



SolidWorks 2004

What's New

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Introduction

About This Book

This book highlights and helps you learn the new functionality in the SolidWorks® 2004 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 2004 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 2004 software. For enhancements that are not in this book, refer to the *SolidWorks Release Notes* by clicking **Help, SolidWorks Release Notes**. Also, refer to **What's New in SolidWorks 2004** in the *SolidWorks Online User's Guide*.

Using This Book

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

To use the example files:

- 1 Install the SolidWorks 2004 software.



Before you install the software, consult the *SolidWorks Read This First* document for information regarding:

- System requirements
- Registration codes
- Backup copies of SolidWorks documents, templates, sheet formats, or Feature Palette™ items from previous releases

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



The example files are placed in the `<install_dir>\samples\whatsnew` folder. For example, if you need to open **knob.sldprt**, then the full path to that example file is `<install_dir>\samples\whatsnew\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	Headings or titles that are not menu items	Measure . Measure the distance between two entities.
Bold Sans Serif	Any SolidWorks tool, menu item, or example file	Click Insert , Mate References .
<i>Italic</i>	Refers to books and other documents, or emphasizes text	Refer to the SolidWorks <i>Read This First</i> .
	Tip	 When you create a 3D model, first make the 2D sketch, then create the extruded 3D feature.

Moving to SolidWorks 2004

Converting Older SolidWorks Files to SolidWorks 2004

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

The SolidWorks Conversion Wizard automatically converts all of your SolidWorks files from an earlier version to the SolidWorks 2004 format. To access the Conversion Wizard, click the Microsoft® **Start** button, select **Programs, SolidWorks 2004, SolidWorks Tools, Conversion Wizard**.

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

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

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

SolidWorks Service Packs

If you are a SolidWorks subscription customer, you can take advantage of SolidWorks service packs that are regularly posted on the SolidWorks Web site. These service packs contain software updates and enhancements to the SolidWorks 2004 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.

SolidWorks 2004 Highlights

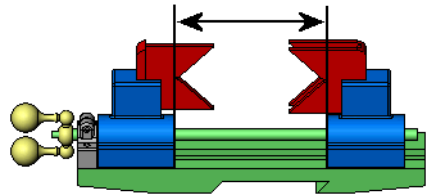
SolidWorks 2004 contains over 250 customer-driven enhancements. This chapter covers some of the highlights:

Assembly Enhancements

Limit Mates

Limit mates allow components to move within a range of values for distance and angle mates.

See **Limit Mates** on page 7-9.



Move and Rotate Components

Move a component by dragging it in the graphics area without opening a PropertyManager or dialog box. Also, you can use a manipulator to move a component along an axis or rotate a component about an axis.

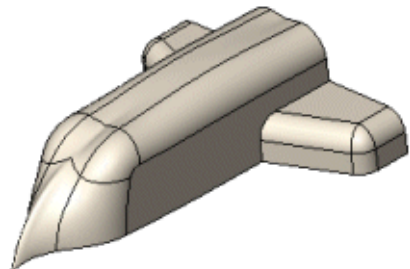
See **Move and Rotate Components** on page 7-3.

Consumer Product Design Tools

Deform Feature

Use the deform feature to alter the shape of a model while retaining relations with respect to the original boundaries. You can pull and stretch a model at a point or deform a model to an existing sketch or curve.

See **Deform Feature** on page 4-9.

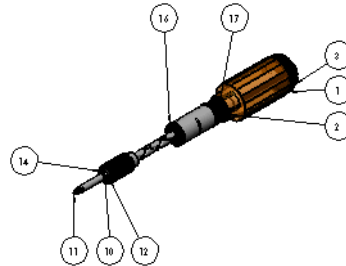


Drawing Automation

AutoBalloons

Add balloons to every component in one or more drawing views in a single operation. You have control over the balloon orientation and alignment.

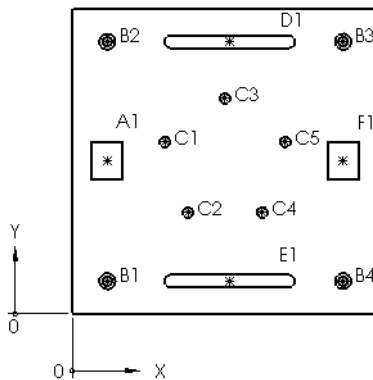
See **AutoBalloons** on page 8-8.



Hole Tables

Hole Tables measure the positions of selected holes from a specified origin datum. Select a face to include all the holes on the face in the hole table.

See **Hole Tables** on page 8-10.



Tag	XLoc	YLoc	Size
A1	33	145.60	
B1	33.16	31.18	$\varnothing 9$ THRU ALL $\varnothing 15 \nabla 4.40$
B2	33.16	258.82	
B3	256.84	258.82	
B4	256.84	31.18	
C1	87.94	163.54	$\varnothing 10$ THRU
C2	109.73	96.46	
C3	145	205	
C4	180.27	96.46	
C5	202.06	163.54	
D1	149.27	258.82	
E1	149.27	31.18	
F1	257	145.60	

Bill of Materials

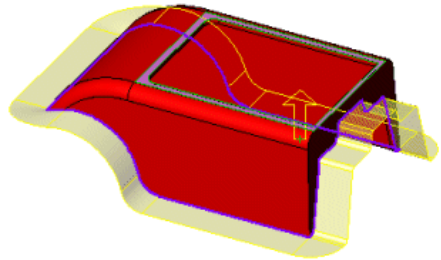
Generate a bill of materials that lists the quantities for each configuration of a part or assembly in a single tabulated BOM. The new BOM tables more accurately retain your custom formats (table splits, word wrapping within cells, and so on.)

See **Bills of Materials** on page 8-11.

Mold Design Tools

Create a mold with a sequence of tools to control the entire mold creation process. Highlights of the new mold design process include the creation of:

- Core-Cavity Block
- Parting Line
- Parting Surface
- Shut-off Surface



See **Mold Core and Cavity Functions** on page 4-12.

Performance

Lightweight Sub-assemblies

Open a sub-assembly as lightweight to reduce the amount of time required to open and work on large assemblies.

See **Assembly Performance** on page 7-12.

Lightweight Drawings

Create drawings while in lightweight mode. The SolidWorks software does not have to load all of the model data to create all types of drawing views or attach annotations to models in views.

See **Lightweight Drawings** on page 8-4.

User Interface Productivity

Enhancements to increase your productivity include:

- **CommandManager.** The CommandManager is a context-sensitive toolbar that dynamically updates based on the toolbar you want to access. By default, it has toolbars embedded within it, based on the document type.
See **CommandManager** on page 2-17.
- **Post-selection.** You no longer have to create and select a sketch before you create a feature. For example, with the **Extrude** feature, you are prompted to select an existing sketch or to select a plane, planar face, or edge for a new sketch.

See **Post Selection** on page 4-4.

RealView

Apply realistic shaders and materials to parts and assemblies. This provides a more realistic environment to visualize designs without prototypes.

See **Display** on page 2-2.



SolidWorks Task Scheduler

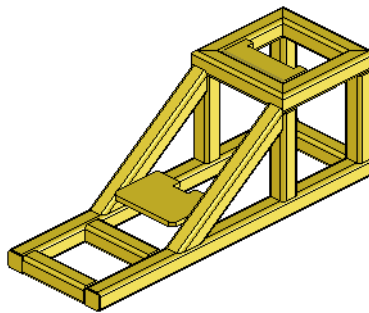
SolidWorks Task Scheduler is a new utility where you set up tasks to run at a future time. You can run tasks such as batch printing, batch import, and batch export at night or anytime your computer is idle.

See **SolidWorks Task Scheduler** on page 9-11.

Weldment Design Tools

Design structural and plate weldments in a single part document. Highlights of the new weldment tools include:

- End Caps
- Fillet Weld Beads
- Gussets
- Structural Members Library
- Trim and Extend Structural Members
- Weldment Cut List



See Chapter 6, **Weldments**.

SolidWorks Fundamentals

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

- ☐ Display
- ☐ Documentation
- ☐ Menus
- ☐ Options
- ☐ Printing
- ☐ SolidWorks documents
- ☐ Toolbars
- ☐ User interface


Display

Color

The **Color** dialog box has been replaced with the **Color And Optics** PropertyManager. The PropertyManager has more options and advanced functionality, including:

- Selection filters
- Color swatches
- Ability to input values for Red, Green, Blue (RGB) or Hue, Saturation, Value (HSV)
- Optical properties such as transparency and ambience
- Configuration control

To apply a color to a model:

- 1 Open **measuring_cup.sldprt**.
- 2 Click **measuring_cup** at the top of the FeatureManager® design tree.
- 3 Click **Edit Color**  on the Standard toolbar.

Measuring_cup.SLDPRT appears in the PropertyManager under **Selection**.

- 4 Under **Color Favorite**:
 - a) Select **50%_palette_pastel** to access the swatch colors.
 - b) Click the color swatch as shown.

This shade of blue is 50% transparent.



- 5 Click **OK** .



Materials


You can apply a specific material to SolidWorks part documents. These materials include steel, iron, plastic, and so on. When you apply a material to a part, the properties propagate to COSMOSWorks, COSMOSXpress, and PhotoWorks™ release 2. They also propagate to mass properties and material properties. See **Materials Interface** on page 5-2.

If you have a supported graphics card, models display in RealView, which applies realistic-looking materials to the model. Otherwise, a material is still applied to the model, but displays with 2D textures.

The following Nvidia® Quadro® graphics cards support RealView display:


- Quadro FX 1000
- Quadro FX 2000
- Quadro4 980 XGL
- Quadro4 900 XGL
- Quadro4 780 XGL
- Quadro4 750 XGL
- Quadro4 700 XGL
- Quadro4 700 Go GL

To apply a material to a part:


- 1 Open **wheel.sldprt**.
- 2 Click **RealView Graphics**  on the View toolbar, or click **View, Display, RealView Graphics**.



RealView Graphics is not available if your graphics card is not supported. If you do not have a supported graphics card, skip step 2.

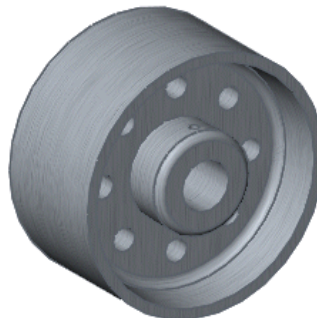
- 3 Click **Edit Material**  on the Standard toolbar.
- 4 In the PropertyManager, under **Materials**:
 - a) Expand **Steel**.
 - b) Click **Cast Stainless Steel**.

Notice the **Physical Properties** in the PropertyManager update based on the material type.

- 5 Click **OK** .




RealView Graphics



Standard graphics

Normal To


You can select a cylindrical or conical face, or any feature created with a single sketch to display the model in a normal to view. Additionally, if you click **Normal to** , then click it again, the model flips 180°. In previous versions, you had to select a plane or planar face to display a model in a normal to view.

Section View

Section views have been enhanced to include:

- Dynamic section views
- Plane rotation
- Colored section faces
- Ability to save section views


To create a section view:

- 1 Open **wheel.sldprt**.
- 2 Click **Section View**  on the View toolbar, or click **View, Display, Section View**.


