks documents. In earlier versions, the Feature Palette window contained only parts and features. All of the previous Feature Palette window functions for features and parts apply to assemblies.

Lightweight Parts

You do not have to resolve parts that are affected by assembly features when opening an assembly. You can keep them as lightweight parts until you choose to resolve them or until another assembly operation, such as editing the part, resolves them automatically.

Save an Assembly as a Part

With the addition of multibody parts, you can save an assembly as a part. For more information, see **Save** on page 5-16.

Mate References

There are a number of enhancements to mate references, including:

- **Assembly mate references**. Assembly documents can contain mate references. You can select assembly geometry (such as a plane in the assembly) or component geometry (such as the face of a component.) In earlier releases, you could only add mate references in part documents.
- **Multiple mate references**. A part can contain more than one mate reference, each with its own name. The **MateReferences** folder in the FeatureManager design tree holds all references.
- **Multiple mated entities**. Each mate reference can contain up to three mated entities. Each of these entities can have an assigned mate type and alignment.

  For example, a shaft can have its cylindrical face assigned to a concentric mate and its planar end face assigned to a coincident mate. When you drag that component into an appropriate location in an assembly, the SolidWorks software adds both mates.

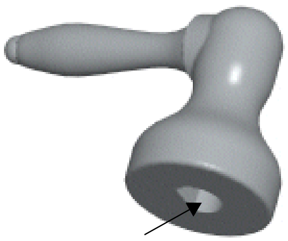
- **Menu item**. The **Mate Reference** menu item is now on the **Insert** menu; it was on the **Tools** menu in earlier versions.
- **Mates in an assembly**. When you drag a component with one or more mate references into an assembly, the SolidWorks software tries to find other combinations of the same mate reference name and mate type.

If you drag a component into an assembly with the **SmartMates** tool active, mate references are ignored when the application considers possible mate combinations.

To add a mate reference to a part:

1. Open **\faucet\faucet_handle.sldprt**.
2. Click **Insert**, **Mate Reference**.

   The **Mate Reference** PropertyManager appears.
3. Type **handle_alignment** in the **Reference name** box.
4. Select the circular face of the hole in the bottom of the handle shown at right as the **Primary reference entity** .



5. Set the **Mate Reference Type** to **Concentric**, and the **Mate Reference Alignment** to **Aligned**.
6. For the **Secondary reference entity,** select the flat face on the bottom of the handle and set the **Mate Reference Type** to **Coincident**, and the **Mate Reference Alignment** to **Anti-Aligned**.
7. For the **Tertiary reference entity**, select the flat face of the hole in the bottom of the handle and set the **Mate Reference Type** to **Parallel**, and the **Mate Reference Alignment** to **Anti-Aligned**.
8. Click **OK** .

   Notice that the **MateReferences** folder in the FeatureManager design tree contains the new mate reference.
9. Save the part.

To add a part with a mate reference to an assembly:

1 Open **\faucet\faucet_assembly.sldasm** and tile the windows.

2 Drag the **faucet_handle** component from the top of its FeatureManager design tree into the assembly window.

A preview of the component in its correct position appears as soon as your pointer moves into the assembly window. You do not have to move the pointer over the **faucet_stem** component because there is only one mate reference in the assembly with the same name and combination of mates as in the dragged component.

3 Drop the **faucet_handle** component into the assembly and expand the **Mates** mategroup in the FeatureManager design tree.

Notice the concentric, coincident, and parallel mates between the **faucet_stem** and **faucet_handle** components. The SolidWorks software adds these mates because of the matching mate references in each component.

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

Mirror Components

In SolidWorks 2003, the software recreates more mates between instanced components than it did in earlier versions.

An instanced component's geometry is identical to the original component; only the orientation of the instanced component is different.

To mirror components:

1 Open **\faucet\faucet_assembly.sldasm**.

If you have not completed the **Mate References** section, you must do so before proceeding.

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

The **Mirror Components** PropertyManager appears.

3 Select the **Right** plane in the assembly as the **Mirror plane** and select the **faucet_stem** and **faucet_handle** components as the **Components to Mirror**.

Use the flyout FeatureManager design tree to select the plane and the components. You access the flyout FeatureManager design tree by clicking the title in the PropertyManager.

4 Select the **Recreate mates to new components** check box.

5 Click **Next**, then click **OK** to add the components and the mates.

Notice in the FeatureManager design tree under **Mates** that there are three new mates between the two instanced components, **faucet_stem<2>** and **faucet_handle<2>**. These mates include a concentric, a coincident, and a parallel mate.

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

Replace Components in an Assembly

There are a number of enhancements to the replace functionality, including:

- **Component choices**. When you want to replace a component in an assembly you can now replace a part with a sub-assembly and vice versa. In earlier versions, you had to replace a part with a part and a sub-assembly with a sub-assembly.
- **PropertyManager**. The replace functionality is now in the **Replace Components** PropertyManager.

> The **Reload** function is the same as it was in earlier versions except that the dialog name is now **Reload**, not **Reload/Replace**.

- **Function access**. Click **File**, **Replace** or right-click a component and select **Replace** to access the PropertyManager. You can no longer access the replace function through the **Component Properties** dialog box.
- **Instances**. You can replace one, more than one, or all instances of a component at the same time.
- **Configurations**. You can manually select the configuration of the replacement component, or you can let the software automatically select the configuration for you. The software tries to match the configuration name of the old component with a configuration in the replacement component. If a match is not found, the last saved configuration of the replacement component is used.

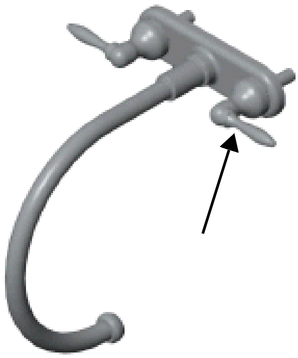
To replace a part with a sub-assembly:

1 Open **\faucet\faucet_assembly.sldasm**.

If you have not completed the **Mirror Components** section, you must do so before proceeding.

2 Click **Replace** on the Assembly toolbar, or click **File**, **Replace**.

The **Replace** PropertyManager appears.



3 Select the **faucet_handle** component shown at right for the **Replace these component(s)** box.

4 Click **Browse**, select **\faucet\handle.sldasm**, and click **Open**.

The replacement component name appears in the **With this one** box.

5 Ensure that the **Re-attach mates** check box is selected then click **OK**.

The **faucet_handle** component is replaced by the **handle** sub-assembly. Only one instance of the **faucet_handle** component is replaced because you did not select both instances for the **Replace these component(s)** box, nor did you select the **All instances** check box.

Note that there are some dangling mates because the mates do not find matching entities. You will correct these in the next section.

6 Close the **Mated Entities** PropertyManager.

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

Replace Mate Entities

The **Mated Entities** PropertyManager helps you reattach dangling mate entities. You can list all of the mated entities in the assembly or in a particular component. Then you can replace any of the mated entities to satisfy the mates. To replace mated entities in earlier versions of the software, you had to edit the definition of each mate individually then replace the entities.

> If you have dangling mates as the result of the **Replace Components** function, the **Mated Entities** PropertyManager appears automatically.

To replace dangling mate entities:

1 Open **\faucet\faucet_assembly.sldasm**.

If you have not completed the **Replace Components in an Assembly** section, you must do so before proceeding.

2 Select the **handle** sub-assembly then click **Replace Mate Entities** on the Assembly toolbar, or right-click the **handle** sub-assembly and select **Replace Mate Entities**.

The **Mated Entities** PropertyManager appears. There are three dangling mate entities in the list. A dangling mate entity is shown with a ✕ in front of the mate entity.

> To view the mate entities in the entire assembly instead of a single component, you can right-click the **Mates** mategroup and select **Replace Mate Entities**.

3 Click the ⊞ in front of the first dangling mate entity in the list to reveal the dangling concentric mate.

4 Select the circular face of the hole shown on the bottom of the **handle** sub-assembly as the **Replacement mate entity**.

The **handle** sub-assembly moves into position to satisfy the concentric mate using the replacement mate entity.

5 Continue selecting the other faces on the **handle** sub-assembly to repair the coincident and parallel mates.

> You can use the **Move Component** and **Rotate Component** tools while the **Mated Entities** PropertyManager is open to position the components for easier selection of the entities.

6 Click **OK**.

The **handle** sub-assembly no longer has any dangling mate entities and its components are aligned properly.

Physical Simulation

Physical Simulation allows you to simulate the effects of motors, springs, and gravity on your assemblies. Physical Simulation combines new simulation elements with existing SolidWorks tools such as mates and Physical Dynamics to move components around your assembly. The new simulation elements include:

- **Linear Motors** - Select an entity to define the direction of motion, and move the slider to control the speed of the movement.
- **Rotary Motors** - Select an entity to define the direction of rotation, and move the slider to control the speed of rotation.
- **Springs** - Select two entities for the spring endpoints. Control the direction of movement by varying the free length of the spring and the strength of the spring by varying the spring constant.
- **Gravity** - Select an entity as the direction of gravitational pull, and vary the strength of the gravitational force.

For step-by-step instructions for linear motors, springs, and gravity, as well as hints for using this functionality, please refer to Physical Simulation in the *SolidWorks 2003 Online User's Guide*.

To add a rotary motor to an assembly:

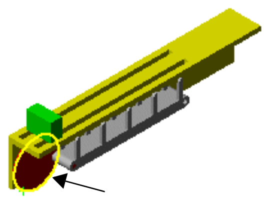
1 Open **conveyor.sldasm**.

2 Click **Simulation Toolbar** on the Assembly toolbar.

The Simulation toolbar appears.

3 Click **Rotary Motor** on the Simulation toolbar.

The **Rotary Motor** PropertyManager appears.



4 Select the circular edge on the red disk, click **Reverse Direction**, and click **OK**.

5 Click **Record Simulation** on the Simulation toolbar to begin the simulation.

The disk begins to spin due to the rotational movement of the rotary motor. The mates between the disk and the teeth cause the teeth to move. Finally, Physical Dynamics causes the teeth to move the block down the track.

6 After a few revolutions, click **Stop Record or Playback** on the Simulation toolbar.

7 Click **Replay Simulation** on the Simulation toolbar to move the block again.

The block component appears in its original position and the simulation replays.

When replaying a simulation, the block component does not actually move to its original location Temporarily, the block appears in its original location in the graphics area. The replay of the simulation is for graphical purposes only. To return the block to its original position after recording a simulation, click **Reset Components**.

7

Design Tables

This chapter describes enhancements to design tables in the following areas:

- General enhancements
- Automatically create design tables
- Automatically add rows and columns to design tables
- Bi-directional design tables
- Linked design tables
- Design table parameters

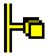
General Enhancements

Toolbar Button

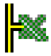
Design Table is a new button on the Tools toolbar. Use this button to insert a design table.

ConfigurationManager

New icons are displayed in the ConfigurationManager based on how the configuration was created: manually or with a design table.



Icon for configurations created manually



Icon for configurations created with a design table

Dimensions

Dimensions that are controlled by design tables can now be shown in a different color.

You can change the color of dimensions that are controlled by design tables in **Tools**, **Options**, **System Options**, **Colors**.

Select **Dimension, Controlled by Design Table** from the **System colors** list, and change the color.

Design Table PropertyManager

There is a **Design Table** PropertyManager where you can specify design table options. You can set options to automatically create a design table (see **Automatically Create Design Tables** on page 7-3), automatically update design tables (see **Automatically Add Rows and Columns to Design Tables** on page 7-4), and so on.

Save Design Tables

You can now save design tables from within the SolidWorks software. Previously, you could not directly save design tables.

To save a design table:

1. In a document with a design table, click **Design Table** in the FeatureManager design tree, then click **File**, **Save As**.

   - or -

   Right-click **Design Table** in the FeatureManager design tree and select **Save Table**.

   The **Save Design Table** dialog box appears.
2. Type a **File name**, then click **Save**.

   The design table is saved as a separate Excel file (***.xls**).

Automatically Create Design Tables

You can have the SolidWorks software automatically create a new design table. When SolidWorks automatically creates a new design table, it loads all configured parameters and their associated values from a part or assembly.

To automatically create a design table:

1. Open **cog.sldprt**.
2. Click **Design Table** on the Tools toolbar, or click **Insert**, **Design Table**.

   The **Design Table** PropertyManager appears.
3. Under **Source**, click **Auto-create**.
4. Click **OK**.

   The design table appears with parameters and values from the part.
5. Click an empty space in the graphics area to close the design table.
6. Save the part.
7. Keep **cog.sldprt** open for the next procedure.

Automatically Add Rows and Columns to Design Tables

In the **Design Table** PropertyManager, you can set options to automatically add new rows and columns to a design table. For example, if you add a new configuration manually, SolidWorks can automatically add the new feature parameters to the design table when you re-open the design table.

SolidWorks adds the rows and columns based on the options you select.

To automatically add new rows and columns to a design table:

1. Open **cog.sldprt** if you do not have it open from the previous procedure.
2. In the FeatureManager design tree, right-click **Design Table** and select **Edit Definition**.
3. In the **Design Table** PropertyManager, under **Options**, make sure the following check boxes are selected:
   - **New parameters**. SolidWorks adds new rows and columns to the design table if you add a new parameter to the model.
   - **New configurations**. SolidWorks adds new rows and columns to the design table if you add a new configuration to the model.
4. Click **OK**.
5. In the ConfigurationManager, add a new configuration to the model by right-clicking **cog Configuration(s)** and selecting **Add Configuration**. Type a **Configuration Name**, then click **OK**.
6. In the FeatureManager design tree, right-click **Design Table**, and select **Edit Table**.

   The **Add Rows and Columns** dialog box appears.
7. Under **Configurations**, select the configuration you added in step 4, then click **OK**.

   The new configuration is added to the design table.
8. Keep **cog.sldprt** open for the next procedure.

Bi-Directional

Design tables in SolidWorks are now bi-directional. Changes made to a model can now propagate back *to* the design table. Previously, you could only update a model from a design table.

You can control the way that models and bi-directional design tables update in the **Edit Control** section of the **Design Table** PropertyManager.

To set up the options for a bi-directional design table:

1. Open **cog.sldprt** if you do not have it open from the previous procedure.
2. In the FeatureManager design tree, right-click **Design Table** and select **Edit Definition**.

   The **Design Table** PropertyManager appears.
3. Under **Edit Control**, make sure **Allow model edits to update the design table** is selected.
4. Click **OK**.
5. In the ConfigurationManager, double-click **Simplified** to switch to that configuration.
6. In the FeatureManager design tree, do the following:
   - Right-click **Design Table** and select **Edit Table**.

     Notice that **$STATE@Chamfer1** for the **Simplified** configuration is set to **S** for suppressed.
   - Click an empty space in the graphics area to close the design table.
   - Right-click **Chamfer1** and select **Unsuppress**.

     A message appears that says the corresponding cell in the design table will update the next time it is edited.
7. Click **OK**.
8. In the FeatureManager design tree, right-click **Design Table** and select **Edit Table**.

   Notice that **$STATE@Chamfer1** for the **Simplified** configuration is set to **U** for unsuppressed.

Linked Design Tables

You can link a design table to a SolidWorks file in the **Design Table** PropertyManager. This way, if you externally update the file in Microsoft Excel, the SolidWorks software updates the design table in the model.

Linked design tables are different from bi-directional design tables, in that a linked table reads its data from an external Excel file.

To link a design table to a SolidWorks file:

1. Open **design_table.sldprt**.
2. Click **Design Table** on the Tools toolbar, or click **Insert**, **Design Table**.

   The **Design Table** PropertyManager appears.
3. Under **Source**, do the following:

   a) Click **From file**.

   b) Click **Browse**, and open **design_table.xls**.

   c) Select the **Link to file** check box.
4. Click **OK**.