In the PropertyManager, under **Section 1**, the **Front** plane is selected by default.

- 3 Type **4** for **Offset Distance** .


- or -


Drag the handle as shown so that **Offset Distance**  is **4**.

The face for **Section 1** is blue.


- 4 Select **Section 2** and click **Top** .

The section view updates and the face for **Section 2** is green.

- 5 Enter **35** for **Y Rotation** .

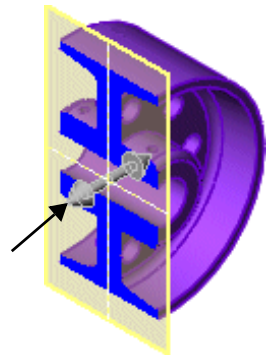
- 6 Click **OK** .



If you click **Section View**  to turn off the section view display, then click it again, the previous state of the section view is applied.

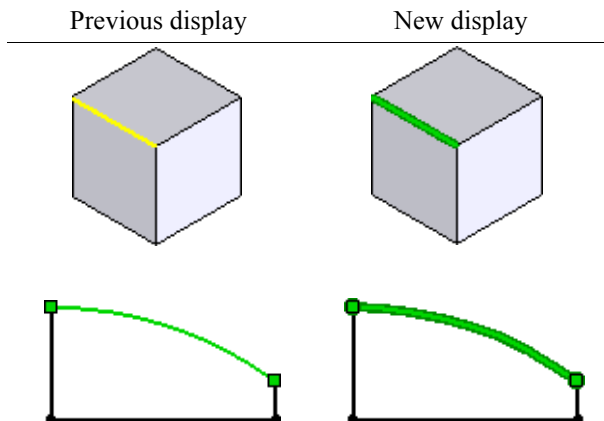
You can also save a section view as a **Named View** in the **Orientation** list.

Zebra Stripes are not available when **Section View** is selected.



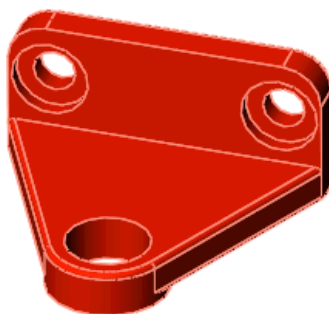
Selecting Entities

Selected entities are easier to identify in the graphics area. Click selected entities in the PropertyManager to highlight them in the graphics area. These examples show how some selected entities are displayed.

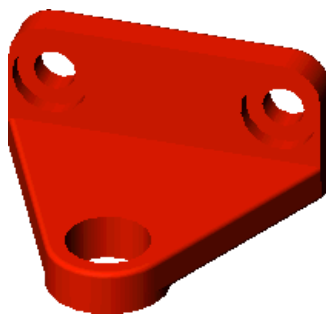


Shaded

The button for **Shaded** view has changed to . Additionally, when a model is in **Shaded** view, the edges are hidden.




Shaded With Edges



Shaded

Shaded With Edges

The **Shaded With Edges** view mode replaces **Display HLR Edges In Shaded Mode**. This mode displays a shaded model and all visible edges. Access this mode by clicking **Shaded With Edges**  on the View toolbar or **View, Display, Shaded With Edges**.



The **Shaded With Edges** button is the same as the **Shaded** button from previous releases. See **Shaded** for information on the new button.

Shaded With Edges is applied on a per-document basis.


Textures

You can apply textures to models for a more realistic display. Textures are 2D images you apply to selected entities of the model.



You cannot apply textures to weld beads or cosmetic threads.

To apply a texture to a model:

- 1 Open **flange.sldprt**.
- 2 Click **Edit Texture**  on the Standard toolbar.
- 3 In the graphics area, click the face as shown.


Face <1> appears under **Selection**.

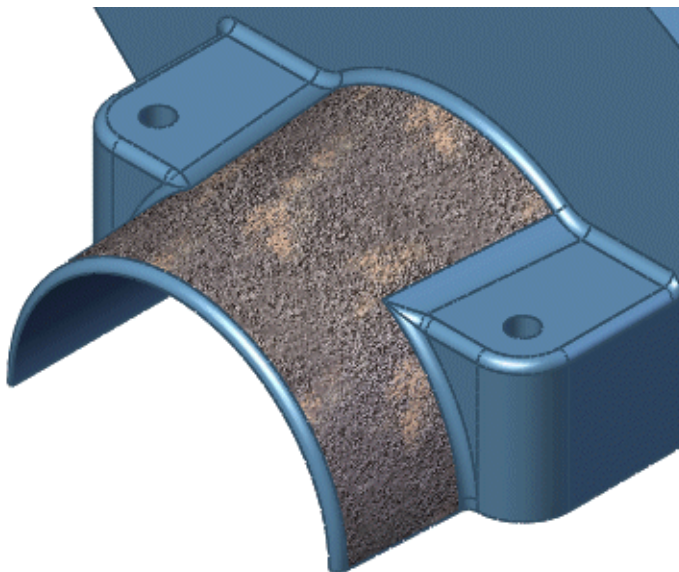
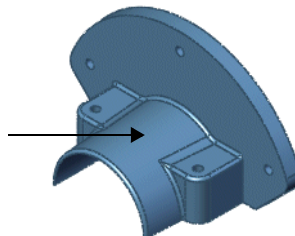
- 4 Under **Texture Selection**:

- a) Expand **Metal** and **Cast**.
- b) Select **Cast Iron**.

- 5 Under **Texture Properties**, drag **Scale** approximately as shown to adjust the texture's grain:



- 6 Click **OK** .



Documentation

Online Tutorial

The *Online Tutorial* has been expanded to include the following lessons:

- Multibody Parts
- PDMWorks™
- SmartFasteners (in the SolidWorks Toolbox tutorial)
- SolidWorks API

Also included are lessons for users with AutoCAD® experience in the AutoCAD and SolidWorks tutorial:

- Importing an AutoCAD Drawing
- Converting an AutoCAD Drawing to 3D
- Creating a Drawing from a 3D Model

Click **Help**, **Online Tutorial** to access these lessons.





Partner Solutions

Click **Help**, **Partner Solutions** to access the **SolidWorks Solution Partner Program** Web site. This replaces the **Partner Products** link on the **Welcome to SolidWorks 2004** screen.

Quick Tips

Quick Tips are a set of pop-up messages that appear while you create SolidWorks documents. These messages give hints and options based on the current SolidWorks mode. Most of the messages have hyperlinks; click a hyperlink to see the associated item in the SolidWorks window.

There are four icons associated with Quick Tips which appear in the lower-right corner of the SolidWorks window:

Icon	Quick Tips status
	Available, but not currently displayed
	Currently displayed
	Not available
	Temporarily disabled


To turn on Quick Tips:

Click **Help**, **Quick Tips**, or click .

To turn off Quick Tips:

Click **Help**, **Quick Tips**, or click .



Quick Tips are mode-specific. For example, if you have Quick Tips displayed when you have a part document open, then you click , Quick Tips are no longer displayed in part modes. However, if you open an assembly document, Quick Tips reappear.

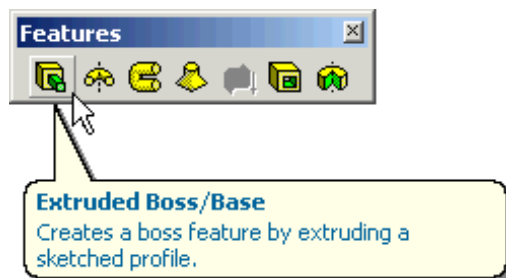
Tooltips

SolidWorks Tools

You have the option to use large tooltips that show both the name of a tool and a brief description of its functionality. In previous versions of SolidWorks, tooltips showed only the name of a tool.

To turn on large tooltips:

- 1 Click **Tools**, **Customize**.
- 2 In the dialog box, select **Show tooltips** and **Use large tooltips**.
- 3 Click **OK**.




Menus

Customization

You can customize each menu in the SolidWorks software so that only the menu items you use are displayed. This functionality is available in main and shortcut menus. When you customize menus, your selections apply to menus in all SolidWorks documents.




By default, not all menu items are shown in every menu.

The double-down arrows  are still available at the bottom of shortcut menus, but you must click them to access the “more commands menu.” You can no longer pause the pointer over the arrows to access the menu.



To customize a menu:

- 1 Click the menu you want to customize and select **Customize Menu**.

Each item currently visible in the menu is preceded by .



This menu is for customization purposes only. You cannot run a command from this menu.

- 2 Click  to clear the check mark next to items you do not want to appear in the menu. The check box appears as  when you move the pointer away from the menu item.
- 3 Click anywhere outside of the menu to save your changes, or press **Enter**. To cancel your changes, press **Esc**.



To display a hidden menu item, repeat step 1 and select the check boxes next to menu items you want to display.

Restructuring

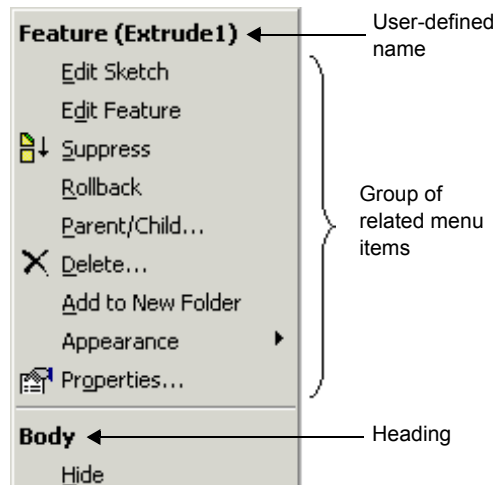
Some menu items have moved to new, more intuitive locations:

Menu Item	New location
Orientation	View, Modify, Orientation
Library Feature	Insert, Features, Library Feature
Bend Table	Insert, Sheet Metal, Bend Table

Shortcut Menu

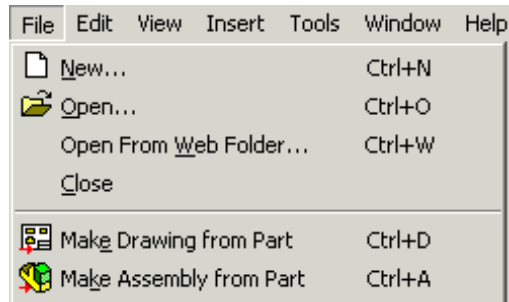
The shortcut menu is restructured so it is easier to find menu items. The restructuring includes:

- **Headings.** Bold headings denote sections of the shortcut menus.
- **User-defined names.** Names that you apply to features, components, and so on appear next to headings. Otherwise, the SolidWorks default name appears.
- **Groups of related menu items.** Menu items that apply to specific contexts are grouped under the appropriate headings.



Toolbar Icons

Menu items have icons based on their toolbar buttons so it is easier to find the items.



Options

Colors

Click **Tools, Options, System Options, Colors** to access new options:

- **PropertyManager Color.** Applies the selected color to PropertyManagers. This option is renamed from **FeatureManager Color**.
- **Use gradient background.** Applies a gradient background to all SolidWorks documents. This option replaces **Use gradient background in parts and assemblies**.
- **Match graphics area and FeatureManager backgrounds.** Applies the same background to the graphics area and to the FeatureManager design tree.