   The design table is linked to the SolidWorks file.
5. Click anywhere in the graphics area, but outside of the design table and model.
6. Click **Save**.

   A progress dialog box shows that the design table is also being saved.

If you update a linked design table in Microsoft Excel, then open the SolidWorks model, you can choose to update either:

- the model with the design table values

  \- or -
- the design table with the model values

You can set the update options in **Tools**, **Options**, **System Options**, **External References**. Set **Update out-of-date linked design tables to** to **Prompt**, **Model** or **Excel file**.

Design Table Parameters

Base Parts

Design tables can control the configuration of a base part. The parameter to control a base part configuration is **$CONFIGURATION@**<*part name*>, where part name is the name of the base part. Row values for this parameter are the base part's configuration names.

For example, to use the default configuration of a base part named **washer.sldprt**, the column heading syntax is **$CONFIGURATION@washer**. The row value is **Default**.

Component Configurations

The **$CONFIGURATION** parameter has been expanded. If you leave the value blank, SolidWorks uses the component's "in-use" or last saved configuration. Previously, you could not leave the **$CONFIGURATION** parameter blank.

If the component uses a derived configuration, and the **$CONFIGURATION** value is left blank, then the configuration referenced is linked from its parent.

Derived Configurations

You can create derived configurations in a design table. The parameter for derived configurations in a design table is **$PARENT**. Row values for this parameter are the parent configuration names.

You cannot specify a parent configuration in a design table if its child configuration was created first, unless the parent configuration already existed in the model.

	A	B
1	Design Table for: Part1	
2		$PARENT
3	configA	
4	configB	configA
5	configC	

Valid: Parent created before derived configuration

	A	B
1	Design Table for: Part1	
2		$PARENT
3	configA	configC
4	configB	
5	configC	

Invalid: Parent created after derived configuration

Equations

You can suppress equations in a design table. The parameter for equations in a design table is **$STATE@**<*equation number*>**@EQUATIONS**.

For example, to control the suppression state of the first equation in a model, the column heading syntax is **$STATE@<1>@EQUATIONS**.

Lighting

Light properties in the **Lighting** folder can be suppressed in a design table. The parameter for light properties in a design table is **$STATE@**<*lighting name*>.

For example, to control the suppression state of directional light, the column heading syntax is **$STATE@Directional1**.

Part Number

The **$PARTNUMBER** parameter now includes the document name or the parent name (derived configurations only) in a bill of materials. In previous SolidWorks releases, if you left the **$PARTNUMBER** row blank, the configuration name was used. If you typed any text, a custom name was used.

In SolidWorks 2003, the following row values can be used for **$PARTNUMBER**:

Value	Property used
$DOCUMENT	Document name
$PARENT	Parent configuration name
$CONFIG	Configuration name
any text	Custom name
blank	Configuration name

Sketch Relations

You can now suppress sketch relations. Sketch relations and sketch entities include an assigned number in the **Sketch Relations** PropertyManager. For example, a coincident relation is now labeled **Coincident**<*number*>, such as **Coincident1**.

The parameter to suppress sketch relations in a design table is **$STATE@**<*sketch relation*>@<*sketch name*>. For example, to control the suppression state of the first fixed relation in **Sketch2**, the column heading syntax is **$STATE@Fixed1@Sketch2**.

8

Drawings and Detailing

This chapter describes the enhancements to drawings in the following areas:

- Predefined views
- RapidDraft functionality
- Break lines
- Silhouette edges
- Dimensions
- Annotations
- Layers
- Blocks
- Fast HLR/HLV in drawings

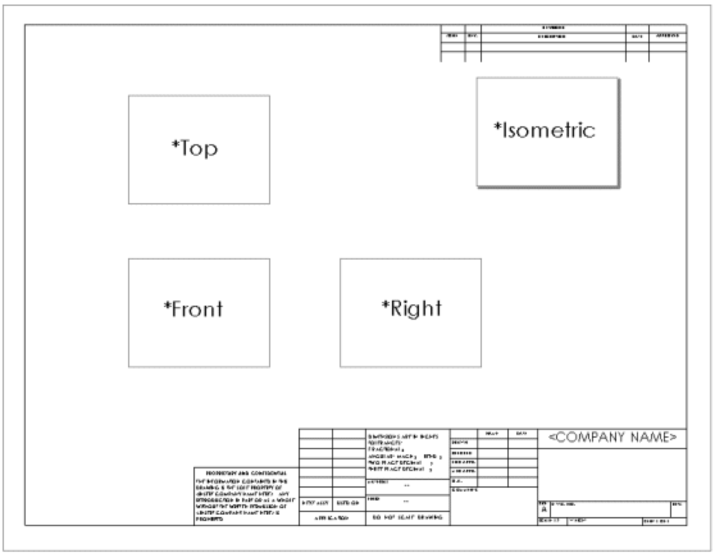
Predefined Views

You can define any orthogonal, projected, or named view in a drawing sheet and populate the views by dragging an open model into the drawing, by selecting a model from a list of open files, or by browsing for a model file. You can save a document with predefined views as a document template.

To create a drawing with predefined views:

1. Open **two_bolt_flange.sldprt**.
2. Create a new drawing document from the **Tutorial** tab.
3. Right-click in the graphics area and select **Properties**.
4. In the **Sheet Setup** dialog box, set **Scale** to **1:1**.
5. Click **Predefined View**, or **Insert**, **Drawing View**, **Predefined**.
6. Click in the graphics area to place a front view as shown, selecting ***Front** from the **View Orientation** list in the **Predefined View** PropertyManager. (The names in the figure have been added to the graphic for clarity and do not appear in your drawing.)
7. Click **Projected View** and project the ***Front** view above and to the right as shown. The ***Top** and ***Right** orientations are selected for you automatically.
8. Copy the ***Front** view and paste it at the upper right, selecting ***Isometric** orientation.

   Now the ***Front**, ***Right**, and ***Top** views are aligned as Standard 3 Views in third angle projection, and the ***Isometric** view remains unaligned.

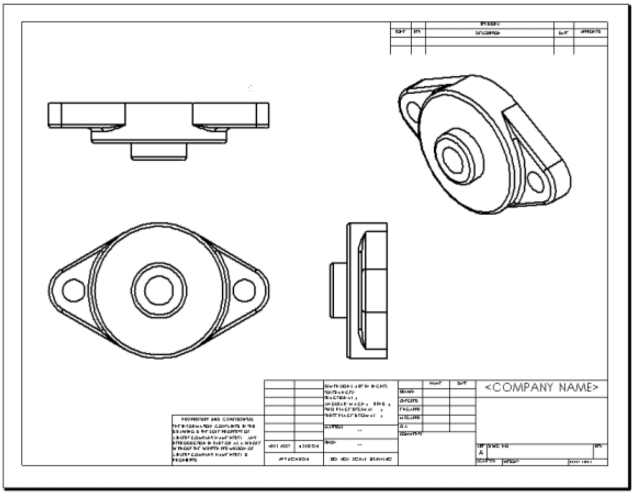


You can save the template as type **Drawing Templates (*.drwdot)** for use as the basis for future drawing documents.

To populate the predefined views:

1. Select the ***Front** view.
2. In the **Predefined View** PropertyManager, under **Insert Model**, select **two_bolt_flange.sldprt** from the list and click **OK** .

   The three related views are populated.
3. Select the ***Isometric** view.
4. In the **Predefined View** PropertyManager, under **Insert Model**, select **two_bolt_flange.sldprt** from the list and click **OK** .



RapidDraft Functionality

You can now use the following items in RapidDraft drawings without loading the model:

- Bill of Materials
- Balloons
- Shaded mode
- Delete break lines

Now no information on bodies is loaded when you open a RapidDraft drawing with more than a few parts, and you cannot select entities (edges, faces, and so on). When you hold the pointer over a body face, information on that body for all views is loaded and you can select its entities. This procedure results in faster loading times for RapidDraft drawings.

You can show RapidDraft drawing views in **Wireframe** and **Hidden Lines Visible** display modes. You can show RapidDraft views in **Shaded** mode if the views have previously been saved in **Shaded** mode. If any view is saved in **Shaded** mode, then all views of the same model can be shown in **Shaded** mode, and new views of the same model can be shown in **Shaded** mode.

Break Lines

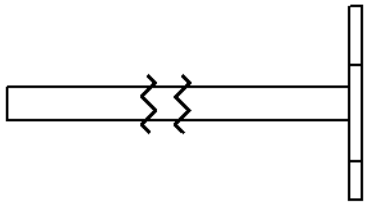
Break lines now extend only as far as the geometry in the drawing view rather than to the view border. You can specify how far the break lines extend beyond the geometry in **Detailing Options** and set the default line font in **Line Font Options**. You can also add break lines to layers.

To add break lines to a drawing and specify break lines options:

1 Open **drw_break_lines.slddrw**.

2 Click **Tools**, **Options**, **Document Properties**, **Line Font**.

3 Under **Type of edge**, select **Break Lines**. Note the default **Style** (**Solid**) and **Thickness** (**Thick**) and click **OK**.



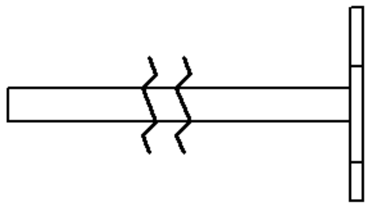
4 Click in the drawing view approximately where you want the break lines to appear.

5 Click **Insert**, **Vertical Break**.

The vertical break lines extend only a short distance from the edges of the part.

6 Click **Tools**, **Options**, **Document Properties**, **Detailing**.

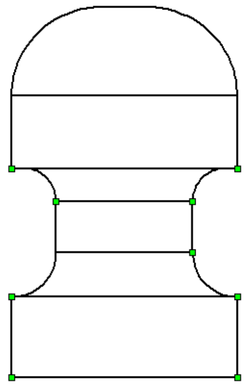
7 Under **Break line**, set the **Extension** value to **9mm** and click **OK**.



The break lines now extend 9mm beyond the edges of the part.

Silhouette Edges

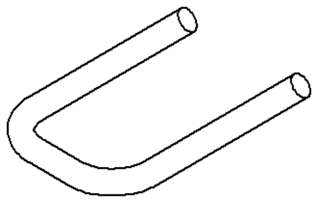
You can now select a vertex where a silhouette edge meets a non-silhouette edge. Selecting edges and vertices in drawings is useful for dimensioning.

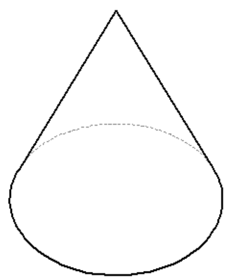


Vertices such as those shown at right are now available for selection.

You can also select vertices on silhouette edges in sketches. See **Silhouettes on page 2-3**.

Other types of silhouette edges that you can now select in drawings include tubes with 180° bends and the silhouettes on conical surfaces.



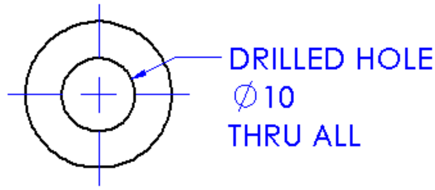


Dimensions

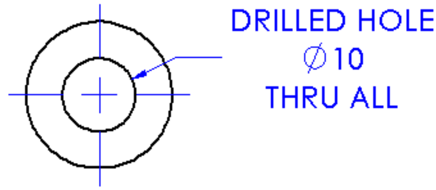
Justify Dimensions

You can now justify a multi-line dimension vertically with the leader (top, middle, bottom) for some standards, such as ANSI, and justify dimension text horizontally (left, center, right). Justification is also available for **Dimension Favorites**.

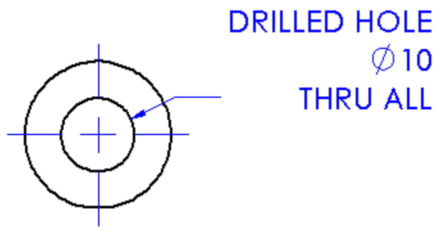
You can set the justification defaults in **Tools**, **Options**, **Document Properties**, **Dimensions**, **Text alignment**.



Justified left and top



Justified center and middle



Justified right and bottom

To justify dimension text:

1. Open **drw_dims_justify.slddrw**.
2. Select the dimension.

   The dimension text is justified left and middle.
3. In the **Dimension** PropertyManager, do the following:
   - **a)** To change the horizontal text justification, click **Justify Center** or **Justify Right**.
   - **b)** To change vertical justification to the leader, click **Top Justify** or **Bottom Justify**.

Fit Tolerances

You can now select a class (Clearance, Transitional, Press, or None) of Fit tolerances. The lists available for **Hole Fit** and **Shaft Fit** are determined by the classification.

If you choose a Fit tolerance with bilateral tolerances, you can let the SolidWorks software calculate the tolerance values for you or click the **Hole Fit** and **Shaft Fit** buttons to set the tolerances manually. The advantage of letting the software calculate the tolerances is that if the dimension changes, the tolerances are updated automatically. You can also choose to show the dimension and the tolerances only.

If the precision display in the following example does not show the tolerance completely, click **More Properties** in the **Dimension** PropertyManager. In the **Dimension Properties** dialog box, clear **Use document's precision**, click **Precision**, and increase the number of digits for **Tolerance** under **Primary units**.

To show a dimension with hole Fit tolerance:

1 Open **drw_dims_fit.slddrw**.

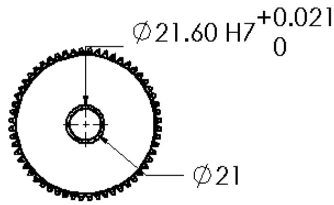
2 In the **Front** view of the gear, select the top dimension, which is the hole dimension.

3 In the **Dimension** PropertyManager, under **Tolerance/Precision**, do the following:

   a) Select **Fit with tolerance** from the **Tolerance Type** list.

   b) Select **Clearance** from the **Classification** list.

   c) In **Hole Fit**, select **H7**.



The Fit tolerance with bilateral tolerance values appears on the dimension as shown.

The tolerances are calculated automatically. To specify the tolerances manually, click **Hole Fit** and type values in the **Maximum Variation** + and **Minimum Variation** – boxes.

To show a dimension with a shaft Fit tolerance:

1 Select the lower dimension, which is the shaft dimension.

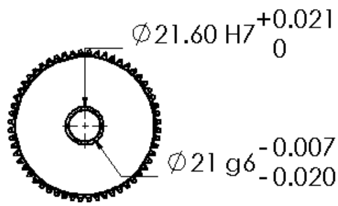
2 In the **Dimension** PropertyManager, under **Tolerance/Precision**, do the following:

   a) Select **Fit with tolerance** from the **Tolerance Type** list.

   b) Select **Clearance** from the **Classification** list.

   c) In **Shaft Fit**, select **g6** from the list.

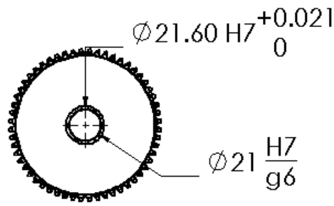
   The list is restricted to shaft tolerances that are compatible with the selected Hole tolerance.



To show two Fit tolerances on one dimension:

1 With the shaft dimension still selected, select **H7** from the **Hole Fit** list.

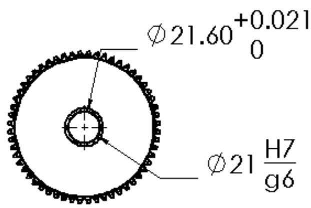
2 Click **Stacked with line display**.



To show a Fit dimension with tolerance only:

1 Select the Hole dimension.

2 In **Tolerance Type**, select **Fit (tolerance only)** to display the bilateral tolerances for the hole.

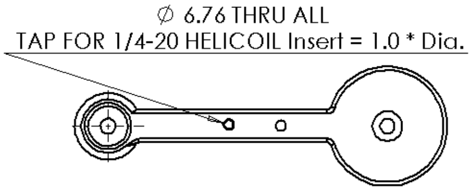


Hole Callouts

Hole Callouts now use Hole Wizard attribute information, and you can edit the text.