Detailing


See **Options** on page 8-14.

Display/Selection

The option, **No edges**, has been removed from **Tools, Options, System Options, Display/Selection**.

This option was under **Edges displayed in shaded mode**, which has been renamed to **Edge display in shaded with edges mode**.



Click **Shaded**  on the View toolbar, or click **View, Display, Shaded** to display a shaded model without edges.

File Locations

The following file types have been added to the **Tools, Options, System Options, File Locations** list:

- | | |
|-----------------------------|----------------------------|
| • Color Swatches | • Textures |
| • Material Databases | • Weldment Profiles |

General

The **Always show full shortcut menus** option has been removed. You can customize shortcut menus using **Customize Menu** instead. See **Customization** on page 2-9.

Click **Tools, Options, System Options, General** to access the **Use English language feature and file names** option. When selected, feature names in the FeatureManager design tree and automatically created file names appear in English.



This option is available when:

- **Use English language menus** is not selected.
- You are running the SolidWorks software on a foreign language operating system and you installed SolidWorks for that foreign language.

Existing feature and file names in a foreign language do not update when you select **Use English language feature and file names**.

Image Quality

Dialog Box Restructure

Options in **Tools, Options, Document Properties, Image Quality** have been restructured:

Old options	New options
Fast, High Quality , and Custom shading options	One slider to adjust Shaded and draft quality HLR/HLV resolution
Optimal and Custom wireframe options	One slider to adjust Wireframe and high quality HLR/HLV resolution

Optimize Edge Length

A new option, **Optimize edge length (higher quality, but slower)**, is available in **Tools, Options, Document Properties, Image Quality**. Select this option if, after moving the shaded and draft quality slider to the highest setting, you want to increase image quality.



This option causes slower system performance and increased file size.

Units

The units options have been expanded to include multiple unit systems, custom units, and dual units.

To set a unit system:

- 1 Click **Tools, Options, Document Properties, Units**.
- 2 Under **Unit system**, select one of the following:
 - **MKS (meter, kilogram, second)**
 - **CGS (centimeter, gram, second)**
 - **MMGS (millimeter, gram, second)**
 - **IPS (inch, pound, second)**
- 3 Click **OK**.

To set custom units:

- 1 Click **Tools, Options, Document Properties, Units**.
- 2 Under **Unit system**, select **Custom**.
- 3 Select the units to use.
- 4 Click **OK**.

To set dual units:

- 1 Click **Tools, Options, Document Properties, Units**.
- 2 Under **Dual units**, set the units.
- 3 Click **OK**.



To show dual units in the SolidWorks software, click **Tools, Options, Document Properties, Detailing**, and select **Dual dimensions display** under **Dimensioning standard**.

Printing

Page Ranges

When you print multiple drawing sheets, you can print selected sheets. In previous versions, if you wanted to print only sheets 1 and 3, you had to print sheets 1, 2, and 3.

High Quality Views

A new option, **Convert Draft Quality views to High Quality**, is available in the **File, Print** dialog box. When selected, the SolidWorks software converts the current draft quality view into high quality for printing purposes only. This option is available for drawing documents only.

Resolution

When you print a drawing, you can control the resolution of the printed output. However, if the drawing does not contain shaded or draft quality views, the resolution setting may not have a noticeable effect.



When you set a DPI for a drawing, the SolidWorks software creates the entire sheet as a raster image at the selected resolution. This requires significantly more time to print than if you use the default resolution.

Larger sheet sizes result in larger images, which further increases print time.

To control the print resolution for a drawing:

- 1 In an open drawing, click **File, Page Setup**.
- 2 In the dialog box, under **Resolution and Scale**:
 - a) Clear **Default DPI**.
 - b) Select a value in **DPI** to set the resolution. This value applies to the current drawing only.



Only those resolution values supported by your printer are available in the **DPI** list.

- 3 Click **OK**.

SolidWorks Documents

Conversion Wizard


When you use the Conversion Wizard, you select a folder that contains previous versions of SolidWorks documents to convert to the SolidWorks 2004 format. Now you can convert any SolidWorks documents contained in subfolders of the main folder. Select **Include subfolders** so the Conversion Wizard converts all SolidWorks documents in the main folder and its subfolders.

New SolidWorks Documents

The **New SolidWorks Document** dialog box includes two different modes:

- **Novice.** Uses a simpler version of the dialog box and provides explanations of part, assembly, and drawing documents. Select **I am following the tutorial** to use a tutorial template.
- **Advanced.** Uses functionality similar to previous versions.

To access the novice mode of the New SolidWorks Document dialog box:

- 1 Click **New**  on the Standard toolbar, or click **File, New**.
- 2 In the dialog box, click **Novice**. If you are already in novice mode, the **Advanced** button appears.

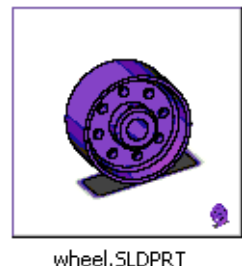
Preview Images

If you use Windows® XP, Windows 2000, or Windows Server 2003, you can see preview images of SolidWorks documents in Windows Explorer®.

To see preview images of SolidWorks documents in Windows Explorer:

- 1 In Windows Explorer, open a folder that contains SolidWorks documents.
- 2 Click **View, Thumbnails**.

Preview images of SolidWorks documents are displayed.



wheel.SLDPRT

SolidWorks Documents in Windows Explorer

Document Information

When you hover the pointer over a SolidWorks document in Windows Explorer, the tooltip displays custom property information about the document.

The following custom properties are displayed:

- **Last Saved By**
- **Description**
- **Revision**
- **Status**
- **Number of sheets**



Last Saved By and **Number of sheets** are shown automatically. You cannot save this information in the **Summary Information** dialog box.

To show SolidWorks document information in Windows Explorer:

- 1 Open **wheel.sldprt**.
- 2 Click **File, Properties**.
- 3 On the **Custom** tab, select **Description**.
Description appears in the **Name** box.
- 4 Type **Caster wheel** for **Value** and click **Add**.
- 5 Repeat steps 3 and 4 for the following properties and values:

Property	Value
Revision	First
Status	Complete



- 6 Click **OK**, then save the part.
- 7 In Windows Explorer, move the pointer over **wheel.sldprt**. You may need to refresh the view to see the document information
The document information is displayed.

Toolbars

Add-In Toolbars

You can activate toolbars for add-in products by clicking **View, Toolbars**. Select the desired toolbars from the menu. To activate the toolbars, you must have the add-in selected in the **Tools, Add-Ins** list.

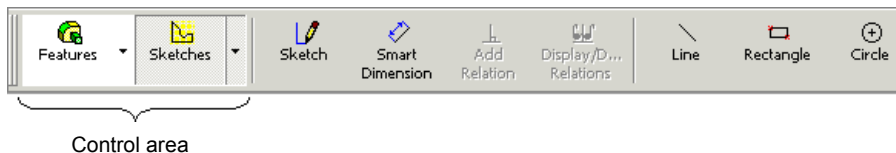
Buttons

The Standard Views toolbar has two new buttons: **Trimetric**  and **Dimetric** .

CommandManager

The CommandManager is a context-sensitive toolbar that dynamically updates based on the toolbar you want to access. By default, it has toolbars embedded within it based on the document type.

When you click a button in the control area, the CommandManager updates to show that toolbar. For example, if you click **Sketches** in the control area, the Sketch toolbar appears in the CommandManager. The image below shows the toolbar button descriptions. You can turn these descriptions off.



Use the CommandManager to access toolbar buttons in a central location and to save space for the graphics area. It can be docked horizontally or vertically.

To access the CommandManager:

- 1 Click **Tools, Customize**.
- 2 In the dialog box, on the **Toolbars** tab, select **Enable CommandManager**.
- 3 Click **OK**.

To use the CommandManager:

- 1 Open **bucket.sldprt**.
- 2 Click **Features**  in the control area.

The CommandManager updates to show the Features toolbar.

- or -

Click  on the Features button in the control area to access the Features flyout toolbar.

To add buttons to the control area of the CommandManager:

- 1 Right-click anywhere in the CommandManager toolbar and select **Customize CommandManager**.
- 2 In the menu, select the toolbars to add.
- 3 Click anywhere outside of the menu to save your changes, or press **Enter**.
- or -
- 1 Click **Tools, Customize**.
- 2 On the **Commands** tab:
 - a) Select **Flyout Toolbars** in **Categories**.
 - b) Under **Buttons**, drag the button to the control area of the **CommandManager** toolbar.
- 3 Click **OK**.



The Layer toolbar is not available in the CommandManager.

To turn off the toolbar button descriptions:

Right-click anywhere in the CommandManager and select **Show Description** to clear the check mark.

Flyout Toolbars

Use flyout toolbars to save space in the graphics area. When you click a flyout button, the toolbar associated with the flyout button appears. You can add flyout buttons to any toolbar, including the CommandManager.


To add a flyout toolbar button to a toolbar:

- 1 Click **Tools, Customize, Commands**.
- 2 In the dialog box, select **Flyout Toolbars** in **Categories**.
- 3 Under **Buttons**, drag a flyout toolbar button to a toolbar in the SolidWorks window.
- 4 Click **OK**.

To access tools with the flyout toolbar:

Click the flyout toolbar button, such as **Standard Views** , then select a tool.

Hidden Buttons

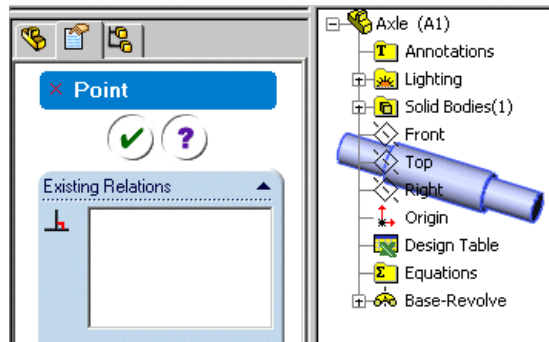
Occasionally a toolbar is longer than the length of your screen. If this occurs, click the arrows  at the end of the toolbar to access the hidden toolbar buttons.

User Interface

Flyout FeatureManager Design Tree

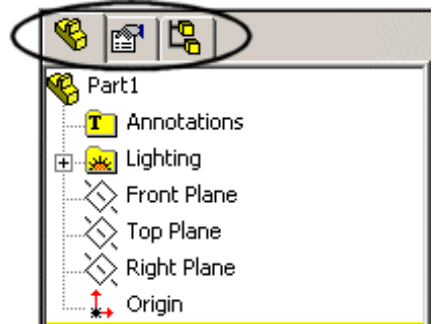
When a PropertyManager is active, the flyout FeatureManager design tree automatically appears.

The flyout FeatureManager design tree is transparent so you can see the model underneath.



Manager View Tabs

Tabs for the FeatureManager design tree, ConfigurationManager, and so on have been moved from the bottom left of the SolidWorks window to the upper left.