To create a hole callout using the Hole Wizard information:

1 Open **drw_hole_callout.slddrw**.

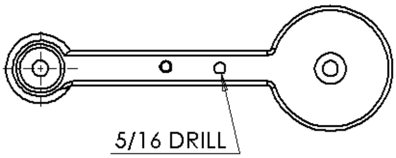


2 Click **Hole Callout** on the Annotations toolbar or click **Insert**, **Annotations**, **Hole Callout**.

3 Select the tapped hole in the center of the part, drag the callout into position, and click to place it. Note the Hole Wizard information in the hole callout.

4 With **Hole Callout** still active, select the drilled hole to the right of the tapped hole, drag the callout into position, and click to place it.

5 In the **Dimension** PropertyManager, under **Dimension Text**, delete the text. Type **DRILL**. Place the pointer before the word **DRILL** and click **Variables** Variables. Select **Fastener Size** from the list and click **OK**.



The fastener size information from the Hole Wizard appears in the Hole Callout.

6 Click **OK**.

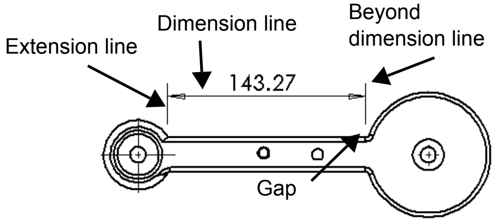
You can toggle between Hole Wizard and geometric information in the Hole Callout. Right-click the Hole Callout and select **Display Options**, **Define by Geometry** or **Define by Hole Wizard**.

The default formats for the Hole Wizard types are stored in *installation directory*\lang\<language>\calloutformat.txt. A second file, **calloutformat_2.txt**, is a simplified version. You can edit either file. If you want to use the second file, you must appropriately rename the file to **calloutformat.txt**, which the SolidWorks software references. You can set the default folder for **Hole Callout Format File** in **Tools**, **Options**, **System Options**, **File Locations**.

Extension Lines

The term **witness line** has been replaced with **extension line** to reflect industry terminology.

The term **Extension** in **Tools**, **Options**, **Document Properties**, **Detailing** has been replaced with **Beyond dimension line**.



Annotations

Font Control

You can now control the font type and size separately for various annotations. Click **Tools**, **Options**, **Document Properties**. Under **Detailing**, click **Annotations Font**. You can choose separate default fonts for the following annotations:

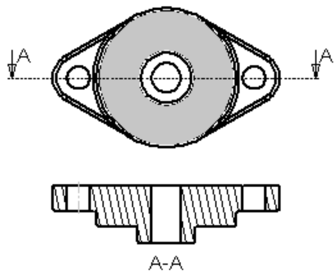
- Note/Balloon
- Dimension
- Detail
- Section
- View Arrow
- Surface Finish
- Weld Symbol

You can now also control Weld Symbol and Surface Finish Symbol text font individually in the symbol dialog boxes.

Solid Color Fill of Area Hatch and Crosshatch

Area Hatch and Crosshatch can now be a solid pattern. For Area Hatch, you can control the color with the Line Format **Color** tool. You can assign Area Hatch and Crosshatch to layers. You can set the default pattern to either **None**, **Hatch**, or **Solid** in **Tools**, **Options**, **System Options**, **Area Hatch/Fill**.

To add solid color to areas in drawings:



1 Open **drw_crosshatch_solid.slddrw**.

A section view and its parent view are shown.

2 In the parent view, select the circular face.

3 Click **Area Hatch/Fill** on the Drawing toolbar, or click **Insert**, **Area Hatch/Fill**.

4 In the **Area Hatch/Fill** dialog box, click **Solid**.

Preview shows a solid color and **Pattern**, **Scale**, and **Angle** are no longer available. You can clear **Always show dialog on creation** to prevent the dialog box from appearing each time you insert an Area Hatch.

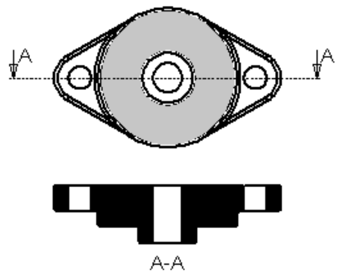
5 Click **OK**.

The circular face displays a solid color.

6 With the circular face still selected, click **Line Color** on the Line Format toolbar. Choose a color and click **OK**.

7 Click outside the circular face to see the new color.

To change crosshatching to solid color fill:



1. With **drw_crosshatch_solid.slddrw** still open, right-click a crosshatched section in the section view and select **Properties**.

   The **Area Hatch/Fill** dialog box appears.
2. Under **Properties**, click **Solid**.
3. In the **Apply to** box, select **View**, and click **OK**.

   For the automatically generated crosshatching in section views in part drawings, you can apply changes to the selected region, or to all crosshatched faces in the view. In assembly drawings, you can also apply changes to the crosshatches in a selected component.
4. Click **Layer Properties** on the Layer or Line Format toolbar.
5. In the **Layers** dialog box, click **Layer1** to make it the active layer.
6. Select a crosshatch in the section view and click **Move** in the **Layers** dialog box.

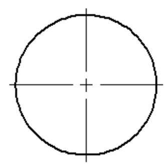
   The crosshatch color changes to that of the layer.

Center Marks and Centerlines

You can now have center marks or centerlines, or both, automatically inserted into drawings when you create a drawing view. You can add a center mark to a hole in a pattern and propagate the center mark throughout the pattern with connection lines.

Centerline is a new Annotation. Centerlines are added automatically to all appropriate entities when you select the automatic option. You can also manually add centerlines on cylindrical, conical, toroidal, and swept faces and between parallel and non-parallel edges. If you select a view, you can add centerlines to all appropriate entities in one step.

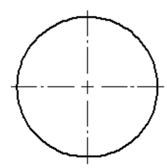
To specify automatic insertion of center marks or centerlines:



Center mark without centerline font (ANSI standard)

1. Create a new drawing document from the **Tutorial** tab.
2. Click **Tools**, **Options**, **Document Properties**, **Detailing**.
3. Under **Auto insert on view creation**, select both **Center marks** and **Centerlines** and click **OK**.

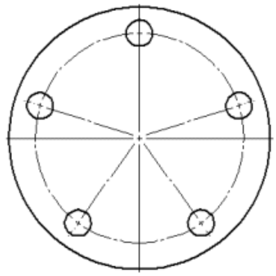
Center Mark styles now include **Single Center Mark**, **Linear Center Mark** (for linear patterns), and **Circular Center Mark** (for circular patterns). When holes are in line with each other, connection lines between the holes indicate their relationships.



Center mark with centerline font (ISO standard)

You can specify the **Centerline font** for Center Marks in either the **Center Mark** PropertyManager or in **Tools**, **Options**, **Document Properties**, **Detailing**.

To insert center marks and centerlines automatically:



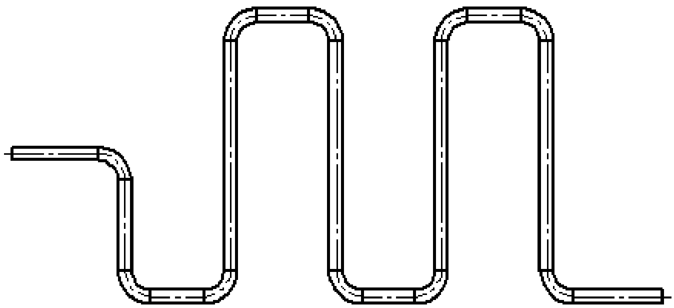
1 Insert a Named View, ***Front** orientation, of **drw_center_marks_circular.sldprt**.

   The patterned holes show the **Circular Center Marks**, including connection lines between the holes.

2 Select a center mark and select **Radial lines** under **Options** in the **Center Mark** PropertyManager.

3 Insert a Named View, ***Top** orientation, of **drw_centerline_manifold.sldprt**.

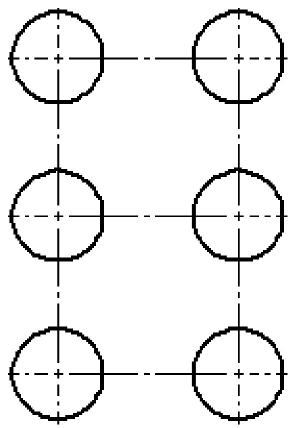
   Centerlines appears in all segments of the tube.



To propagate center marks in a pattern:

1 Click **Tools**, **Options**, **Document Properties**, **Detailing**.

2 Under **Auto insert on view creation**, clear **Center marks** and click **OK**.



3 Insert a Named View, ***Front** orientation, of **drw_center_marks_rectangular.sldprt**.

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

5 In the **Center Mark** PropertyManager, click **Linear Center Mark**, select **Connection lines**, and select a hole in the rectangular block.

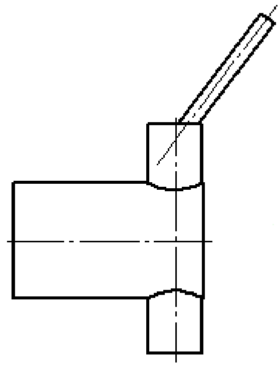
   A center mark appears with a **Propagate** button.

6 Click **Propagate** .

   The center mark propagates to all the holes in the pattern, with connection lines.

7 Click **OK** .

To insert centerlines manually:

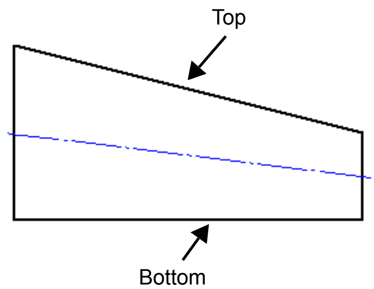


1. Click **Tools**, **Options**, **Document Properties**, **Detailing**.
2. Under **Auto insert on view creation**, clear **Centerlines** and click **OK**.
3. Insert a Named View, ***Front** orientation, of **drw_centerlines_revolve.sldprt**.
4. Press **Ctrl**, select two of the cylindrical faces, and click **Centerline** on the Annotations toolbar, or click **Insert**, **Annotations**, **Centerline**.

   Centerlines appear in the selected cylindrical features.
5. With the **Centerline** tool still active, select the view.

   Now centerlines appear in all three features, with no duplications.
6. Insert a Named View, ***Front** orientation, of **drw_centerlines_trapezoid.sldprt**.
7. Click **Centerline** on the Annotations toolbar, or click **Insert**, **Annotations**, **Centerline**.
8. Select the top and bottom lines of the trapezoid as shown.

   A centerline appears from the midpoint of the left line to the midpoint of the right line. You can add centerlines to many shapes, but not to isometric views or to splines.



Links to Properties in Notes

You can now link to a property for a **Component to which the annotation is attached**, in addition to previous choices of **Current document**, **Model in view to which the annotation is attached**, and **Model in view specified in sheet properties**. Also, configuration names are now included in the list of custom properties.

To link a note to a property of an assembly component:

1. Open **drw_dims_fit.slddrw** and **drw_dims_fit_gear.sldprt**.
2. In the part document, click **File**, **Properties**.
3. In the **Summary Information** dialog box, select the **Custom** tab. Under **Properties**, note the variable **Teeth** and its **Value**, then click **Cancel**.
4. In the drawing document, click **Note**, or click **Insert**, **Annotations**, **Note**.

5 Click the gear in the **Front** view to place the leader, then click in the graphics area to place the note.

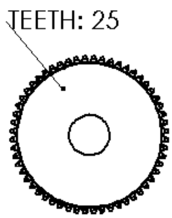
6 Type **TEETH:**, then click **Link to Property** in the **Note** PropertyManager.

7 In the **Link to Property** dialog box, select **Component to which the annotation is attached**.

When you choose this selection, the custom properties for the selected component are available in the list of properties along with the SolidWorks system properties.

8 Select **Teeth** from the list of properties and click **OK**.

The note shows the custom property with a new system variable, **$PRPMODEL**, that connects the property to the selected model component, and the variable name. The system variable and variable name are replaced by the value of the variable when you exit from editing.



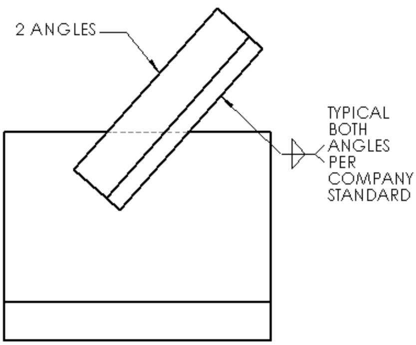
9 Click outside the note, then click **OK** .

The note now displays your typed text plus the value of the custom property in the component variable.

Weld Symbols

You can now have more than three lines of text for the specification process in a Weld Symbol.

In addition to setting the default font for **Weld Symbols** in the **Annotations Font Options**, you can also control the text font individually in the **Weld Symbol** dialog box.



Layers

You can now add the following items to layers:

- Cosmetic threads
- Section lines
- Break lines
- Detail circles
- Area hatch
- Crosshatch
- Blocks

Blocks

Blocks can now:

- Include note leaders and borders
- Have a base insertion point
- Link to external files
- Be inserted as instances
- Move onto layers
- Snap to grid

Blocks now have *definitions* and *instances*. Edits to definitions apply to all instances of the same block, but changes to instances apply only to the selected instance. Examples of changes to instances include leader display, scale, angle, attribute display, and attribute values. When you save blocks to file, the scale and rotation angle information are saved with the block definition.

The block editor now opens the definition on a temporary sheet already exploded. The block automatically becomes a block again when you close the editor. In addition to the editor for block definitions, the **Block Instance** PropertyManager has a new attribute value edit utility.

You can also edit blocks in files. Click **Tools**, **Block**, **Edit File**.

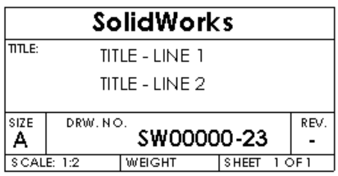
When you add blocks to drawing documents, the block definitions are stored in a **Blocks** folder in the FeatureManager design tree.

You can copy an instance of a block by pressing **Ctrl** while dragging the block.

Block files now have the file name extension **.sldblk**. The SolidWorks software still supports **.sldsym** for inserting blocks and editing blocks, but all new blocks saved into external files use the **.sldblk** extension.

For information on importing blocks, see **DXF/DWG Files** on page 9-7.

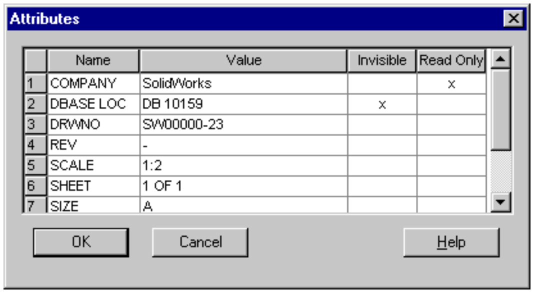
To insert a block instance:



1 Open a new drawing document from the **Tutorial** tab.
2 Click **Insert Block** or **Insert**, **Block**.
3 In the **Block** PropertyManager, under **Source**, click **Browse** and browse to **drw_block.sldblk.**
4 Click **Open**, then click in the graphics area to place the title block.
5 Click **OK**.

   In the FeatureManager design tree, note the **Blocks** folder and the block definition name (**drw_block**). The block definition is created automatically when you insert a block instance. You can insert as many instances at a time as you want.