Sketching

This chapter describes enhancements to sketching in the following areas:

- ☐ General
- ☐ Dimensions
- ☐ Move or Copy
- ☐ Splines
- ☐ Fit Splines
- ☐ Selection

General

Starting a Sketch

You can select sketch entities (line, circle, and so on) before entering a sketch. If an existing sketch, plane, or planar face is not pre-selected, you are prompted to select one.

When creating the first sketch in a part, the orientation of the planes is trimetric. When you select a plane, the orientation changes to be normal to the sketch plane.

Preview

When you sketch entities using the click-click mode and end a line or arc on the endpoint of an existing sketch entity, the tool stays active but does not show a preview from the last location.

Sketch Toolbars and Menus

The Sketch Tools toolbar has been merged into the Sketch toolbar and references to the Sketch Tools toolbar have been removed. The **Tools** menu still divides the sketch tools into **Sketch Entities** and **Sketch Tools**. The **Auto-activate sketch toolbars** check box has been removed from the **Tools, Customize** dialog box.




Partial Ellipse is the new name for **Center Ellipse**. The tool now appears on the Sketch toolbar.



Create Sketch from Selections is the new name for **Create New Sketch**. The tool has moved from the 2D to 3D toolbar to the Sketch toolbar.

Dimensions

Previously, to access a particular type of dimension such as baseline or ordinate, you would click the **Dimension**  tool, then right-click in the graphics area and select **Baseline**, **Ordinate**, and so on. Now tools for various types of dimensions appear on a new Dimensions/Relations toolbar.



Smart Dimension



Horizontal Dimension



Vertical Dimension



Baseline Dimension



Ordinate Dimension



Horizontal Ordinate Dimension



Vertical Ordinate Dimension



Chamfer Dimension

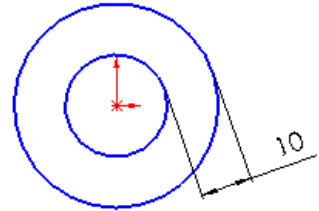


Autodimension

You can also select all the dimension types from the shortcut menu in the graphics area.


The **Smart Dimension** tool also appears on other toolbars, such as Annotation and Sketch. **Add Relation**, **Delete/Display Relations**, and **Scan Equal** have moved from the Sketch Tools toolbar to the Dimensions/Relations toolbar.

You can dimension between concentric circles. The dimension snaps to quadrants of the circles. To toggle between using extension lines or not, right-click *before* placing the dimension. To add extension lines after placing the dimension, right-click and select **Display Options**, **Show extension lines**.





You can mark dimensions in part or assembly sketches to be inserted into drawings. See **Dimensions Marked for Drawings** on page 8-5.

You can control dimension and tolerance precision independently of document precision. See **Options** on page 8-14:

To help define a sketch on the face of a model with **Autodimension** , you can select reference model edges for horizontal and vertical dimensions. Dimensions are applied between the sketch and the model edges as appropriate to define the sketch. The default is now for dimensions to be placed below and to the left of the sketch.

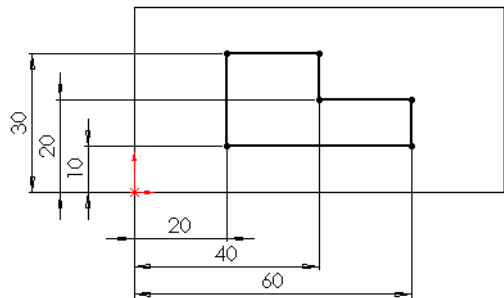
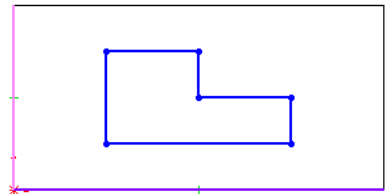
To autodimension a sketch to model edges:

- 1 Open **sketching\sketching_autodimension.sldprt**
- 2 Edit **Sketch2**.
- 3 Click **Autodimension**  on the Dimensions/Relations toolbar.
- 4 In the PropertyManager, note the edges under **Horizontal Dimensions** and **Vertical Dimensions**, with colors matching the colors of the corresponding edges in the graphics area.
- 5 Click **OK** .

All the dimensions appear and the sketch is fully defined.




If **Autodimension** does not fully define a sketch, you can:

- Dimension to a model edge
- Dimension to the origin
- Fix a point



Move or Copy

The **Move/Copy** PropertyManager was previously available only from sketch entity PropertyManagers. Now you can access the PropertyManager (called **Move or Copy**) from the **Tools**, **Sketch Tools** menu, the shortcut menu, or tools on the Sketch toolbar. In each case, you can choose:

-  **Move or Copy**
-  **Rotate or Copy**
-  **Scale or Copy**

The **Move or Copy** PropertyManager opens with **Move**, **Rotate**, or **Scale** chosen based on the tool you selected. In the PropertyManager, you can:

- Select **Move**, **Rotate**, or **Scale**
- Select sketch entities
- Select or clear **Copy** and specify the number of copies
- Specify a scale factor or rotation angle
- Specify a **Base Point**, **Destination**, and whether it is a **Relative offset**


A preview is displayed. You can control rotation angle with a manipulator.


Inferencing is shown to original geometry but not to new copies. When inferencing a rotation to geometry with the manipulator, the exact angle value is displayed.


You cannot copy or scale annotations in part or assembly documents.



Relationships are not added automatically with **Move or Copy** as they are with a drag operation.

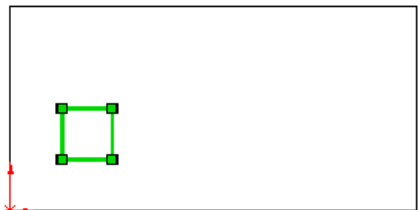
To copy, scale, and rotate sketch entities:

- 1 Open `\sketching\sketching_move_copy.sldprt`
- 2 Edit **Sketch2** and select the four lines of the square.
- 3 Select **Move or Copy**  on the Sketch toolbar.

The four lines of the square are listed in **Sketch items or annotations** .

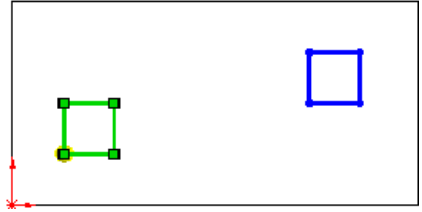
- 4 In the PropertyManager, click **Keep Visible** .
- 5 Under **Operation**, select **Copy** (**Move** is already selected).
- 6 Select the lower left corner of the square to specify the base point.

Under **Base Point**, the **Base Point**  changes to **Base Defined**. The pointer changes to .





- 7 Click in the upper right of the model face to place a copy of the square.

You can place more copies of the square or select another operation.

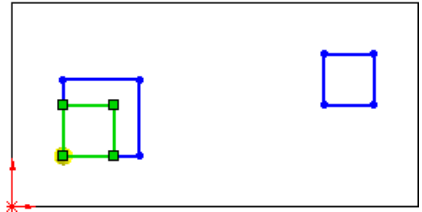


- 8 Under **Operation**, select **Scale**.

- 9 Increase **Scale Factor**  to **1.5**, then click **OK** .



The scale is calculated from the base point.



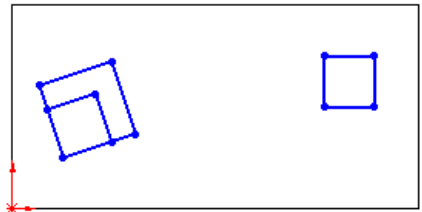
- 10 Click **Select lines or annotations** and select the lines of the scaled copy.



- 10 Select **Rotate**, clear **Copy**, and re-establish the base point at the lower-left corner of the two squares.

- 11 Click in the graphics area and drag the pointer to the top left corner.









Angle snap increments vary depending on the distance from the base point. Exact angle values are displayed when you inference to model geometry.



- 12 Release the pointer and click **OK** , then click **Cancel** .

Splines

A Spline Tools toolbar and **Tools, Spline Tools** menu have been added. The tools include the **Fit Spline** tool from the Sketch toolbar and functionality that was previously on the shortcut menu. The **Spline** tool is still on the Sketch toolbar.

-  **Insert Spline Point**
-  **Show Inflection Points**
-  **Show Minimum Radius**
-  **Show Curvature**
-  **Simplify Spline**
-  **Fit Spline**

Fit Splines

The fit spline is now parametrically linked to underlying geometry so that changes to the geometry update the spline.

Fit splines now fill in gaps in geometry. Previously, splines fit only to continuous geometry. You can specify that the fit spline be closed.

Fit spline chooses the most logical fit to the geometry you select, but you can modify the fit manually. If you select an entity that has been fit, the entity is no longer part of the spline. If you select an entity that is not part of the spline, the spline adjusts to include the selected entity.

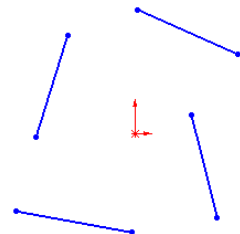
To edit the order of the elements in the fit spline, click **Edit Chaining**. Editing applies only to the order of the non-contiguous spline elements, not the sketch entities.


You can include model edges in addition to sketch entities in a fit spline.

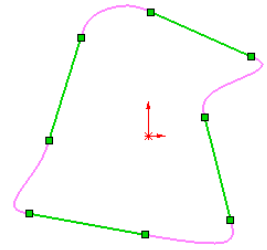
A limitation of fit splines is that you cannot constrain other geometry to the splines.

To fit a spline to non-contiguous sketch entities:

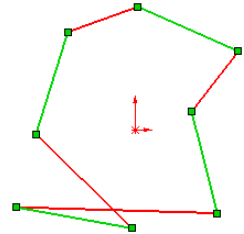
- 1 Open `\sketching\sketching_fit_spline.sldprt`.
- 2 Edit **Sketch1** and select all the sketched lines.




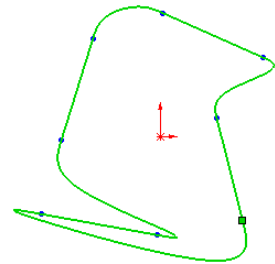
- 3 Click **Fit Spline**  on the Spline Tools toolbar.
The fit spline fills the gaps between the sketched lines.



- 4 In the **Fit Spline** PropertyManager, click **Edit Chaining**.
5 Select the lower two spline segments and click **Edit Chaining** again.
The fit spline is re-ordered.



- 6 Click **OK**  and leave the sketch open for the next exercise.



Selection

Contour Select Tool

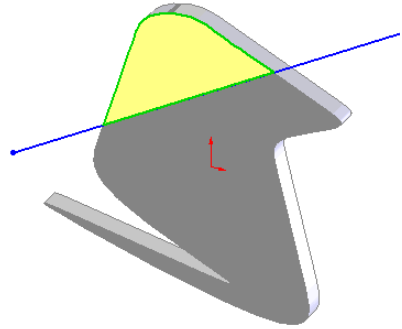
The **Contour Select Tool** has been enhanced to enable selection of model edges so you can select contours made up of sketch entities and model edges.

A tooltip message appears when a contour cannot be selected for one of the following reasons:

- The part has too many edges.
- Edges were created on offset planes.
- Edges are chamfered.
- Edges are filleted.