To edit a block instance:

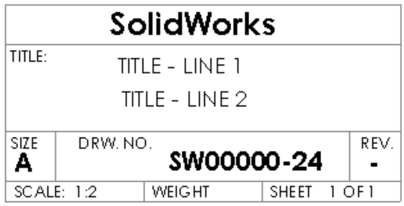


1 Select the block instance in the graphics area.

2 In the **Block Instance** PropertyManager, under **Block Display**, click the up arrow to increase the **Scale** to **1.2**.

3 Under **Text Display**, click **Attributes**.

The **Attributes** dialog box contains columns for the attribute **Name**, **Value**, **Invisible**, and **Read Only**. **Invisible** and **Read Only** are for your information. **Value** is the column that you edit.



4 In row 3, **DRWNO**, change **SW00000-23** to **SW00000-24**, click **OK**, and click **OK**.

5 Click **Insert Block**, select **drw_block** from the block definition list, and click in the graphics area to place another instance of the block.

The new instance of the block retains the size and text of the block definition. The changes to the first instance do not apply to new instances.

6 Click **OK**.

To edit a block definition:

1 Select the block in the FeatureManager design tree.

2 Click **Tools**, **Block**, **Edit Definition**.

The block appears, already exploded, on a temporary drawing sheet. Notice the text (**DB 10159**) that is invisible in the drawing.

You can also edit the block definition as follows: Right-click an instance in the graphics area or the definition in the FeatureManager design tree and select **Edit Definition**.

You can edit the definition in the block file. Click **Tools**, **Block**, **Edit File**.

3 Click **Note** and place a note without a leader on the right side of the block. Type **NOTE:**, click outside the note, and click **OK**.

If you select the **Link to file** check box under **External Reference** in the **Block** PropertyManager, changes to the external file update any instances of the block in the current document.

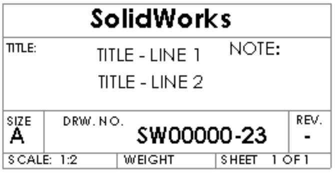
You can also move the block base point by dragging the two orthogonal arrows, or by entering the X and Y positions of the point in the **Block** PropertyManager.

4 Click **OK** to close the **Block** PropertyManager.

5 Click **Insert Block** , select **drw_block** from the list, and click in the graphics area to place another instance of the block.

The new instance of the block displays the new note.

6 Click **OK** .



To move a block instance onto a layer:

1 Select a block instance.

2 Click **Layer Properties** on the Layer or Line Format toolbar.

3 Create a new layer and click **Move**.

The block instance retains its line color, style, and thickness properties, but you can hide and show the block by hiding and showing the layer.

4 Close the **Layer Properties** dialog box.

When you add items (sketch elements and notes) to blocks, color and line font are retained, but the layer name for individual entities is removed automatically. When the block is exploded, all the items have a layer of **None**, and all physical attributes are retained.

Fast HLR/HLV in Drawings

Fast HLR/HLV on the View toolbar is now available in drawing views for both creation and rebuild.

Fast HLR/HLV in drawings does not work for Detail, Crop, Broken, or Alternate Position views or for Hide/Show Edge, Layers, Component Line Font, and Line Fonts set in Document Properties. Line Fonts and Component Line Fonts remain thin solid lines in Fast HLR/HLV mode.

9

Import and Export

This chapter describes enhancements to import and export in the following areas:

- General information
- ACIS files
- CADKEY files
- DXF/DWG files
- IGES files
- MDT files
- Parasolid files
- Pro/ENGINEER files
- STEP files
- STL files
- VRML files

General Information

Translator Add-ins

All translator add-ins are now integrated into the SolidWorks software and are always available as file types in the **Open** and **Save As** dialog boxes. These translators load and unload dynamically, as needed. They no longer require activation and no longer appear as add-ins under **Tools**, **Add-Ins**.

- Embedded ACIS® data from DXF™ files import
- Autodesk® Inventor™ import
- CATIA® HCG export
- Hoops export
- JPEG export
- Mechanical Desktop® (MDT) import (for more information, see **Autodesk Products Import Interface** on page 9-8)
- Pro/ENGINEER® import/export
- RealityWave® export
- Solid Edge® import
- Unigraphics® import

To see the new translator integration:

1. Click **Open**.

   The **Open** dialog box appears.
2. Expand the **Files of type** list.

   Note for example, that the **ProE Part (*.prt;*.prt.*;*.xpr)** and **ProE Assembly (*.asm;*.asm.*;*.xas)** are available as file formats. You did not have to activate the Pro/ENGINEER SldTrans 1.0 translator because it loaded dynamically.
3. Close the **Open** dialog box.
4. Click **New** and open an empty part document from the **Tutorial** tab.
5. Click **Save**.

   The **Save As** dialog box appears.
6. Expand the **Save as type** list.

   Note for example, that **RealityWave ZGL (*.zgl)** is available as a file format. You did not have to activate the SolidWorks ZGL translator because it loaded dynamically.
7. Close the **Save As** dialog box.

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

The **Add-Ins** dialog box appears. None of the translators listed in this section appear in the add-ins list.

9 Close the **Add-Ins** dialog box.

Import/Export Options Interface

There are two new dialog boxes, **Import Options** and **Export Options**. To access the dialog boxes from the **Open** and **Save As** dialog boxes, select the file type, then click **Options**. The selected file type is highlighted on the **File Format** tab.

Previously, there were individual import and export dialog boxes for each translator. The two new **Import Options** and **Export Options** dialog boxes replace these individual translator dialog boxes.

The **Output coordinate system** option, previously located in the **Save As** dialog box, is now located in the **Export Options** dialog box.

To see the new import and export dialog boxes:

1 Click **Open**.

2 In the **Open** dialog box, set **Files of type** to **IGES (*.igs;*.iges)**, then click **Options**.

The **Import Options** dialog box appears. **General** is highlighted on the **File Format** tab. The general options, applicable for ACIS, IGES, STEP, and VDA files, appear.

3 Close the **Import Options** and **Open** dialog boxes.

4 Click **New** and open an empty part document from the **Tutorial** tab.

5 Click **Save**.

6 In the **Save As** dialog box, select **STEP AP203 (*.step)** from the **Save as type** list, then click **Options**.

The **Export Options** dialog box appears, displaying the options that were previously located in the **STEP Export Options** dialog box. **STEP** is highlighted on the **File Format** tab.

Import Options - Units for ACIS, IGES, STEP, or VDA Files

When you import ACIS, IGES, STEP, or VDA files, you can now set the units to be either the units from the imported file or the units specified in the SolidWorks template files under **Tools**, **Options**, **System Options**, **Default Templates**. Previously, the imported files used the template-specified units; no option to use the imported file's units was available.

To see an example of the new units import option:

1. Click **Open**.
2. In the **Open** dialog box, set the **Files of type** to **IGES (*.igs;*.iges)**, then click **Options**.

   The **Import Options** dialog box appears.
3. Under **Unit**, click **File specified unit**.
4. Click **OK** to accept the other default settings.
5. Open **units.igs**. The IGES file unit of measure is feet.
6. Click **Tools**, **Options**. On the **Document Properties** tab, click **Units**.

   Under **Linear units**, **Feet** is shown because it is the unit specified in the IGES file.
7. Open **units.igs** again, but this time set the **Unit** option in the **Import Options** dialog box to **Document template specified unit** and click **OK** to accept the other default settings.
8. Verify the units used under **Units** on the **Document Properties** tab again.

   The units used are now the units specified in the SolidWorks part template (the default is **Millimeters**).

Improve Geometry

The **Improve Geometry** tool in the **Import Diagnosis** PropertyManager has been renamed to **Simplify Geometry**. The **Improve Geometry** dialog box that reported simplification results has been removed, and the results are now reported under **Geometry** in the **Import Diagnosis** PropertyManager.

When you diagnose an imported feature, if no geometry is detected for simplification, the **Simplify Geometry** button is unavailable. Previously, the **Improve Geometry** dialog box appeared and reported the results, even if no entities were simplified.

To see an example of the Simplify Geometry tool:

1. Open **simplify_geometry.sldprt**.
2. Right-click one of the **Surface-Imported** features in the FeatureManager design tree and select **Diagnosis**.

   The **Import Diagnosis** PropertyManager appears. Under **Geometry**, 6 B-surfaces need simplification.
3. Click **Simplify Geometry**.

   The number of entities requiring simplification changes to zero and the **Simplify Geometry** button becomes unavailable, indicating all entities were simplified.
4. Click **OK**.

Insert Imported Geometry

The **Insert**, **Surface**, **Import** option has been changed to **Insert**, **Features**, **Imported**. This option now supports import of solids, sketches, surfaces, curves, and graphics models (CGR, STL, or VRML files only). This option now also supports CATIA files (CGR graphics files, view only) and STL files. Previously, this option supported import of only surface features from only ACIS, IGES, Parasolid, STEP, VDAFS, and VRML files. For more information about this option, see **User Interface** on page 3-4.

To insert imported geometry:

1. Open a new part document from the **Tutorial** tab.
2. Click **Insert**, **Features**, **Imported**.

   The **Open** dialog box appears.
3. Set the **Files of type** to **STEP AP203/214 (*.step;*.stp)** and select **insert_feature.step**.
4. Click **Options**.

   The **Import Options** dialog box appears.
5. Select the **Free point/curve entities** check box, click **Import as 3D curves**, then click **OK** to accept the other default settings.
6. Click **Open**.

   In the FeatureManager design tree, note the new **ImportedCurve1** feature that contains the STEP file curve entities you inserted.

Multibody Export

When you export a multibody part document as another file type, you have the option to export selected solid bodies or all solid bodies. For more information about multibody documents, see **Multibody Parts** on page 5-2.

To export a multibody part document:

1. With a multibody part document open, select at least one solid body in the graphics area or from the **Solid Bodies** folder in the FeatureManager design tree, and click **Save**.

   The **Save As** dialog box appears.
2. Set the **Save as type** to the desired file type and click **Save**.

   The **Export** dialog box appears.
3. To export only the selected solid body, click **Selected bodies**. To export all the solid bodies, click **All bodies**.
4. Click **OK** to export the multibody part document.

ACIS Files

Curves and Wireframes

The ACIS translator now supports import and export of curves and wireframe geometry. Previously, the ACIS translator did not support these entities.

Entity Attributes Retention

You can export the entity attribute information of faces and edges to ACIS files, and this information is retained in the ACIS file. Previously, the entity attribute information was not retained during export.

If you import that same ACIS file back into SolidWorks, you can select any import options for faces and they retain the entity attribute information for faces. However, if you import edges, you must select the **B-Rep mapping** check box in the **Import Options** dialog box to retain the entity attribute information for the edges.

To see an example of entity attribute retention:

1. Open a new part document from the **Tutorial** tab.
2. Sketch a square, then create an extruded feature.
3. Right-click any face and select **Face Properties**.

   The **Entity Property** dialog box appears.
4. Under **Entity Information**, in the **Name** box, type **Test**, then click **OK**.
5. Click **Save** and save the document as an **ACIS (*.sat)** file.

   The **Export** dialog box appears.
6. Click **All bodies** then click **OK**.
7. Open the **ACIS (*.sat)** file you just saved, then right-click the same face that you did in step 3, and select **Face Properties**.

   The **Entity Property** dialog box appears. The **Entity Information**, in this case the name assigned to the face, was retained during export.

CADKEY Files

The SolidWorks software has added a CADKEY® translator. You can now import CADKEY part and assembly files into SolidWorks documents. All CADKEY files have the same **.prt** file extension. In the **Open** dialog box, set **Files of type** to **CADKEY (*.prt)** to open all CADKEY files. This translator supports all CADKEY versions up to and including version 19.

DXF/DWG Files

General Items

AutoCAD Version Support

SolidWorks supports import and export of AutoCAD files through version 2002. The SolidWorks software has updated the export option to read **R2000-2002** under **Version** in the **Export Options** dialog box to support this.

Copy and Paste from AutoCAD to SolidWorks

You can now copy and paste entities from an AutoCAD DXF or DWG file into SolidWorks part, assembly, and drawing documents. Previously you had to import the entire AutoCAD file, then delete entities you did not need or wanted to replace.

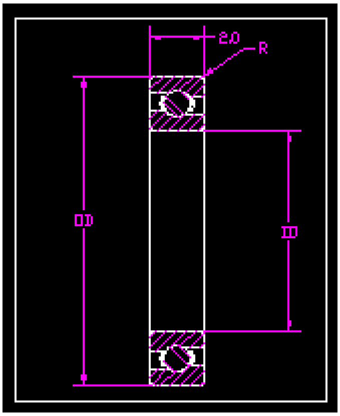
In the SolidWorks drawing document, the lines, arcs, notes, annotations, and so forth that you paste are attached to either a drawing view or the sheet, whichever is active. The pasted entities inherit their scaling, grouping, visibility, and other properties from the drawing view or sheet.

In SolidWorks part and assembly files, you must select a planar face onto which you paste the entities as a sketch.

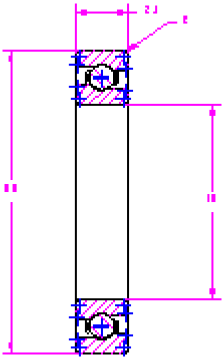
To copy and paste entities from AutoCAD to a SolidWorks drawing document:

1. In AutoCAD, open a DXF or DWG file. This example shows a DWG file. Select the entities inside the box, then click **Edit**, **Copy**.
2. In SolidWorks, open the SolidWorks drawing document into which you want to paste the entities. Click inside the sheet in the graphics area where you want to paste the entities.

3 Click **Edit**, **Paste** to paste the entities onto the active sheet.



AutoCAD file entities



AutoCAD file entities pasted into SolidWorks document

Import Items

Autodesk Products Import Interface

You can now import all supported Autodesk products (DXF, DWG, MDT, and 3D DXF files) with the **.dxf** or **.dwg** file types in the **Open** dialog box. Functionality from the SolidWorks DXF3D and SolidWorks MDT translators has been added to the **DXF (*.dxf)** and **DWG (*.dwg)** translators. For more information about translators, see **Translator Add-ins** on page 9-2.

MDT DWG files and DXF files with embedded ACIS data can now be imported through the **DXF/DWG Import Wizard**. The wizard determines automatically if a DWG file contains MDT data.

The wizard interface has been redesigned and contains view, zoom, rotate, pan, and standard view items to change the preview. You can select the **White background** check box to change the preview background color. You can also click the **Model** and **Layout** tabs below the **Preview** window to switch between model and layout views.

Previously, several different translator add-ins were required to import the various Autodesk file types.

AutoCAD Mechanical Annotations (Proxy Entities)

The SolidWorks software can now display AutoCAD Mechanical annotations (such as surface finish symbols or GTOL frames) and automatically drawn objects (such as cams and springs) when importing DXF or DWG files into SolidWorks drawing documents. SolidWorks converts these imported items to equivalent SolidWorks objects, or creates them as blocks of primitive geometry, as appropriate. Previously, SolidWorks did not recognize these proxy entities.

To see an example of the AutoCAD Mechanical annotations import:

1 Open **mechanical_proxy.dwg**.

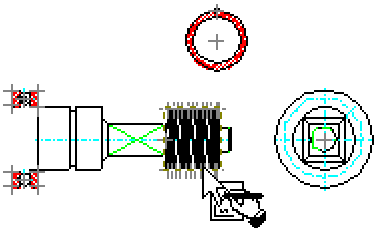
The **DXF/DWG Import Wizard** appears.

2 Make sure **Import to drawing** and **All selected layers** are selected, then click **Finish**.

In the FeatureManager design tree, the **Blocks** folder contains the imported proxy entities. The translator added the prefix **Proxy_** before each proxy entity.

3 Move the pointer over the entities in the SolidWorks drawing file.

The translator imported the proxy entities as simple geometry and blocks. Note that the pointer changes to  when moved over entities in a block.



DWG File Preview

When you import DWG files, you can now see a thumbnail image of the file in the **Preview** panel of the **Open** dialog box. Previews now appear for DWG files created by both SolidWorks and AutoCAD. In AutoCAD, the bitmap preview option must be enabled when the file is last saved. The **Open** dialog box remembers the **Preview** check box state from the last time you opened a DWG file.

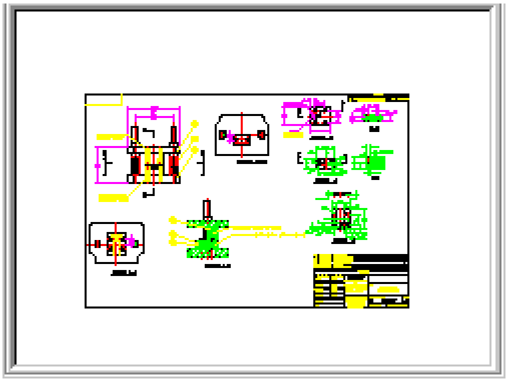
Previously, the SolidWorks software did not create preview images for SolidWorks drawing documents that you saved as DWG files, and you could not preview any DWG file.

To see an example of the DWG file preview:

1 Click **Open** .

2 In the **Open** dialog box, select **DWG (*.dwg)** as the **Files of type**, and select **attributes.dwg**.

The thumbnail image of the file appears in the **Preview** pane.



3. Click **Open**.

   The **DXF/DWG Import Wizard** appears.

4. Click **Finish** to open the ***.dwg** file.

   The file matches the preview shown in the **Open** dialog box.

5. Keep this new SolidWorks drawing document open because you use it in the next section, **Blocks**.

Blocks

The SolidWorks software now fully supports import of AutoCAD block definitions and instances with properties and attributes. The **Explode blocks** option in the **DXF/DWG Import Wizard** has been removed because it is no longer relevant. No performance gain is made by exploding block instances during import.

Previously, the **DXF/DWG Import Wizard** did not fully support the capabilities of AutoCAD blocks. For more information about blocks, see **Blocks** on page 8-14.

To see an example of the new block support:

1. Make sure the SolidWorks drawing document you opened in the previous section, **DWG File Preview**, is still open.

   SolidWorks imports each AutoCAD block insert as a block instance on the sheet. In the FeatureManager design tree, note that each AutoCAD block definition is shown with the block icon and AutoCAD block name.

2. Zoom to the revision block area in the upper right corner of the drawing, and click the text **REV**.

   The **Block Instance** PropertyManager appears. This block instance corresponds to the AutoCAD block insert. The block properties, such as the scale and rotation, appear under **Block Display** in the PropertyManager, and these now match the properties of the AutoCAD block. Previously, the properties did not necessarily match.

3. Under **Text Display**, click **Attributes**.

   The **Attributes** dialog box appears, displaying the block attributes that match the original AutoCAD block attributes.

4. Click **Cancel** and exit the SolidWorks document without saving.

Insert DXF/DWG Files

You can now insert DXF or DWG files directly into the current SolidWorks drawing or part document with the new **Insert**, **DXF/DWG** tool. The menu item activates the **DXF/DWG Import Wizard** at the appropriate dialog box, with simplified options to help you insert these files.

When you insert DXF or DWG files into SolidWorks drawing documents, the SolidWorks software inserts a new sketch on the current sheet. For SolidWorks part documents, the SolidWorks software inserts a new sketch, and the software prompts you to select a plane or face for the sketch if you have not selected one.

Previously, you had to create a new SolidWorks part or drawing file for the imported DXF or DWG file, then copy and paste the documents together.

To insert a DXF file into a SolidWorks part document:

1 Open **master_power_panel.sldprt**.

This part represents a sheet of metal onto which you insert the DXF file as a sketch.

2 Select the front face of the part and click **Normal To**.

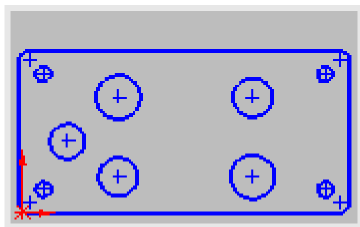
The file is inserted as a sketch onto the face or plane you select.

---

3 Click **Insert**, **DXF/DWG**.

The **Open** dialog box appears.

4 Open **master_power_panel_punched.dxf**.

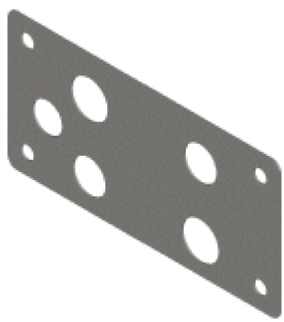


The **DXF/DWG Import Wizard** opens to the **Part Document Options** dialog box, with the appropriate options selected.

5 Click **Finish** to accept the default settings.

The **master_power_panel_punched.dxf** entities are inserted into the SolidWorks part document as a sketch on the selected face.

Now you can use the inserted sketch to cut the pattern from the part.



1 Click **Extruded Cut**.

2 Under **Direction1**, do the following:

- Set **End Condition** to **Through All**.
- Select the **Flip side to cut** check box.

3 Click **OK**.

The imported DXF sketch creates the cut on the SolidWorks part, resulting in the finished master power panel.

Non-associative Crosshatches

The SolidWorks software now supports import of non-associative crosshatches as area hatches. Previously, the SolidWorks software imported non-associative crosshatches as individual sketch lines.

To see an example of non-associative crosshatch import:

1. Open **nonassoc_crosshatch.dwg**.

   The **MDT File Import** dialog box appears.

2. Under **Import method**, click **Import as a drawing with the DXF/DWG translator**, then click **OK**.

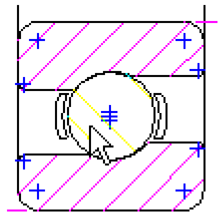
   The **DXF/DWG Import Wizard** appears.

3. Click **Finish** to accept the default settings.

4. Zoom to the crosshatch pattern at the bottom of the drawing.

5. Right-click the crosshatch in the circle shown and select **Properties**.

   The **Area Hatch/Fill** dialog box appears because the non-associative crosshatch was imported as a SolidWorks area hatch. You can change the hatch properties if desired.



XREF Support

SolidWorks now supports import of XREFs in AutoCAD DWG files.

- If an imported block is an XREF, the symbol **->** appears next to the block name in the FeatureManager design tree.
- If the XREF has a dangling definition, the symbol **->?** appears.

Previously, the **DXF/DWG Import Wizard** did not support XREFs.

Export Items

Crosshatch Export

SolidWorks crosshatch patterns are now translated into AutoCAD hatch patterns when you save SolidWorks documents as DWG or DXF files. The SolidWorks software translates the SolidWorks crosshatch patterns as non-associative hatch definitions, and preserves the layer and color of the original crosshatch. SolidWorks also supports crosshatch export when you map layers using a mapping file. Previously, SolidWorks hatch patterns became individual line segments when exporting drawings as DWG or DXF files.

Layer Map Export Option

When you export SolidWorks drawing documents as DXF or DWG files, you now have the option to map only those items whose layers are not otherwise defined. Previously, the mapping file settings took priority over all previous settings.

All entity types that currently can be assigned to AutoCAD layers through the mapping file now support layering in the SolidWorks drawing format.

To see an example of the layer map export option:

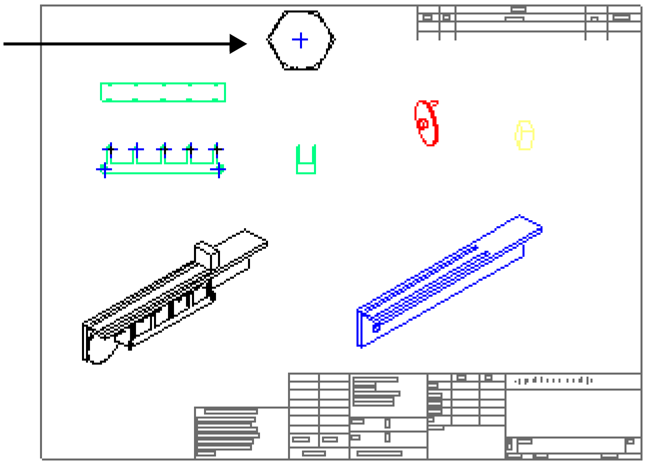
1 Open **layer_map.slddrw**.

2 Open the list on the Layer toolbar.

Note the six layers used in the drawing document.

3 Close the Layer toolbar.

4 Select a side of the hexagon-shaped object.



The **Line** PropertyManager appears. Under **Options**, note that this entity is not assigned to a layer, as shown by **Layer** **-None-**.

5 Click **OK** to close the PropertyManager.

Now you create the custom map file.

1 Click **File**, **Save As**.

2 In the **Save As** dialog box, select **Dwg (*.dwg)** from the **Save as type** list, then click **Options**.

The **Export Options** dialog box appears.

3 Select the **Custom Map SolidWorks to DXF** check box, then click **OK**.

4 In the **Save As** dialog box, click **Save**.

The **SolidWorks to DXF/DWG Mapping** dialog box appears. Note the default layers **0** and **DEFPOINTS**.

5 Select the **Keep existing SolidWorks drawing layers for entities** check box.

By selecting this option, the mapping file settings apply only to those entities whose layers are not defined. All existing layers in the SolidWorks drawing file are preserved in the exported file.

> If you do not select this option, the mapping file definitions overwrite all of the current SolidWorks drawing file layers.

6 Click **Add**.

7 In the **Add New Layer Definition** dialog box, do the following:

- Name the layer **TEST**
- Select **Color 1** (red)
- Set the line style to **Hidden Lines / Thin line**

8 Click **OK**.

9 Select the **Map Entities** tab, then click **Add**.

The **Add New Entity Mapping** dialog box appears.

10 To map the hexagon sketch, which consists of sketch lines, to the new layer **TEST**, do the following:

- Under **Entity** select **Sketch Lines**.
- Under **Layer** select **Test**.

11 Click **OK** twice to save the file.

To verify that the layers mapped correctly:

1 Open the file you just saved, **layer_map.dwg**.

The **DXF/DWG Import Wizard** appears.

2 Make sure **Import to drawing** is selected, then click **Finish**.

3 Click **Layer Properties** on the Layer toolbar to display the layers.

All existing layers originally in the SolidWorks drawing document were preserved when you exported the document as a DWG file. The new layer, **TEST**, was added. Previously, only the layers from the mapping file appeared in the exported file because the mapping file overwrote all existing layers in the SolidWorks drawing document.

4 Double-click the **On/Off** icon for the **TEST** layer.

The hexagon-shaped object, along with the sheet border and title block entities, appear and disappear from the graphics area because they were added to the **TEST** layer during the layer map export. Previously, these entities would have been mapped to layer **0** during export.

IGES Files

BREP Data Export

You can now export BREP data from solids and surfaces in SolidWorks part and assembly documents to IGES files. Previously, you could only import BREP data.

To export BREP data to IGES files:

1 Open a SolidWorks part or assembly document that you want to export.

2 Click **File**, **Save As**.

3 In the **Save As** dialog box, select **IGES (*.igs)** from the **Save as type** list, then click **Options**.

   The **Export Options** dialog box appears.

4 Under **Solid/Surface features**, select the **IGES solid/surface entities** check box. Select **Manifold Solid B-rep Object (type 186)** from the menu.

5 Click **OK**, then click **Save** in the **Save As** dialog box to export the document using BREP data.

Error Report File

The IGES error file (**.err**) has been merged into the IGES report file (**.rpt**). The IGES report file now contains error information, in addition to process and file information.

Imported Curve Colors

The IGES translator now supports color when you import curves. Previously, the IGES translator did not support curve color import.

Import Surface Options

The IGES **Surface options** section has been removed from the new **Import Options** dialog box because these options are obsolete.

MDT Files

You must have the Mechanical Desktop (MDT) software installed, but not necessarily running, on your computer to complete the procedures described in this section.

The SolidWorks MDT translator uses the MDT application during the file conversion process. If the SolidWorks MDT translator appears to stop during the conversion process, check the MDT application, which may have stopped because it requires user input or intervention. For example, the MDT application may be unable to find a referenced file and opens a dialog box that requires you to select the referenced file.

Assembly Mates

The MDT translator now preserves point-to-point and line-to-line mates when you import MDT assembly files into the SolidWorks application. Previously, the MDT translator did not preserve these mates.

To see an example of the MDT assembly mates support:

1 Open **roller_asm.dwg**.

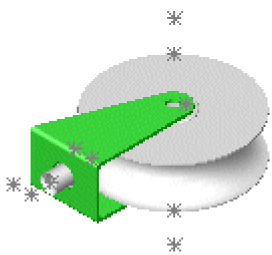
The **MDT File Import** dialog box appears.

2 Make sure **Import as a part with MDT translator** is selected, then click **OK**.

The **DXF/DWG Import Wizard** appears.

3 Click **Finish** to accept the default settings.

The **Mechanical Desktop to SolidWorks Converter** progress box reports on the conversion process.



4 In the FeatureManager design tree, expand **Mates** and note the two mates.

5 Right-click the **Distance1** mate, and select **Edit Definition**.

The **Distance1** PropertyManager appears.

Under **Mate Settings**, in the **Entities to Mate** box, note that two points are listed. The MDT translator preserved this point-to-point mate as a **Distance** mate when you imported the MDT assembly file.

Combined Features

The MDT translator now imports MDT combined features, which are sometimes referred to as “toolbodies.” Previously, the MDT translator did not support combined features.

To see an example of the combined features support:

1 Open **pivot_combined.dwg**.

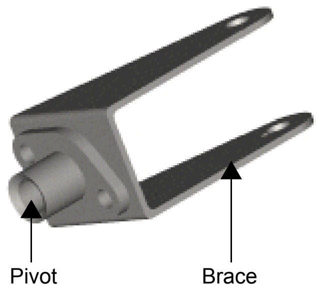
The **MDT File Import** dialog box appears.

2 Make sure **Import as a part with MDT translator** is selected, then click **OK**.

The **DXF/DWG Import Wizard** appears.

3 Click **Finish** to accept the default settings.

The **Mechanical Desktop to SolidWorks Converter** progress box reports on the conversion process.



The complete combined feature appears in the graphics area. Two MDT features, the pivot and the brace, were successfully imported. In earlier versions, the pivot feature was not imported and multiple rebuild error messages appeared in the FeatureManager design tree.

Cosmetic Thread Import for Tapped Holes

The MDT translator now recognizes MDT tapped hole features when you import MDT files. The translator creates an equivalent cosmetic thread annotation in the SolidWorks document. Previously, the translator imported tapped holes as simple holes without cosmetic thread annotation.

To see an example of cosmetic thread import for tapped holes:

1 Open **frame.dwg**.

The **MDT File Import** dialog box appears.

2 Make sure **Import as a part with MDT translator** is selected, then click **OK**.

The **DXF/DWG Import Wizard** appears.

3 Click **Finish** to accept the default settings.

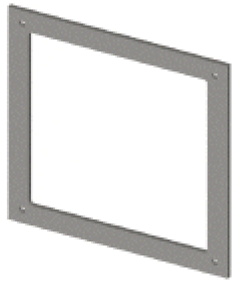
The **Mechanical Desktop to SolidWorks Converter** progress box reports on the conversion process.

Note that the model imports as a SolidWorks assembly document that contains four holes.

4 In the FeatureManager design tree, expand **Part1**, then **Hole1**.

Note the **Cosmetic Thread** icon, which indicates that the hole contains a cosmetic thread annotation.

5 Keep the imported files open because you use the part document in the next section, **Design Tables**.



Design Tables

The MDT translator now imports MDT design tables (Design Variable Tables - Global Variable Sheets) into SolidWorks documents. Previously, the MDT translator did not support design table import.

To see an example of design table import:

1 Click **Window**, **Part1** to open the part document that you created in the previous section, **Cosmetic Thread Import for Tapped Holes**.