To select sketch entities and model edges:

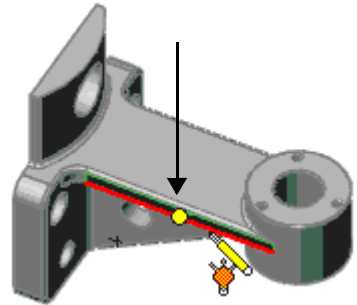
- 1 In the sketch created in the previous section, extrude the closed spline to a depth of **10**.
- 2 Open a sketch on the front face of the part and draw a line approximately as shown.
- 3 Right-click and choose **Contour Select Tool**.
- 4 Select the portion of the face above the sketched line.



Midpoint Selection

Midpoint selection in sketch mode has been improved. When you select a sketch tool and dynamically highlight a sketch entity or model edge that contains a midpoint, a small circle appears indicating the midpoint.

Once you select a midpoint, inferencing to the midpoint works like inferencing to vertices or sketch points.



Features

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

- ☐ General features functionality
- ☐ Deform feature
- ☐ Mold core and cavity functions
- ☐ Lofts
- ☐ Sweeps
- ☐ Dome feature
- ☐ Shell feature diagnostics
- ☐ Extrude direction along vector (2D and 3D sketches)
- ☐ Contour selection and sketch reuse capabilities for additional features
- ☐ Surface trim options
- ☐ Surface fill options
- ☐ Surface drafts

General Enhancements

General enhancements to features and surfaces include:

- ☐ Previews added to face fillets and full round fillets
- ☐ Propagate chamfer along tangent edges
- ☐ Variable radius fillet on surfaces
- ☐ Knit surfaces to solid model
- ☐ Measure option always available
- ☐ Menu availability based on functional availability
- ☐ Post selection functionality extended
- ☐ Delete and replace face options extended
- ☐ Select partial loops for surfaces and solids
- ☐ Revolves with selectable axis of revolution
- ☐ Selectable shaded previews
- ☐ New tools for features

Previews

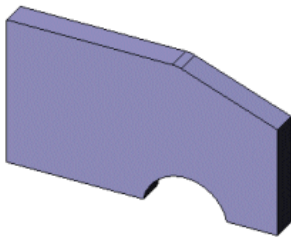
Previews in the graphics area are now available for all fillet feature types, including:

- Face fillets
- Full round fillets

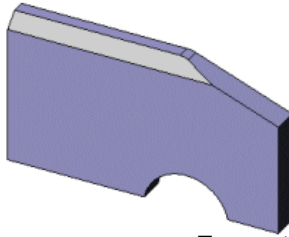
Chamfer

The chamfer feature includes graphic previews as in the fillet feature, and the following:

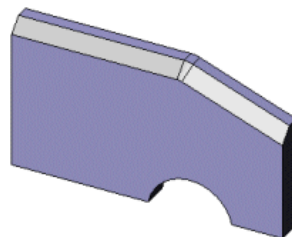
- Select **Full preview**, **Partial preview**, or **No preview**.
- Select **Tangent propagation**.



No chamfer applied



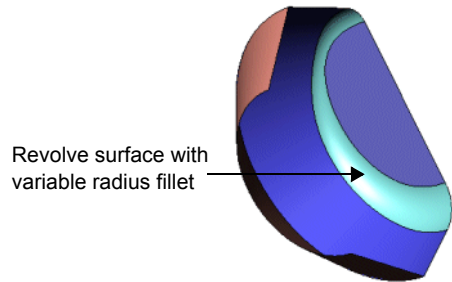
Chamfer applied and **Tangent propagation** cleared



Chamfer applied and **Tangent propagation** checked

Variable Radius Surface Feature



You can now use variable radius fillet with surfaces as well as with solid models.



Knit Surfaces

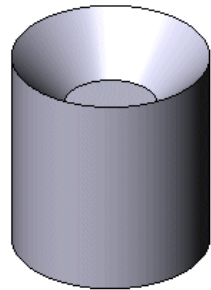
You can immediately create a solid body with the **Knit Surface** tool when the surfaces form a closed volume.

To create a solid from surfaces:


- 1 Create a surface with an enclosed volume.
- 2 Click **Knit Surface** .
- 3 Select the surfaces to knit.
- 4 Check **Try to form solid**.
- 5 Click **OK** .

The following appears in the FeatureManager design tree:

- **Surface-Knit** feature.
- The **Solid Bodies** folder .



Measure Option

The **Measure**  tool is now available whenever you create a feature or when you click **Edit Feature** in the FeatureManager design tree. If the **Measure** dialog box is displayed, you can select valid items in the graphics area, and add them to the **Selected items** list.

Menu Availability

Only menu functions that are compatible with the current geometry are available. Previously, menu choices were available even when the existing geometry rendered those menu functions invalid.

For example, previously **Planar surface**, **Fill surface**, **Composite Curve**, and **Mate Reference** were available in all situations. Now, these features are only available when there is either a sketch, a curve, a surface body or a solid body in the part.

Post Selection


You can now select the following features without pre-selecting a sketch, a face, or a plane:

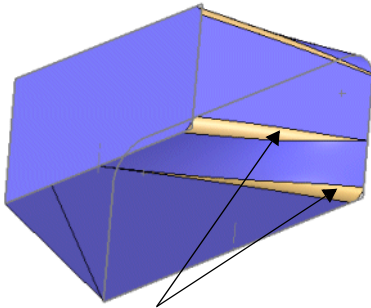
Features	Sheet Metal
• Ribs	• Base Flange
• Extrudes (including base/boss, cut, and surface)	• Miter Flange
• Revolves (solids and surfaces)	• Sketched Bend
• Simple Hole	• Jog

With post-selection, you select the feature and the PropertyManager prompts you to select any of the following, based on the context and the feature:

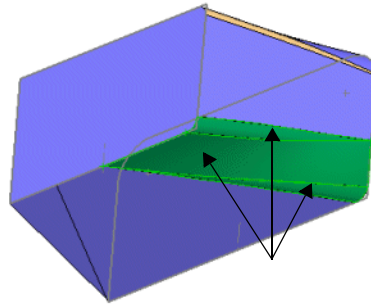
- Plane, planar face, or edge on which to sketch
- Existing sketch with which to create the feature

Delete and Replace Face

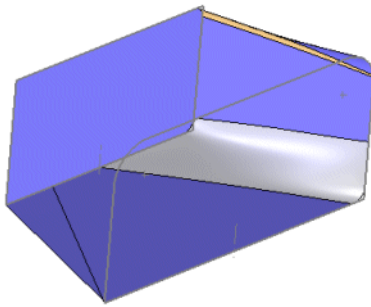
Delete and fill is a new option in the **Delete Face**  tool. **Delete and fill** generates a single face to close any gap. With the **Delete and fill** option, you can also specify **Tangent fill**. This creates the fill with tangent edges as opposed to contact edges. The existing option, **Delete and Patch**, removes faces and trims to close the gap.



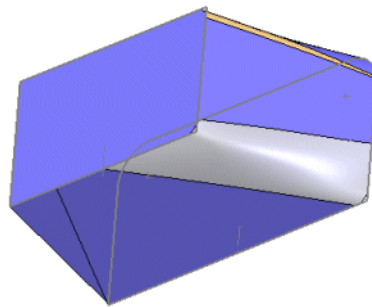
Solid loft with fillets applied to smooth out the edges between faces.



Use **Delete and Replace Face**, and select the three faces shown.



Tangent fill cleared. The system deletes the three faces and replaces them with a single face. The fill uses contact edges.



Tangent fill checked. The system deletes the three faces and replaces them with a single face. The fill uses tangent edges.

Select Partial Loop for Surfaces and Solids

Imported models can contain many small edges. To repair or edit imported models, you often need to select a series of connecting edges.

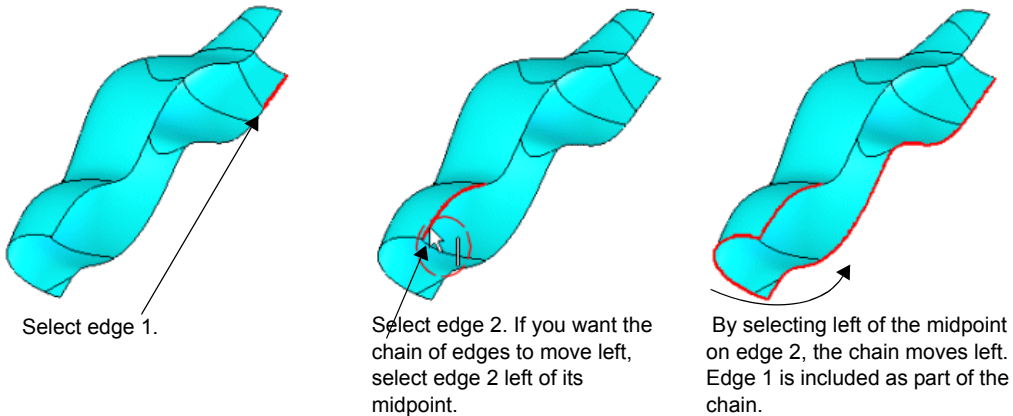
When surfaces or solids contain small edges, you can use **Select Partial Loop** to choose a chain of edges that inter-connect without having to select each edge individually. The direction of the chain is based on the relative position of the second open end edge that you select. If you select the edge left of midpoint, the selection chain moves towards the left. The chain always includes the first edge you first selected.

With solid bodies, the edges you select must be on the same face to use the **Select Partial Loop** option.

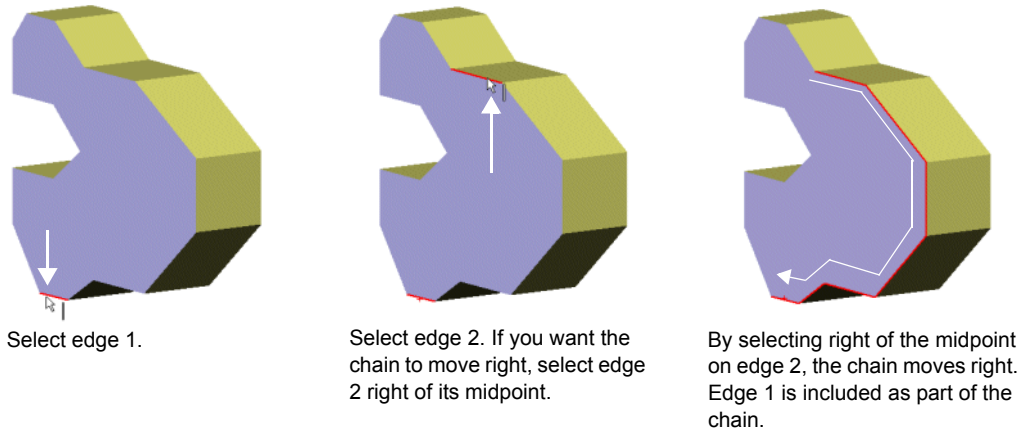
To select a partial loop:

- 1 Select the first edge in the chain.
- 2 Select the second edge, and do either of the following:
 - Position left of the midpoint to create a chain that moves left.
 - Position right of the midpoint to create a chain that moves right.
- 3 Right-click and choose **Select Partial Loop**.



Surface Bodies



Solid Bodies - 3D sketch



Revolves with Selectable Axes of Revolution

You can now revolve around an existing sketch line, eliminating the need to create a construction line as the axis for revolution with **Revolved Boss/Base**  or **Revolved Cut** . You can select any linear entity from the same sketch or from a different sketch on the same plane, as the axis of revolution. In sketches with an existing, single construction line, the construction line is automatically placed as the axis of revolution.

Select Detailed Previews



Improvements to previews enhance detail and allow you to choose what to display. You can select:

- **Highlight new or modified faces.** Inspect what was changed from the previous edit. This option displays any new or modified faces in the model. The modified faces are highlighted in color, as opposed to appearing shaded.
- **Show only new or modified bodies.** Display only new or modified bodies. All other bodies are hidden.

Detailed Preview is available with the following features:

- Ribs
- Extrudes
- Draft

To use detailed preview:

- 1 Click **Detailed Preview**  in the PropertyManager.
Detailed View PropertyManager appears.
- 2 Click **Highlight new or modified faces** or **Show only new or modified bodies**.
- 3 Click **Detailed Preview**  to return to the PropertyManager.

New Tools on the Features Toolbar

The following tools on the Features toolbar activate new functionality:



Deform. Applies global deformation to solid or surface bodies (see **Deform Feature** on page 4-9).

The following new tools were previously available only through the **Insert** menu:



Lofted Cut. Cuts a solid model by adding material between two or more profiles.



Swept Cut. Cuts a solid model by sweeping a closed profile along an open or closed path.



Cut with Surface. Cuts a solid model by removing material with a surface.



Thickened Cut. Cuts a solid model by thickening one or more adjacent surfaces. Adjacent surfaces must first be knit.



Insert Base. Inserts another part as the base feature in the current part.



Library Feature. Inserts a library feature onto a part.



Join. Joins components into a merged part and save as an assembly.



Deform Feature

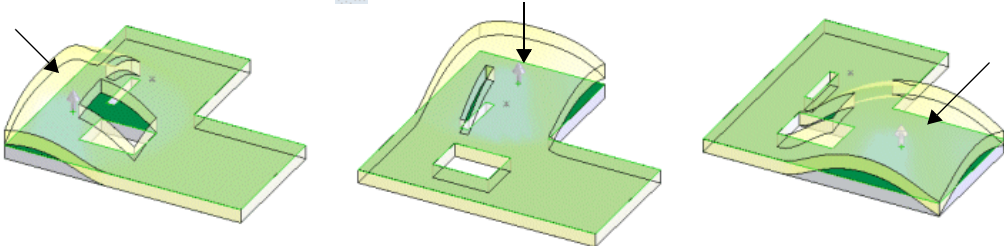
The deform feature allows you to alter the shape of a model locally or globally. You can apply the deform feature to both SolidWorks parts, and to imported parts.

To deform, you first select the **Deform Type** as either **Point to point** or **Curve to curve**.



Point to Point

To deform a single body, select from these options under **Deform point**:

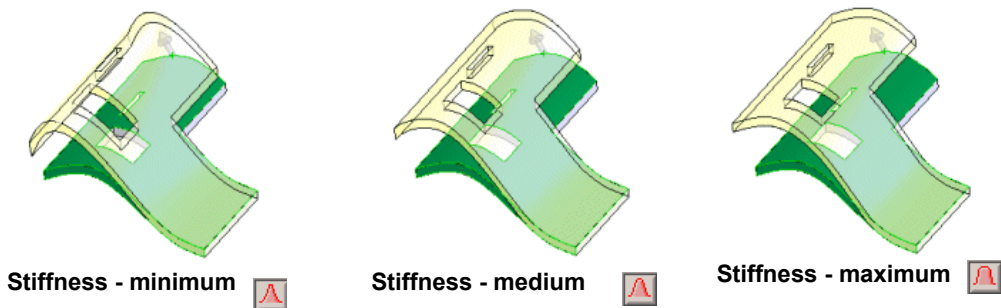
- **Deform point** . Select the point at which the deformation originates. You can select a point on a face, a point on an edge, or a point in space.
- **Deform direction**. Use the default deform direction, or specify a linear edge or a line. With point on a face, the default direction is perpendicular to the face. With point on an edge, the default is the middle between normals on adjacent faces.
- **Deform distance** . Select the magnitude of the pull.



Face selected at three different points. Default deform direction is normal to the face.

To control the deform area, you can specify the **Deform radius** , the **Deform region**, and the **Bodies to be deformed** .

You can control the deformed shape, by choosing one of three levels of **Stiffness**.





Point to point deformation that affects a small area may not display using default image quality settings. Click **Tools, Options, Document Properties, Image Quality**, to increase **Shaded and draft quality HLR/HLV resolution** using the slider. Or click **Optimize edge length**.

Curve to Curve



With the **Curve to curve** type, you base the deformation on initial and target curves. The curves can be open or closed chains of adjacent edges or sketch curves.

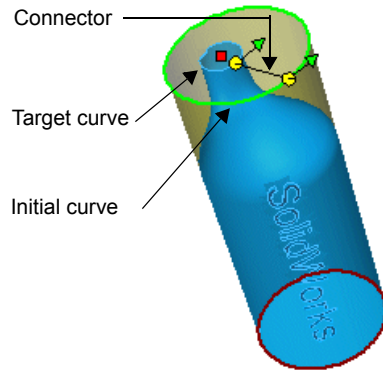
Under **Deform curves**:

- **Initial curve** 
- **Target curve** 

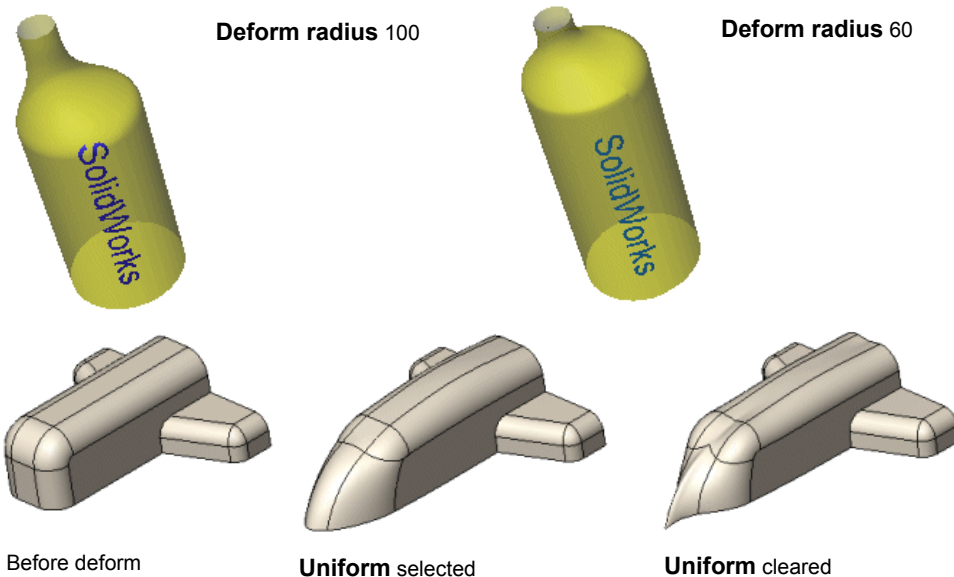
A connector with handles appears between the curves. You can drag the handles along either curve to modify the correspondence between them.

You can also right-click and select **Add connector**.

To control the deform area, you can specify **Deform radius** , the **Deform region**, and the **Bodies to be deformed** .





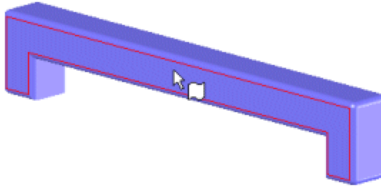
Other options under **Deform region** allow you to specify fixed entities (edges and faces), and additional entities you want to deform. You can also control the shape of the deform using the **Uniform** option.






You can control the deformed shape, by choosing one of three levels of **Stiffness**. You can also modify the influence (weight) between fixed and moving curves.

To apply a deform feature:



- 1 Open **deform_feature.sldprt**.
- 2 Click **Deform**  on the Features toolbar, or click **Insert, Features, Deform**.
- 3 In the PropertyManager, under **Deform Type**, select **Point to point**.
- 4 Under **Deform Point** :
 - Select the front face, as shown.



- Select **Normal to picked**.
 - Set the **Deform distance**  to **50**.
 - Select **Show Preview**.
- 5 Under **Deform Region**, set the **Deform Radius**  to **130**.
 - 6 Under **Shape Options**:
 - Select the **Top Plane** for **Deform axis** .



The **Deform axis** option is only available when the **Deform region** check box is cleared. **Deform axis** restricts the shape of the deformation using the radius you specify as it applies to a cylinder with that radius. **Deform region**, on the other hand, restricts the deformation shape using the radius you specify as it applies to a sphere with that radius.

- Click **Stiffness - medium** .
- 7 Click **OK** .



Mold Core and Cavity Functions

You can now create a mold by using a sequence of tools to control the entire mold creation process. The process is as follows:

Parting Line. Verifies draft and creates the parting line. User interaction includes:

- Specify a pull direction.
- Verify draft with **Draft Analysis**.
- Select an edge that automatically propagates.
- Create parting line.

Shut-off Surface. Creates surfaces to prevent leakage in the core and cavity segments. User interaction includes:

- Select an edge that automatically propagates.



If there is no draft on the faces along the edges of the hole you want to shut-off, the automatic propagation does not work. You need to individually select the edges.

- Fill the holes using shut-off surface.

Parting Surface. Creates the parting surface using the data from the previous two processes.

Core-Cavity Block. Creates the block.

Two additional tools are used for repair:

- **Undercut Detection.** Analyzes and locates undercut faces on solid models. Undercut detection does not detect with imported surface parts, or surface parts generated in SolidWorks. Undercut analysis is not based on the parting line.
- **Ruled Surface.** Construct draft surfaces by inserting a ruled surface derived from an edge.

Mold Tools

Click **Insert, Mold** to access the Mold Tools toolbar. The following tools are new:



Ruled Surface. Inserts ruled surfaces from edges.



Parting Lines. Inserts mold parting line.



Insert shut-off surfaces/Shut-off surfaces. Finds and creates mold shut-off surfaces.



Parting Surfaces. Inserts mold parting surfaces.




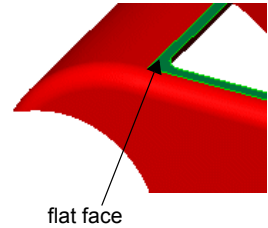
Core/Cavity Solids. Inserts a core/cavity solid feature.





Undercut Detection. Identify faces that form undercuts.