2 In the FeatureManager design tree, right-click **Design Table** and click **Edit Table**.

The design table opens. You can edit this design table, as necessary, to change the model geometry.

3 Click outside the design table within the graphics area to close the design table.

Large Assemblies

The MDT translator can now import larger MDT assembly files than was previously possible, in excess of 130MB in size, depending on the complexity of the data. A new dialog box reports that the MDT assembly tree is being built.

Work Features

The MDT translator now imports MDT Work Features (Work Planes, Work Axes, and Work Points) into equivalent SolidWorks reference geometry. Previously, the MDT translator did not support Work Feature import.

To see an example of MDT Work Feature import:

1 Open **pivot.dwg**.

The **MDT File Import** dialog box appears.

2 Make sure **Import as a part with MDT translator** is selected, then click **OK**.

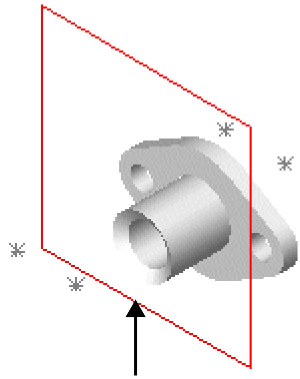
The **DXF/DWG Import Wizard** appears.

3 Click **Finish** to accept the default settings.

The **Mechanical Desktop to SolidWorks Converter** progress box reports on the conversion process.

4 In the FeatureManager design tree, expand the **Pivot** component and move the pointer over **WorkPlane1**.

The imported MDT Work Plane appears in the graphics area and was imported as a SolidWorks plane. Note that the MDT Work Axes were also imported. The SolidWorks software creates Axis Points when it trims MDT Work Axes. Axis Points are SolidWorks points that mark the trimmed ends of the imported axes.



Parasolid Files

Curve and Wireframes

The Parasolid translator now supports import and export of curves and wireframes. Previously, the Parasolid translator did not support these entities.

Pro/ENGINEER Files

The Pro/ENGINEER translator now supports import of free curves, wireframes, and surface data. Previously, the Pro/ENGINEER translator did not support import of these entities.

STEP Files

Configuration Data Import

You now have the option to import STEP configuration data. Previously, the STEP translator always imported STEP configuration data - there was no option. To import STEP configuration data, select the **Map configuration data** check box in the **Import Options** dialog box.

To see an example of STEP configuration data import:

1 Click **Open** .

2 In the **Open** dialog box, set **Files of type** to **STEP AP203/214 (*.step;*.stp)**, then click **Options**.

The **Import Options** dialog box appears.

3 Under **STEP**, select the **Map configuration data** check box.

This option imports all STEP configuration data.

4 Click **OK** to accept the other default settings.

5 Browse to **STEPconfig.step** and click **Open**.

6 Click **File**, **Properties**, and select the **Custom** tab.

7 Scroll through the **Properties** list to view the imported STEP configuration data.

Curve Color

For STEP AP214 files, the STEP translator now supports import and export of color in curves. Previously, the STEP translator did not support curve color import or export.

STL Files

The STL translator now supports import of STL files into SolidWorks documents. In the **Import Options** dialog box, you have the option to import STL files as graphical data, solids, or surfaces. When you import STL files as graphical data, you can select the **Import texture information** check box to import texture information if this data exists. Previously, the SolidWorks software supported only export as STL files.

You can now assign a unit of measure to the model for both import and export.

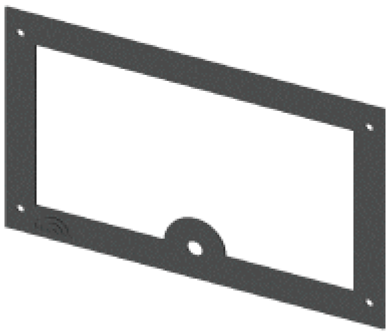
To see an example of STL file import:

1 Click **Open**.

2 In the **Open** dialog box, set **Files of type** to **STL (*.stl)**, then click **Options**.

The **Import Options** dialog box appears. Under **Import as**, there are three options:

- **Graphics Body**. Import data as graphical data
- **Solid Body**. Import data as solids if applicable
- **Surface Body**. Import data as surfaces



3 Click **Graphics Body**, then click **OK**.

4 Browse to **gasket.stl** and click **Open**.

Note the icon in the FeatureManager design tree that indicates the file contains STL graphical data.

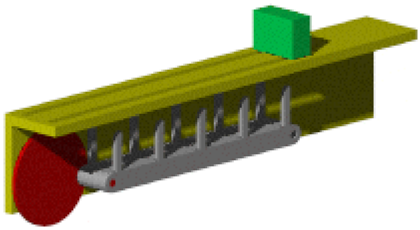
VRML Files

The VRML translator now supports import and export of standard version 2.0 VRML (VRML 97) files. You have the option to import VRML files as graphical data, solids, or surfaces. When you import VRML files as graphical data, you can select the **Import texture information** check box to import texture information if this data exists. The shading and color viewing support is enhanced. Previously, the VRML translator supported only version 1.0 VRML files, and you did not have any import options.

When you export to VRML files, you can choose the version to export. Color support has been improved. Previously, no version option existed because it was not required.

You can now assign a unit of measure to the model for both import and export.

To see an example of the new VRML export functionality:



1 Open **conveyor.sldasm**.
2 Click **File**, **Save As**.
3 In the **Save As** dialog box, select **VRML (*.wrl)** from the **Save as type** list, then click **Options**.

   The **Export Options** dialog box appears.
4 Under **Version**, select **VRML 97** from the list. This exports the document as a version 2.0 VRML file.
5 Click **OK** to accept the other default settings, then click **Save** in the **Save As** dialog box to save the file.

10

Sheet Metal

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

- Individual bend control
- Conic bends
- Lofted bends
- Edge flanges
- Miter flanges
- Flat patterns
- Bend deduction
- Bend tables

Individual Bend Control

You can now set the bend allowance values in each sheet metal feature and in individual bends within a sheet metal feature. This is helpful for parts with brake bends and progressive die bends in the same model.

For example, if you have a sketched bend, you can set the bend allowance value for both the **Sketched Bend** feature and for the **SketchBend** within the feature.

Conic Bends

You can now use the **Fold** and **Unfold** tools with conical sheet metal bends. Previously, you had to suppress the **Process-Bends** feature to unfold a conical sheet metal bend.

To unfold a conical sheet metal bend:

1 Open **conical.sldprt**.

2 Click **Unfold** on the Sheet Metal toolbar, or click **Insert**, **Sheet Metal**, **Unfold**.

The **Unfold** PropertyManager appears.

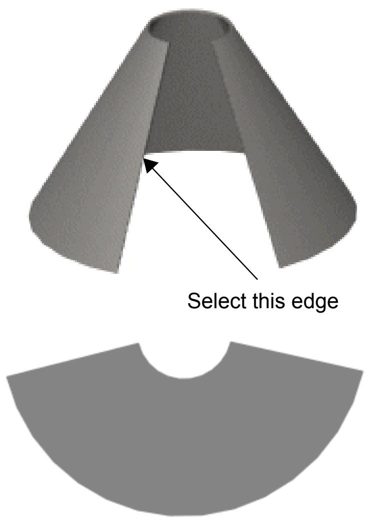
3 In the graphics area, select a linear edge as the **Fixed face** .

4 In the PropertyManager, click **Collect All Bends**.

RoundBend1 appears in the **Bends to unfold** box.

5 Click **OK** .

The conical bend unfolds.



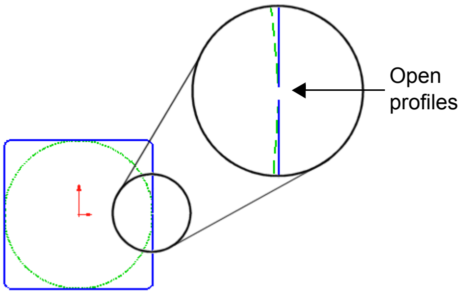
Lofted Bends

Lofted Bend Feature

You can now create a lofted bend in sheet metal parts. A lofted bend is similar to a thin feature loft. It starts with two open profile sketches that are connected by transitions between the profiles. The **Base-Flange** feature is not used with the **Lofted Bend** feature.

When you create the sketches for a lofted bend, *both* sketches must be open profiles, and they cannot have sharp edges. You can use the **Sketch Fillet** tool to round off sharp edges.

Both profile openings should also be aligned for flat-pattern accuracy.



To create a lofted bend:

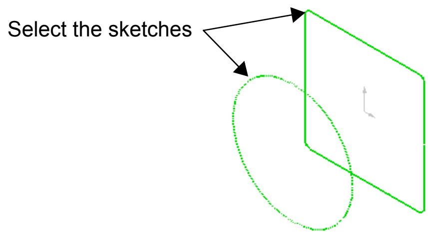
1 Open **lofted_bend.sldprt**.

Notice that the circular sketch is offset from the rectangular sketch.

2 Click **Lofted Bend** on the Sheet Metal toolbar, or click **Insert**, **Sheet Metal**, **Lofted Bends**.

The **Lofted Bends** PropertyManager appears.

3 In the graphics area, select both sketches in the areas as shown. Make sure to select the points from which you want the path of the loft to travel.



Sketch1 and **Sketch2** appear under **Profiles** in the PropertyManager.

4 Set **Thickness** to 1mm.

5 Click **OK**.

The lofted bend is complete.

6 Keep **lofted_bend.sldprt** open for the next procedure.

Bend Deviation

In general, lofted bends create deformations in the flat pattern. You can measure these deformations in the **Bend Deviation** PropertyManager. The **Bend Deviation** PropertyManager displays the surface area and curve lengths of the lofted bend.

To examine bend deviation:

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

2 In the FeatureManager design tree, do the following:

a) Right-click **Flat-Pattern1** and select **Unsuppress**.

b) Click ⊞ to expand **Flat-Pattern1**.

c) Right-click **Flatten-<Freeform Bend1>1**, and select **Bend Deviation**.

The **Bend Deviation** PropertyManager appears, and displays the following:

Under **Bend Surface Area**:

- **Folded**. Surface area of the lofted bend when in the folded state.
- **Flat**. Surface area of the lofted bend when in the flattened state.
- **Deviation**. **Flat** value minus the **Folded** value.
- **Percentage change (%)**. **Deviation** value divided by the **Folded** value, multiplied by 100.

Under **Curve Lengths**:

- **Max deviation only**. Select this check box to show only the maximum deviation of the curve.

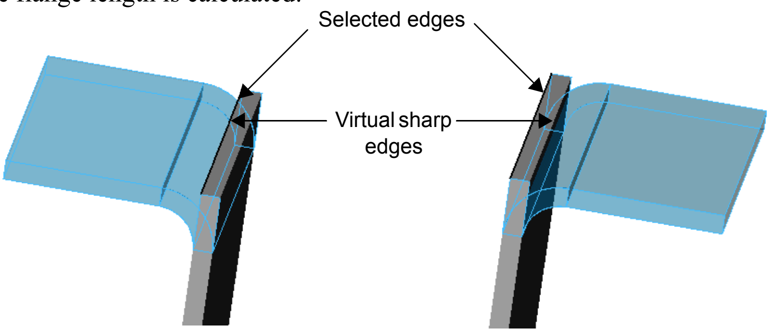
In the graphics area, the corresponding values are displayed for the bend deviation of each edge.

3 Click **OK** ✔.

Edge Flanges

When you add an edge flange to a model, you can now set the flange length from the virtual sharp.

The flange length is measured from the virtual sharp edge; the following images illustrate how the flange length is calculated:



To set an edge flange length from the virtual sharp:

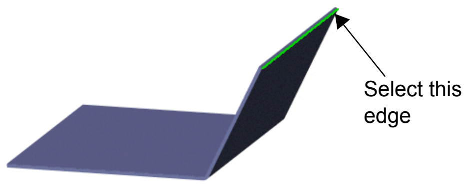
1 Open **edge_flange.sldprt**.

2 Click **Edge Flange** on the Sheet Metal toolbar, or click **Insert**, **Sheet Metal**, **Edge Flange**.

The **Edge-Flange** PropertyManager appears.

3 In the graphics area, select the edge as shown.

Edge <1> appears in the **Edge** box.



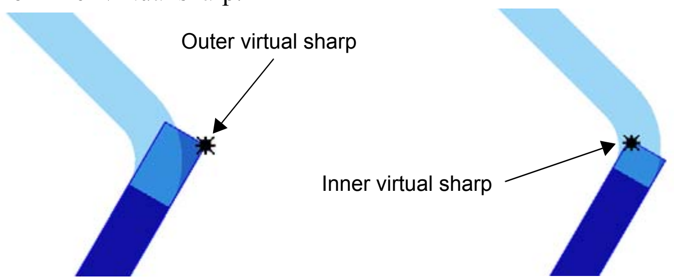
4 In the PropertyManager, do the following:

a) Set **Flange Angle** to 75°.

b) Under **Flange Length**, set **Length** to 50mm, and click **Outer Virtual Sharp**.

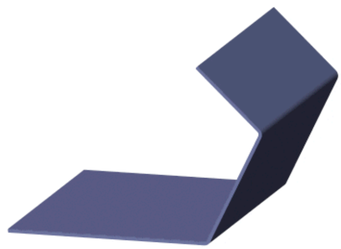
c) Under **Flange Position**, click **Bend from Virtual Sharp**.

The **Flange Length** option determines whether the bend is on the outer or inner virtual sharp.



5 Click **OK**.

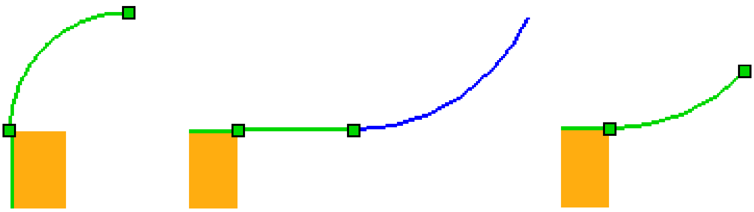
The edge flange is 50mm in length from the virtual sharp.



Miter Flanges

You can now create miter flanges with an arc. Previously, only line segments could be used to make miter flanges.

The sketched arc cannot be tangent to the thickness edge. The arc can be tangent to the long edges, or by putting a small sketch line between the arc and the thickness edge.



Valid sketch: Arc is tangent to long edge

Valid sketch: Line is coincident to thickness edge, and arc is tangent to line

Invalid sketch: Arc is tangent to thickness edge

To create a miter flange with an arc:

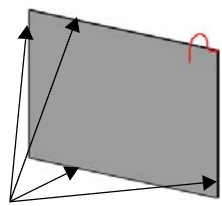
1 Open **miter_arc.sldprt**.

The part contains a rectangular base-flange and a sketch that is normal to an edge. The sketch was made with the **Line** and **Tangent Arc** tools.

2 In the FeatureManager design tree, click **Sketch2**.

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

The **Miter Flange** PropertyManager appears.



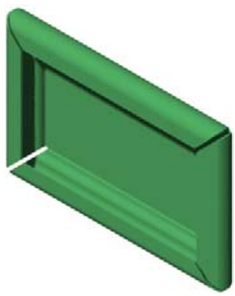
4 In the graphics area, select the edges as shown.

The edges appear in the **Along Edges** box in the PropertyManager.

5 Set **Gap distance** to 2.5mm.

6 Click **OK**.

The mitered edges appear on the model.



Sharp corners are allowed between arcs and adjacent edges. Appropriate bends are added to the sharp intersections.

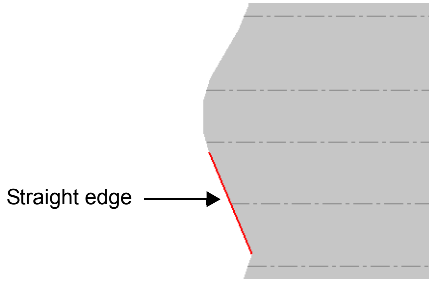
Flat Patterns

When a sheet metal part is flattened, and if there is only one edge on each side of a bend, you can use **Simplify bends** to straighten curved edges in the flat pattern. In previous SolidWorks releases, edges were automatically simplified; now you can turn the option off to keep complex edges.