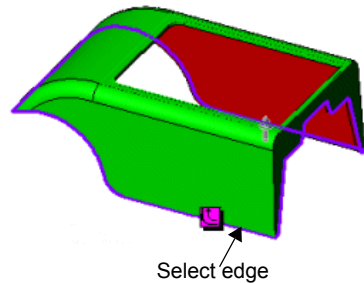
To create the parting line:

- 1 Open **hotrod.sldprt**.
- 2 Click **Parting Lines**  on the Mold Tools toolbar, or click **Insert, Molds, Parting Line**.
- 3 In the PropertyManager, under **Mold Parameters**, select the flat face, as shown, for **Direction of Pull**.
- 4 Click **Draft Analysis**.
- 5 Inspect the model for draft, and compare the colors to those displayed under **Mold Parameters**.




The colors on the model show that all outer faces are **Positive**, and all inner faces are **Negative**. No draft is required.

- 6 Under **Parting Lines**, select the edge shown for **Edges** .
- 7 Click **Propagate**  to select all the edges of the parting line.



- 8 Click **OK** .

To create shut-off surfaces:

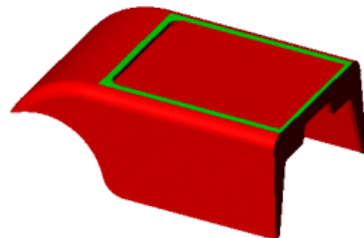
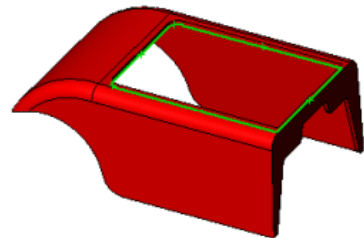
- 1 Click **Insert shut-off surfaces/Shut-off surfaces**  on the Mold Tools toolbar, or click **Insert, Molds, Shut-Off Surfaces**.

All the edges on the perimeter of the area that requires a surface to shut-off, appear under **Edges**.




Knit is checked by default, indicating that all surfaces are knit.

- 2 Click **OK** .




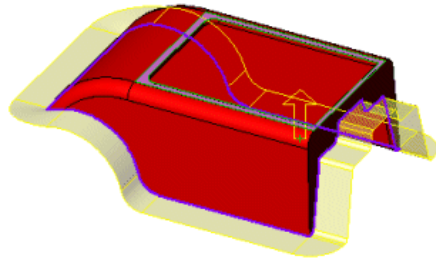
To create the parting surface:

- 1 Click **Parting Surfaces**  on the Mold Tools toolbar, or click **Insert, Molds, Parting Surface**.
- 2 Under **Surface Parameters**, type **20** for **Parting Surface Width**.





In this instance, there is no need to change the **Mold Parameters**. Note that **knit all surfaces** is checked.

- 3 Click **OK** .






To create the core-cavity:



- 1 Select the face, as shown, to open a new sketch.
- 2 Click **Core/Cavity Solids**  on the Mold Tools toolbar, or click **Insert, Molds, Core/Cavity Solids**.
- 3 Click **Normal To**  on the Standard View toolbar.
- 4 Sketch a rectangle, as shown, making sure that it extends beyond the model boundaries, but within the parting surface boundaries.
- 5 Close the sketch to open the **Core-Cavity PropertyManager**.

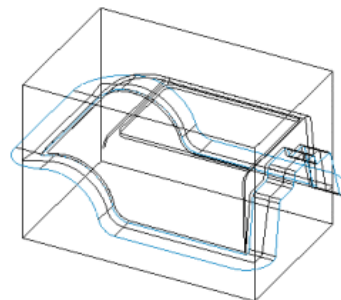
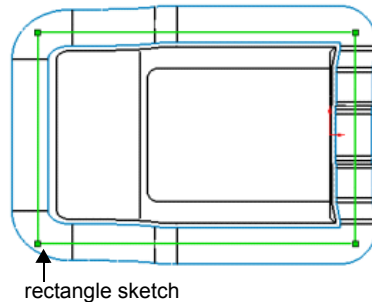
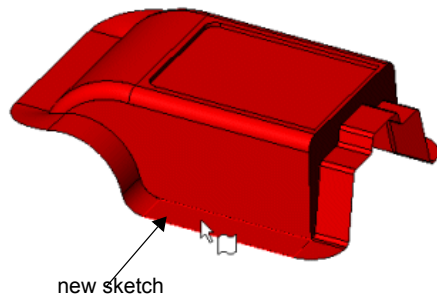


Note how the **Core**, **Cavity**, and **Parting Surface** areas include the sheet bodies.

- 6 Under **Block Size**, type **90** for **Extrusion length in direction 1** , and **30** for **Extrusion length in direction 2** .
- 7 Click **OK** .



Expand **Surface Bodies**  and note that the folder includes **Cavity(1)**, **Core(1)**, and **Parting surfaces(1)**. Expand **Solid Bodies** , and note that the folder includes **Parting Line1**, **Core-Cavity[1]**, and **Core-Cavity[2]**.



Lofts

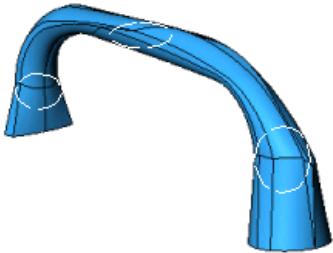
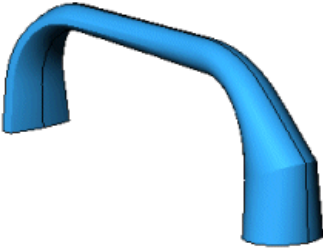
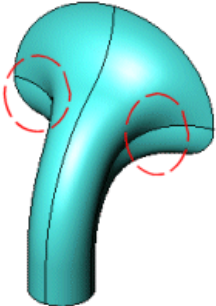

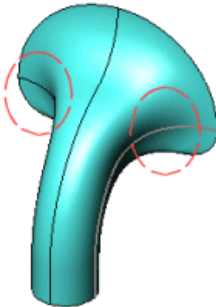
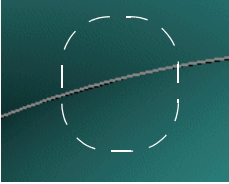
Changes to the loft feature include enhancements to the knitability, and the capability to adjust loft profiles by moving and adding connector points along the profile.

Loft Knit

Improvements to the way in which faces knit include:

- More uniform tangency maintenance
- Better matches between curves and faces
- Improvements to overall performance.

These improvements apply to both solid and surface lofts. There are no options required to create the improved results. The examples in the table below compare the same solid (first row) and surface lofts (bottom row) before and after the improvements to the software.

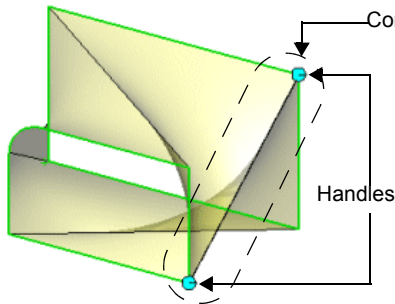
SolidWorks 2003 and Earlier	SolidWorks 2004
 <p data-bbox="178 1043 639 1095">Solid loft. Note segmented intermediate sections and the lack of alignment between profiles.</p>	 <p data-bbox="700 1043 1177 1095">Note the increased smoothness along the loft path and the correct alignment between profiles.</p>
  <p data-bbox="178 1524 655 1550">Note the small fluctuations along the guide curves.</p>	  <p data-bbox="700 1524 1177 1576">Note how the loft more accurately follows the guide curves, and increases the smoothness of the loft.</p>

Loft Synchronization

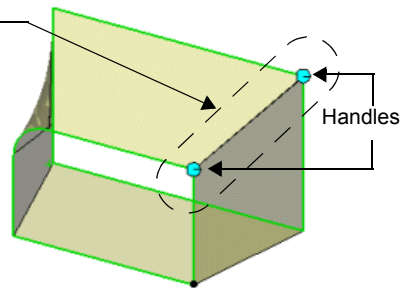
This capability allows you to modify the synchronization between loft profiles. You adjust the synchronization by changing the alignment between the profiles. To adjust the alignment, you manipulate the handles which appear in the graphics area as part of the connector. The connector is the poly line connecting corresponding points in both directions.



The loft synchronization feature is available with new or legacy models.



Undesirable loft. The initial synchronization creates an undesirable alignment between the loft profiles.



Correct loft. Move the handle to modify the connector. This corrects the alignment between the loft profiles.


The type of control you can apply to connectors depends on:

- **Selection sequence.** Whether it is the first connector you modify, or whether you modify connectors that you add to the profile.
- **Type of profile.** Connectors in smooth contour profiles react differently than connectors on profiles with non-tangent edges.
- **Guide curves.** With guide curves and closed contour profiles, the guide curve defines the start point for the profiles.

Connector control is either global or local:

- **Global twist control.** The first connector on a profile with non-tangent edges moves from vertex to vertex along an edge.
- **Local twist control.** When you add connectors to profiles with vertices, those connectors have local twist control. Local twist control means that you can select the connector and drag it along the profile.

To control connectors using global twist control:

- 1 Click a handle  on one of the profiles.

The handle changes color .



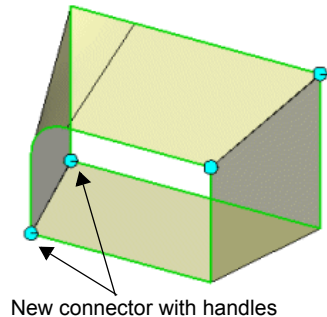
Regardless of the profile you select, the behavior is identical.
However, the shape of the loft varies accordingly.

- 2 Start to drag towards the vertex on which you want to relocate the connector.
The connector moves to the next vertex along the edge you specified.
The loft preview updates with the new synchronization.

To add a connector:


- 1 Select and edge.
- 2 Right-click and chose **Add Connector**.

Once you add connectors to profiles, you can reposition each of these connectors. The number of connectors you can add is based on the number of vertices in the profile.



Once you new connectors to profiles with vertices, or with smooth contour profiles, you can drag the handles to reposition the connector.

To control connectors using local twist control:

- 1 Click a handle  on one of the profiles.

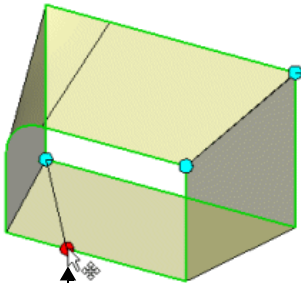
The handle changes color .

- 2 Drag the connector along the edge to the location you want.
- 3 Release the handle.

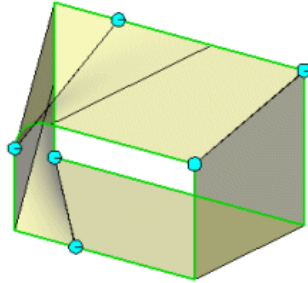
The loft preview updates with the new synchronization.

To move *additional* connectors:

- 1 Select a handle on the new connector.
- 2 Drag the connector along the edge in between the vertices.



Drag the additional connector handles along the edge between vertices.



Add more than one connector and drag the handles along the edge between vertices.

You can also manipulate connectors using the following techniques:

- **Undo Manipulations.** Removes all changes implemented by moving connectors.
Right-click and select **Undo Manipulations**.
- **Hide Connector.** Removes a connector from the profiles, but does not delete it.
Right-click and select **Hide Connector**.
- **Coincident vertices.** If you have coincident vertices, you need to select the corresponding vertex to move the initial handle.
Press **Shift** and select the handle to twist the corresponding connector.