To turn off the simplify bends option:

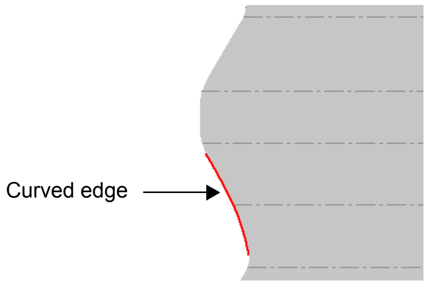
1. Open **simplify_bends.sldprt**.

   The part is flattened, with **Simplify bends** on.



2. Drag the pointer over the sheet metal edges, and notice how they are straight.
3. In the FeatureManager design tree, right-click **Flat-Pattern1** and select **Edit Definition**.

   The **Flat-Pattern** PropertyManager appears.
4. Under **Parameters**, clear the **Simplify bends** check box.
5. Click **OK**.
6. Drag the pointer over the sheet metal edges, and notice how they are curved.



Corner-Trim

When the flat pattern of a sheet metal model is unsuppressed, you can apply a corner trim to corner edges. The **Corner-Trim** feature adds reliefs to inner corners and breaks outer corners. In the FeatureManager design tree, the **Corner-Trim** feature appears after the **Flat-Pattern** feature.

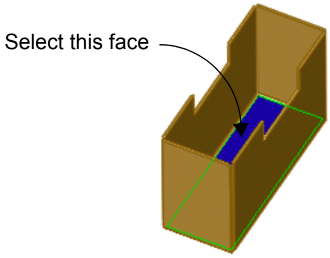
To apply a corner trim to sheet metal edges:

1 Open **flat_pattern.sldprt**.

2 In the FeatureManager design tree, right-click **Flat-Pattern1** and select **Edit Definition**.

The **Flat-Pattern** PropertyManager appears.

3 In the graphics area, select the bottom inside face as the **Fixed Face** .



4 Under **Corner Options**, do the following:

a) Select the **Add Corner-Trim** check box.

b) Select **Circular** from the **Relief type** list.

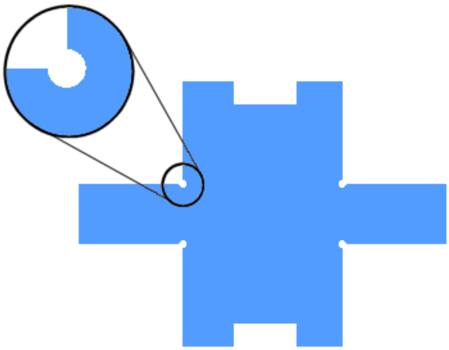
c) Select the **Ratio to thickness** check box.

d) Set the **Ratio of radius/distance to sheet metal thickness** to 2.

5 Click **OK** .

The corner trim is applied.

6 In the FeatureManager design tree, right-click **Flat-Pattern1** and select **Unsuppress** to see the corner trim feature.



Bend Deduction

When you create a sheet metal feature, you can now set the bend deduction value directly in the PropertyManager. In previous SolidWorks releases, bend deduction values could be set through a bend table only.

Bend Tables

Edit

You can edit a bend table in a separate Microsoft Excel window. To edit a bend table in a separate window, click **Edit**, **Bend Table**, **Edit Table in New Window**.

Microsoft Excel and Text Formats

You can now use Microsoft Excel or text formatting for all bend table types.

K-Factor

When you select k-factor as your bend allowance method, you can specify a k-factor table. In previous releases, k-factor was a single numeric entry.

SolidWorks 2003 comes with a k-factor table in Microsoft Excel format. A template of this table is located in *installation directory***lang****English****Sheetmetal Bend Tables****kfactor base bend table.xls**.

Multiple Bend Angles

Bend tables in Microsoft Excel format now support multiple angles in a single bend table file. Previously, these tables allowed for only one angle; other angles were interpolated.

Units

Bend tables in the text format now support millimeters, centimeters, inches, and feet. In previous SolidWorks releases, the bend tables were in meters only.

You set the unit of measure in the units row at the top of the bend table.

11

SolidWorks Office Add-Ins

This chapter describes enhancements to the following SolidWorks Office add-ins:

- SolidWorks Office toolbar
- eDrawings
- eDrawings Professional
- FeatureWorks
- SolidWorks Animator
- SolidWorks Toolbox
- SolidWorks Utilities

SolidWorks Office Toolbar

If you installed SolidWorks 2003 with a registration code for SolidWorks Office, you can display the SolidWorks Office toolbar. This toolbar allows you to activate any add-in applications included in the SolidWorks Office package, such as FeatureWorks® and SolidWorks Animator.

To display the SolidWorks Office toolbar:

1. In an open SolidWorks document, click **Tools**, **Customize**.

   The **Customize** dialog box appears.
2. On the **Toolbars** tab, select the **SolidWorks Office** check box.

   The **SolidWorks Office** check box is unavailable if you did not install SolidWorks 2003 with a SolidWorks Office registration code.
3. Click **OK**.

   The SolidWorks Office toolbar appears.

eDrawings

Context-sensitive Tabs

Context-sensitive vertical tabs appear in the eDrawings Manager for the following:

- **Analysis** (**.eprt** and **.easm** files with COSMOS/Works analysis data only)
- **Components** (assemblies only)
- **Configurations** (eDrawings Professional or Review-enabled parts and assemblies only)
- **Cross section** (eDrawings Professional or Review-enabled parts and assemblies only)
- **Markup** (eDrawings Professional or Review-enabled documents only)
- **Measure** (eDrawings Professional or Review-enabled documents only)
- **Sheets** (drawings only)

The **Drawing Views** pane has been replaced by the **Sheets** tab, and the **Review** pane has been replaced by the **Markup** tab.

Modes

Markup and Animate are no longer modes. The **Markup** and **Animate** toolbar buttons that activated these toolbars have been removed. The Animate toolbar is always available, and the Markup toolbar is always available if you have eDrawings Professional, or if you open a review-enabled document.

Quick Help

Context-sensitive Quick Help boxes appear in the eDrawings Viewer for on-screen help with tasks and tools. Quick Help is available for the following:

- Animate tools
- **Cross section** tool (drag in graphics area, and options)
- Markup tools
- **Measure** tool selection filters
- **Move Component** tool
- Reply to comments
- **Rotate** tool during Animation (drawings only)

Quick Help is smart help. If you perform a function that Quick Help is describing, it assumes you have learned how to do that function, and disappears. Quick Help does not reappear if you perform that same function later. You can turn Quick Help on or off with the **Turn on Quick Help** menu item in the **Help** menu. If you turn Quick Help off and then turn it back on, all Quick Help boxes are reactivated.

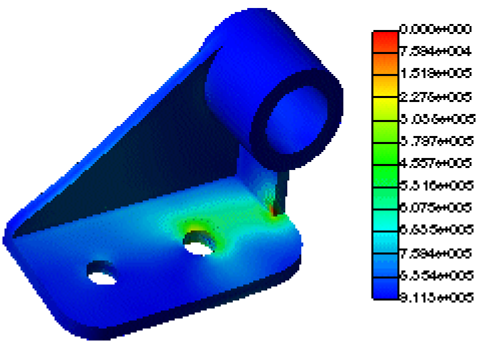
DXF/DWG Files

You can open DXF and DWG files in the eDrawings Viewer. Click **File**, **Open** and in the **Files of type** box select **DXF Files (*.dxf)** or **DWG Files (*.dwg)** to open these file types. The eDrawings Viewer supports the following:

- DXF and DWG files version 2.5 and above.
- AutoCAD fonts (SHX), if AutoCAD is installed. Otherwise the text is rendered in a default font.
- TrueType fonts, outlines only (not filled).

Analysis Data

You can display COSMOS/Works analysis data, when available, in eDrawings part or assembly files. You can display an analysis mesh, legend, and title for the analysis data in the model. In the **Tools**, **Options** dialog box, the **Analysis** tab contains options for automatically opening analysis files and saving analysis information



Mass Properties

You can display Mass Properties for part and assembly documents saved in SolidWorks 2003. In the eDrawings Viewer, click **Tools**, **Mass Properties** to display the **Mass Properties** dialog box containing density, mass, volume, and surface area information. You can set the units and precision in the **Mass Properties** dialog box under **Length unit** and **Decimal places**.

SpaceBall and SpaceMouse Devices

The eDrawings Viewer supports the SpaceBall® and SpaceMouse® space devices, using the 3DxWare device driver, version 2.0 and above, available from www.3Dconnexion.com. You can use these space devices to manipulate the model in the eDrawings Viewer as if you held the model in your hand. The numbered buttons on the space devices enable tools in the eDrawings Viewer. You can customize the button mapping to enable the available tools listed in the space device dialog box.

Show All Hidden Components

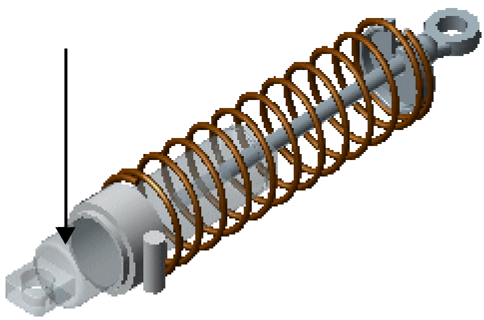
In assembly files, you can show all hidden components. Select a component in the graphics area or on the **Components** tab, right-click and select **Show All** to show all hidden components.

Hide Others

In assembly files, you can keep a single component visible, and hide all other components. Select a single component in the graphics area or on the **Components** tab, right-click and select **Hide Others** to hide all other components except the selected component.

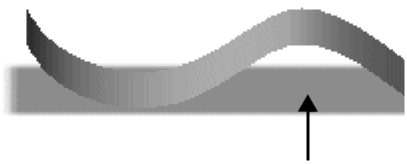
Transparency

In assembly files, you can make components transparent. Select components in the graphics area or on the **Components** tab, right-click and select **Make Transparent** to make the components transparent. Select **Make Solid** to make transparent components solid.



Shadows

You can display shadows for models with the **View**, **Shadows** option. Shadows are enabled by default.



Status Bar Icons

Icons in the status bar indicate the review-enabled and measure status of documents, as follows:

- Review-enabled. Review-enabled eDrawing files can be marked up by anyone with an eDrawings Viewer.
- Not review-enabled
- Measure file entities is enabled
- Measure file entities is not enabled

Toolbar Buttons

The following context-sensitive toolbar buttons are now in the **eDrawings Manager**:

- **Sheets** tab - **3D Pointer** tool, **Overview Window** tool, **Create Layout** tool
- **Components** tab - **Explode/Collapse** tool

Professional Menu

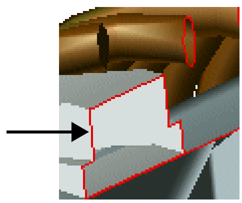
The **Professional** menu has been removed and its menu items have been moved to the **Tools** menu.

Virtual Fold

The **Virtual Fold** tool has been renamed as the **Create Layout** tool.

Cross Section Edge Color

When you view cross sections, the edges that touch the cross section plane are shown in red.



Tools

Tools are context-sensitive. The tools that appear depend on the document type that is active. For example, the drawing tools **Overview Window**, **3D Pointer**, and **Create Layout** appear only when you open drawing documents.

Activate Drawing Sheets

For drawings, you can double-click sheets on the **Sheets** tab to activate them.

Display Drawing Views

For drawings, you can double-click a view on the **Sheets** tab or graphics area to zoom to that view.

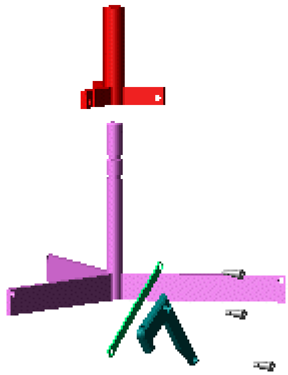
eDrawings Professional

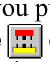
This section describes new features and enhancements that are available only in the eDrawings Professional version.

Multiple Configurations

You can save multiple configurations when you publish parts and assemblies from SolidWorks 2003. In the **Configurations to Save to eDrawing** dialog box, you can save the current configuration, all configurations, or selected configurations. In eDrawings Professional, images of the configuration appear on the **Configuration** tab, and tabs with the configuration names appear under the graphics area. Click the images or tabs to switch configurations.

Exploded Views



Exploded views in the SolidWorks assembly document are exported automatically when you publish from SolidWorks 2003. Click **Explode/Collapse**  on the **Components** tab to explode or collapse the model.

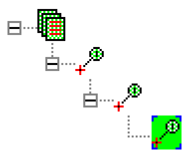
Drawings Sheets

When a SolidWorks 2003 drawing document contains multiple sheets, you can select the drawings sheets to publish. In the **Sheets to Save to eDrawing** dialog box, you can choose to save the current sheet, all sheets, or selected sheets.

Markup Enhancements

The following enhancements have been made to Markup:

Threaded Discussion.



Markup comments are displayed as a threaded discussion. When you reply to a comment, the discussion thread appears on the **Markup** tab.

Dimensions

You can create dimensions as markup notes contained in comments. Click **Dimension**  on the Markup toolbar to add dimensions to comments.

Long Descriptions

In the **Description** box on the **Markup** tab, you can type optional text that is contained in comments. This optional text can be part of a comment or the only note in a comment, and it does not appear in the graphics area.

Comments Font and Color

You can specify a default font and color for comments on the **Markup** tab under **Tools**, **Options**. You can also click **Options** on the **Markup** tab to display the **Options** dialog box. You can change font properties of your own comments on a comment-by-comment basis.

Bold Comments

Unread comments are displayed in bold in the threaded discussion on the **Markup** tab.

New Comments

New comments are created automatically if the state of the model changes from one comment to the next. "State of the model" includes visibility (hidden/shown components, cross sections, transparencies), orientation, zoom level, and pan position.

Editing Comments

You can no longer edit comments made by other users.

Model State

Markup comments remember the model state at time of creation.

Snap Lines

When you use the **Line** tool on the Markup toolbar, horizontal and vertical snaps are now available.

Wrap Text

To automatically wrap text in the text box, select the **Wrap text** check box in the text box. This setting is enabled by default.

FeatureWorks

Hem Flange Features

FeatureWorks now supports interactive feature recognition of hem flange features on sheet metal parts. FeatureWorks can recognize open or closed type hem flanges.

Open hem flange

Closed hem

Hole Features

FeatureWorks now recognizes holes as independent features in automatic feature recognition. You now select just the **Holes** check box under **Automatic Features** to recognize hole features. Previously, you had to select both the **Basic features** and **Holes** check boxes to recognize holes, which increased processing time. FeatureWorks recognizes hole features on planar faces only. Additionally, interactive hole feature recognition has been enhanced.

Multibody Models

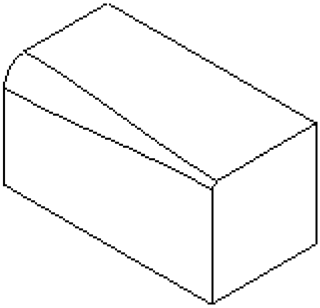
FeatureWorks supports multibody part documents. When you recognize a part containing multibody parts, the **FeatureWorks** dialog box appears, and you can recognize one imported body at a time. For more information on multibody parts, see **Multibody Parts** on page 5-2.

Sweep Features

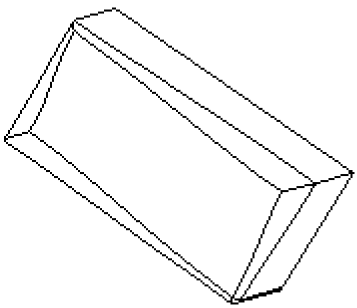
FeatureWorks now supports interactive recognition of base sweep features.

Variable Radius Fillet Features

FeatureWorks now supports automatic feature recognition of variable radius fillet features. Support includes simple variable radius fillets, chained variable radius fillets, and chains of simple and variable radius fillets.



Simple variable radius fillet



Chained variable radius fillets

SolidWorks Animator

Animator includes new scheduling functions. To use any of these functions, you must have at least one existing path, that you can select from the Animator Tree in the Feature Manager design tree. New functions include:

- **Reverse Path**. Change the direction of an existing path.
- **Copy and Move Paths**. Duplicate a path on a part or assembly component, then change the start time or duration of a path.
- **Mirror Animation**. Duplicate the existing animation, reverse it, and append the mirror copy to the original animation.
- **Reverse Animation**. Reverse an existing animation.

The functions are described in detail in the following sections.

Reverse Path

You can take an existing path and reverse the animation.

To reverse path:

1. Select one or more paths from the Animator Tree in the FeatureManager design tree.
2. Click **Animator, Schedule, Reverse Path**.

   The selected path now plays in the opposite direction. The duration and start times do not change.

Copy and Move Path

You can duplicate paths, as well change the start time and the duration of the paths.

Copy Paths and **Move Paths** use the same interface. If a part or assembly component includes multiple paths, you can copy or move more than one path simultaneously. However, to prevent paths from overlapping, it is more efficient to select each path individually.

Using the **Copy and Move Paths**, you can make the following changes:

- Keep the current start time.
- Move the start time earlier or later.
- Specify no movement.
- Change the scale factor to specify a different length for the copied path.
- Reverse the direction of each path to play the copied path in the reverse direction of the original.
- Create a copy of each path to create new paths.

To copy or move paths:

1 Select one or more paths from the Animator Tree in the FeatureManager design tree.

2 Click **Animator, Schedule, Copy Paths**.

- or -

Click **Animator, Schedule, Move Paths.**

3 If necessary, under **Move paths**, select a method to move the start time. The default is to play back the copied path immediately after the existing path.

Note: If you select **Don't move at all**, the new path is copied in place. The new path collides with the existing path, causing unpredictable results during playback.

4 Under **Scale paths**, select a **scale factor** to change the length of the copied path.

For example, .5 cuts the length of the copied path in half, whereas 2 doubles the length.

5 Select **Reverse direction of each path** to play back the copied path in the opposite direction from the original. The default is to play back the copied path in the same direction as the original.

6 Select **Create a copy of each new path** to create a new path. The default is to create a new copy when you select **Copy Paths**, and *not* to create a copy when you select **Move Paths**.

7 Click **OK**.

Mirror Animation

You can create a mirror image of every path in an animation.

To mirror animation:

Click **Animator, Schedule, Mirror Animation**.

Each path in the animation is copied, reversed, and appended to the original animation.

Reverse Animation

Unlike **Reverse Path**, which affects one path at a time, **Reverse Animation** affects all paths in the animation.

To reverse animation:

Click **Animator, Schedule, Reverse Animation**.

Each path in the animation is reversed and automatically moved to a new start position. The new animation plays the entire sequence in reverse.

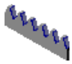
SolidWorks Toolbox

Toolbox Browser

The icon for the Toolbox Browser tab  is new.

Gears

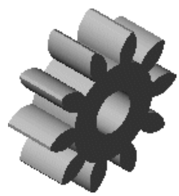
There are new gears available for the ANSI, DIN, ISO, and JIS standards in the Toolbox Browser. The gear types include:

- Spur 
- Internal (Spur) 
- Rack (Spur Rectangle) 
- Helical 
- Straight Miter 
- Straight Bevel (Pinion) 
- Straight Bevel (Gear) 

To add a gear to an assembly:

1. Create a new assembly from the Tutorial tab.
2. In the Toolbox Browser  tab, change the **Catalog** to **ISO**, the **Chapter** to **Power Transmission**, and the **Page** to **Gears**.
3. Drag a **Spur** gear into the assembly.
4. Click **OK** to accept the default parameters.

   The spur gear appears in the assembly.



Configure Browser - Colors

You can now set the default color for any catalog, chapter, page, or catalog document. For example, you can choose to add the color green to all the ISO parts that you add to your assembly. Or you can choose to add the color yellow to all the washers for a selected catalog.

To change the color of SolidWorks Toolbox documents:

1. In a part or an assembly document, click **Toolbox**, **Browser configuration**.

   The **Configure Browser** dialog box appears.
2. On the **Browser** tab, expand **Document Properties** and select **Colors**.
3. In the **Applies to** list, expand **ISO** and select **Pins**.

   Note that **ISO** appears with a gray check indicating that one of the chapters below it is selected.
4. Click **Change Color**, select a blue color from the **Color** dialog box, and click **OK**.
5. Click **OK** to close the **Configure Browser** dialog box.

   All ISO pins that you drag into your assembly will be blue.

If you want to change the color of all SolidWorks Toolbox components, change the **Default color** in the **Configure Browser** dialog box.

Configure Browser - Custom Properties

You can add a custom property to any catalog, chapter, page, or catalog document. When you drag a component into an assembly, you can assign a value to this custom property. There are two methods for assigning a property:

- **Choice List**. You populate the list of properties with default values to create a list of values when you insert the component into an assembly. Use **Choice List** to restrict the property to only approved values when you add a component to an assembly. For example, if your custom property is material, you can populate the list with brass, steel, and so on.
- **Input**. You assign a default value which you can change when you insert the component into an assembly. For example, if your custom property is cost, you can assign a value of $1.25. When you insert that component into an assembly, you can change the value if the price has changed.

To assign a custom property:

1. In a part or an assembly document, click **Toolbox**, **Browser configuration**.

   The **Configure Browser** dialog box appears.
2. On the **Browser** tab, select **Custom Properties**.
3. Click **Add**.

4 In the **Add property** dialog box, do the following:

   a) Type **Cost** as the **Property name**.

   b) Set the **Property type** to **Input**.

   c) Click **OK**.

5 In the **Applies to** box, expand **ISO**, **Nuts**, **Hex Nuts**, select **Grade C (4034)** and set **Value** to **1.25**, and click **OK**.

6 Drag a **Grade C (4034)** nut into your assembly from the Toolbox Browser  tab.

   Notice the **Cost** property appears in the list with a default value of **1.25**. You can change this value if you choose.

7 Accept the default properties and click **OK** to add the component to the assembly.

Configure Browser - Part Numbers

You can now create more than one configuration of a part in the SolidWorks Toolbox database with the same part number if the parts are geometrically equal. One common configuration change between components would be the **Thread Display** property. You may want to change the value of this property but retain the same part number in the SolidWorks Toolbox database.

To create two parts with the same part number:

1 In a part or an assembly document, click **Toolbox**, **Browser configuration**.

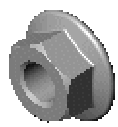
   The **Configure Browser** dialog box appears.

2 On the **Browser** tab, select **Part Numbers**.

3 Select the **Allow duplicate part numbers for geometrically equal components** check box, and click **OK**.

4 On the Toolbox Browser  tab, change the **Catalog** to **ISO**, the **Chapter** to **Nuts**, and the **Page** to **Hex Nuts**.

5 Drag a **Flange** nut into the assembly.



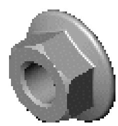
   a) Leave the **Thread Display** as **Simplified**.

   b) Click **Add** to add a **Part Number** called **Flange_Nut**.

   c) Click **OK** to add the part.

      The flange nut appears in the assembly.

6 Drag a second **Flange** nut into the assembly.



   a) Change the **Thread Display** to **Cosmetic**.

   b) Add a **Part Number** called **Flange_Nut**.

      Note that SolidWorks Toolbox allows you to type the same name as before. If the check box in Step 3 were cleared, you could not use the same name twice.

   c) Click **OK** to add the part.

      The second flange nut appears in the assembly.

SolidWorks Utilities

Compare Documents

Compare Documents is a new utility that compares two SolidWorks documents and identifies the following property types:

- **File properties**. Properties in the **Summary Information** dialog box when you click **File**, **Properties**, **Summary** in a SolidWorks document. These properties include **Size**, **Last saved on**, and so on.
- **Document-specific properties**. Properties in the **Summary Information** dialog box when you click **File**, **Properties**, **Custom**, **Mass Properties**. These properties include **Mass**, **Volume**, and so on.
- **Document properties**. Properties in the **Document Properties** dialog box when you click **Tools**, **Options**, **Document Properties**. These properties include **Units**, **Dimensions**, and so on.

To use the Compare Documents tool:

1. Open **\faucet\faucet.sldprt** and **\faucet\faucet_handle.sldprt**.
2. Click **Compare Documents** on the Utilities toolbar, or click **Utilities**, **Compare Documents**.

   The **Compare Documents: Select Documents** dialog box appears.
3. Under **Document1**, select **faucet** from the list.
4. Under **Document2**, select **faucet_handle** from the list.
5. Click **Compare**.
6. In the **Compare Documents: Results** dialog box, do the following:
   - **a)** Expand **File Properties**.
   - **b)** Click **General**.

     The general properties for both documents appear in the **Details** box.
   - **c)** Click **Part Properties**.

     The properties of both parts appear in the **Details** box.
7. Close the dialog box.

Compare Features

The **Compare Features** utility now supports curve-driven patterns and sheet metal parts.

Compare Geometry

The **Compare Geometry** utility supports multibody parts. For more information, see Chapter 5, “Multibody Parts.”

Feature Paint

The **Feature Paint** utility now supports the following feature types:

- Curve-driven patterns
- Lofts
- Scales
- Shells
- Sweeps

Geometry Analysis

The **Geometry Analysis** utility supports multibody parts. For more information, see Chapter 5, “Multibody Parts.”

Find/Modify/Suppress

The **Find**, **Modify**, and **Suppress** utilities now support the following feature types:

- **Curve-driven patterns**. You can now use the **Find**, **Modify**, and **Suppress** utilities for curve-driven pattern features.
- **Lofts**. You can now specify parameters for loft features and use the **Modify** utility.
- **Sweeps**. You can now specify parameters for sweep features and use the **Modify** utility.

Power Select

The **Power Select** utility now supports the following:

- Use the **Feature name** filter to select features by name.
- Select the **Pick color from graphics** check box to select a color for the **Face color** or **Feature color** filters.

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SolidWorks 2001Plus Service Pack Enhancements

This appendix contains information about new and changed functionality introduced in the Service Packs between the release of SolidWorks 2001Plus and the release of SolidWorks 2003.

3D Instant Website

You can publish more than one model at a time. You can either publish each model to its own web page, or publish multiple models to the same web page.

Assemblies

Envelopes

A new option, **Select Components in Top Assembly Only**, is available in the **Apply Envelope** dialog box. Use this option to treat sub-assemblies as a single entity for selection with envelopes. When this check box is selected, you can apply an envelope to an entire sub-assembly if one or more of its components meets the selection criteria.

Over Defined Mates

There is now a yellow flag on a mategroup with one or more over defined mates and mates that are all satisfied. In earlier service packs, a red flag was shown on the mategroup.

Detailing

Blocks

The range of Block display scales allowed in the **Block** PropertyManager was previously limited to 0.1 -10.0. Now the scale can be any positive non-zero number. If you enter an invalid number (zero or negative), the scale is reset to the previous valid number. The spin increment of the **Scale** box is now 0.01 rather than 1.0.

Dangling Dimensions

SolidWorks now always displays dangling dimensions in the system status color set for **Dangling Dimensions** in **Tools**, **Options**, **System Options**, **Colors**. Previously, you could specify a different color for a dangling dimension by using the **Line Color** tool or by moving the dimension into a layer. The **Line Color** or layer color was displayed when toggling the **Color Display Mode** tool on the Line Format toolbar.

Notes

Editing Notes

During on-screen editing of notes, **Pan**, **Zoom**, and **Rotate** operations are not available.

Linking Notes

When a Note in a drawing is linked to a custom property or dimension and then the property or dimension is deleted, the Note now displays **ERROR!<variable name>**. Other items in the Note are not affected.

If a missing property or dimension is detected when opening a drawing yet to be saved in SolidWorks 2001Plus SP 01 or later, a message asks if you want to break the link. If you choose to break the link, SolidWorks replaces the parametric text with the last known value. If no value is available and the Note contains no other text, the Note is deleted. The message appears only once, and only for drawings saved before SolidWorks 2001Plus SP01. If a broken link is detected in a drawing saved in later releases of SolidWorks, **ERROR!** is appended to the property or dimension name without any message.

A new item on the View menu, **Show Annotation Link Errors**, toggles the display of link errors.

Stacked Balloons

Changes in text, properties, and so on of stacked balloons apply only to the selected balloon or balloons. Previously, changes in properties to one balloon applied to all balloons in the stack. To change all the balloons in the stack, select all of them.

Features

You can delete variable radius control points using the shortcut menu.

Fundamentals

A new option, **Enable selection through transparency**, is available on the **Tools**, **Options**, **System Options**, **Display/Selection** tab. Use this option to select opaque objects located behind transparent objects in the graphics area.

Import/Export

Autodesk Inventor

SolidWorks now supports the import of Autodesk Inventor R5 files.

Virtue Translator

The SolidWorks application no longer supports the Virtue files (.**vtu**) translator add-in. Therefore, the Virtue files translator add-in has been removed from the SolidWorks application.

VRML Translator Options

A new option, **Save assembly with unique component files**, is now available when you save a SolidWorks assembly document as a VRML (.**wrl**) file. When you select this option, SolidWorks saves the assembly document using multiple VRML files.

Installation

AMD Athlon

SolidWorks supports Microsoft Windows XP Professional on systems running the AMD Athlon processor.

Master Setup

You can double-click an icon in the **SolidWorks Master Setup** dialog box to install that item (for example, SolidWorks Toolbox). Previously, you had to select the item, then click **Install**.

Polish Language Support

The SolidWorks software is now supported in the Polish language.

SolidNetWork Licensing

SolidNetWork licensing can now be used with either a parallel port or USB port GLOBEtrotter® FLEXid™ hardware key (dongle). USB dongles are only available upon request for those customers with license servers that do not support parallel port dongles, which are shipped by default to SolidNetWork License customers. USB dongles are not supported on Windows NT 4.0.

SolidWorks eRegistration

You can now register SolidWorks on the Web at http://swcustomer.solidworks.com/swlogin_reg.asp.

Sketching

The shortcut menu item **Inspect Curvature** is now **Show Curvature**, and **Remove Curvature Information** is now **Hide Curvature**.

You can scale the curvature comb for all combs shown in a sketch. Right-click a sketch segment and select **Modify Curvature Scale**. The **Curvature Scale** PropertyManager appears with a slider for adjusting the comb scale.

SolidWorks Toolbox

There are two additions to the shortcut menu that you access by right-clicking a part in the Toolbox Browser:

- **Insert Into Assembly** - You can populate one or more holes in the assembly by pre-selecting the circular edges of the holes, then selecting **Insert Into Assembly** from the shortcut menu.
- **Create Part** - You can create a new part from the Toolbox Browser. The part can be from any catalog, chapter, and page. The part appears in its own window, and you set the size of the part.

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SolidWorks 2003 Document Order Form

Introducing SolidWorks Manual

SolidWorks will ship the *Introducing SolidWorks* manual, as ordered on this form, to registered customers who have upgraded or are on the SolidWorks Subscription Service plan.

NOTE: *Introducing SolidWorks* is a book intended for new SolidWorks users. The book introduces concepts and design processes in a high-level approach. If you are currently using the SolidWorks software in production, this book is likely too basic for your needs.

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