Sweeps

Improvements to the sweep feature include:

Sweep Knitability

New sweep features created along existing edges knit more accurately to existing solids. Sweeps now follow their paths and the guide curves more closely.

Sweeps with Guide Curves without Pierce Point

The **Pierce Point** relation is no longer required for sweeps with guide curves. You can use guide curves without specifying a pierce point between the guide curve and the profile sketch. You can also create the guide curves and the profile sketches in any order. Previously, the guide curve sketch always had to be created first.

Orientation/Twist Type with Follow Path Option

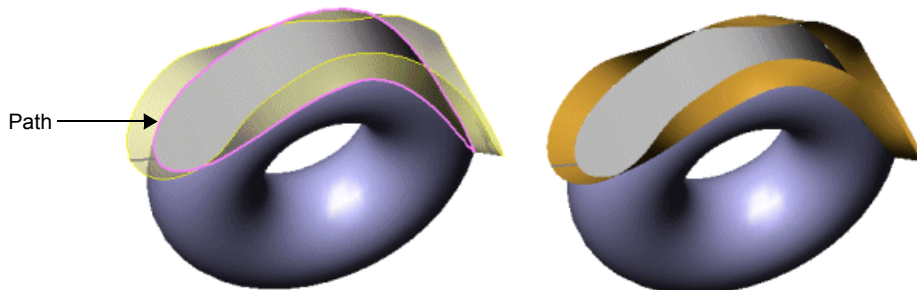
Three new options are available under sweep **Options**, when you select **Follow path** as the **Orientation/twist type**. These options stabilize the profiles when small and uneven curvature fluctuations along the path cause the profiles to misalign.



Profiles always maintain a normal to the path. Profile fluctuations are due to uneven curvature fluctuations along the path. Unless you correct the fluctuations by selecting one of the new options, the profiles follow the path and shift to adjust to the misaligned path.

The three options are:

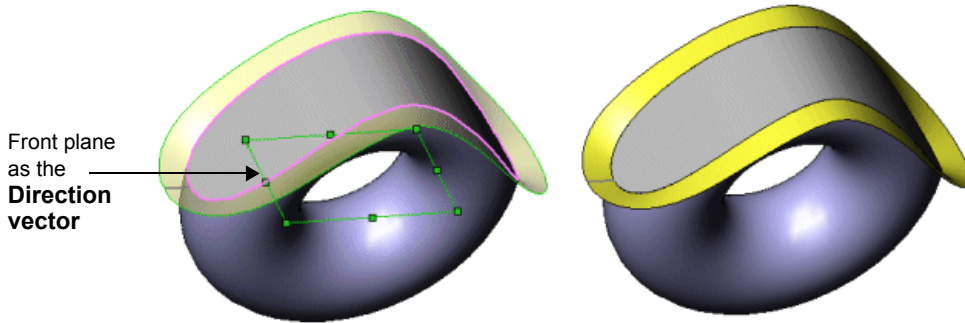
- **None.** The default value, indicating that no correction is applied.



Surface sweep. **None** is selected as the **Orientation/twist type** option. The profile fluctuates preventing a smooth sweep.

- **Direction vector.** Select a plane, planar face, or line to define the vector. When you select a plane or planar face, the normal of the plane or planar face is applied. If you select a line, the line direction provides the direction vector.

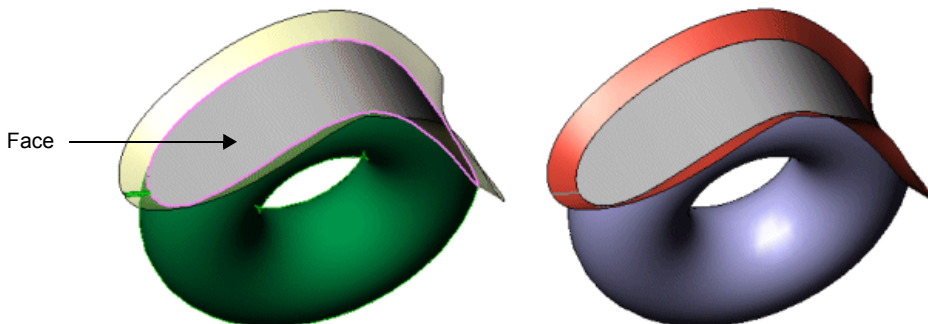
The section plane always includes the direction vector as its X-axis. The normal of the plane, the Z-axis, is the plane that contains the direction vector and the path tangent at that point.



Surface sweep. **Direction Vector** selected as the **Orientation/twist type**. The **Front** plane is the **Direction vector**. The profiles do not fluctuate, remaining aligned along the entire path, and creating a smooth sweep.

- **All faces**. If the path includes neighboring faces, you can select **All faces**. When the path includes 2 alternative faces, click **Next Face** to switch faces.

The section plane always include the face normal as its Y-axis. The normal of the plane, the Z-axis, is the tangent of the path at that point.



Surface sweep. **All faces** selected as the **Orientation/twist type**. The profiles do not fluctuate, remaining aligned along the entire path, and creating a smooth sweep.

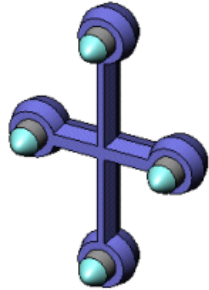
Dome Feature

The Dome feature now uses the PropertyManager instead of a dialog box. Additional enhancements to the dome feature include:

- Select any type of face as the basis for the dome
- Select multiple faces
- Select a constraint sketch
- Select a direction
- Use centroid located outside the face



Previously, the dome feature failed if the centroid was located outside the face.



Shell Feature Diagnostics

The shell feature now displays new error messages and includes new tools to help you identify why the shell feature failed. The new diagnostic tool **Error Diagnostics** is available in the **Shell** PropertyManager.

Under **Diagnosis scope** select:

- **Entire body**. To diagnose all areas in the model.
- **Failing faces**. To diagnose and identify the faces that failed to shell.

To run error Diagnostics:

- 1 Click **Error Diagnostics**.
- 2 Select **Entire body** to diagnose all areas of the model

or

Select **Failing faces** to diagnose and identify the faces that failed to shell.

- 3 Click **Check body/faces** to run the diagnostic tool.

The results display in the graphics area, using a callout to pinpoint the specific area on the model that needs correction.




For example, the shell feature can fail because the **Thickness** at a point is too large relative to one of the selected faces. A message that displays the minimum radius of curvature indicates that the shelling thickness is too large at that point.

You can also select **Display mesh** or **Display curvature**.

- **Display mesh** shows a uv mesh. This mesh operates in the same manner as the **Preview mesh** option in **Filled Surface**.
- **Display curvature** shows a curvature plot for the bodies.

Because problems related to surface gaps and curvature often relate to surface inconsistencies, you can click **Go to offset surface**. This opens the **Offset Surface** PropertyManager where the offset surfaces in the model are displayed.

Extrude Direction Along Vector

Both the **Base/Boss Extrude**  and the **Extruded Surface**  tools include a **Direction of Extrusion**  option. You can specify extrude direction with 2D and with 3D sketches by specifying a direction vector. Direction vectors include:

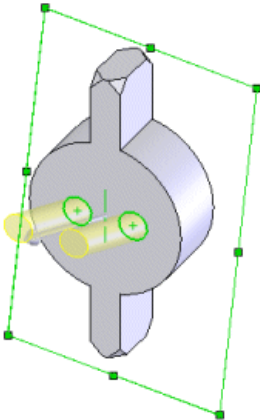
- Cylindrical faces
- Linear edges
- Reference axes
- Points in reference geometry
- Conical faces
- Linear sketch entities
- Reference planes
- Vertices
- Planar faces
- Sketch points



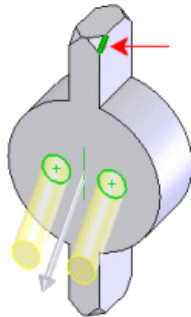
With either 2D or 3D sketches, the extrude direction cannot be perpendicular to the normal of the base sketch plane.

Extrude Direction with 2D Sketches

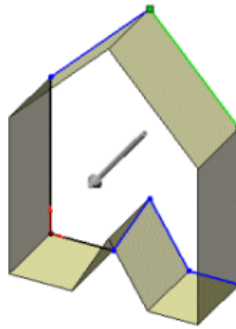
You specify the **Direction of Extrusion**  for the 2D sketch selecting any of the direction vectors.



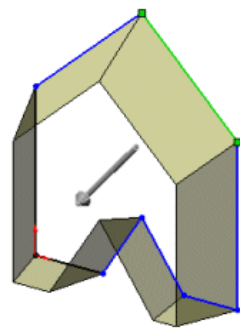
Plane as Direction of Extrusion



Edge as Direction of Extrusion



Standard surface extrude




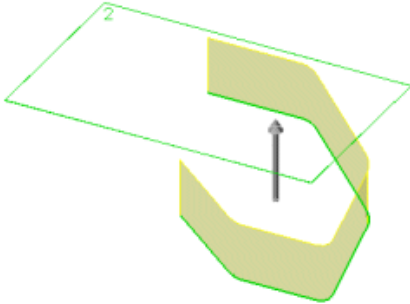
Standard surface extrude with draft



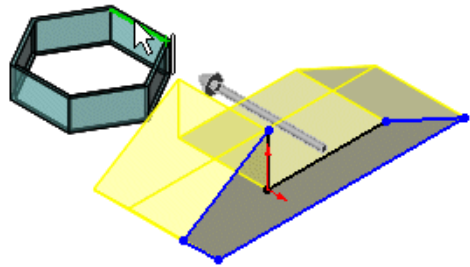
You can select the **Draft on/off**  option, and a **Direction of Extrusion**  with both solid and surface extrusions.

Extrude with 3D Sketches

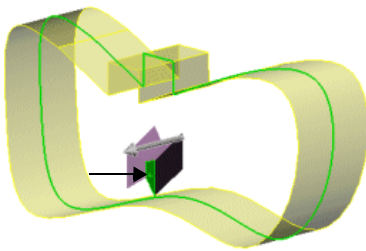
You can extrude 3D sketches with both surface and solid geometry by specifying the **Direction of Extrusion** . Valid direction vectors are the same as with 2D sketches (see page 4-22).



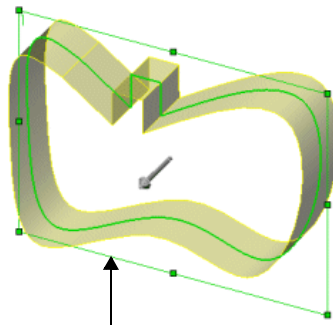
3D sketch for **Surface Extrude** using the **Top Plane** as the **Direction of Extrusion**



3D sketch for **Base/Boss Extrude** using an edge as the **Direction of Extrusion**



3D non-planar sketch for **Surface Extrude** using a face (example on the left), and a plane (example on the right) as the **Direction of Extrusion**




Contour Selection and Sketch Reuse


The capabilities of contour selection for sketches with intersecting profiles, and re-using the same sketch to create multiple features were extended to the following features:

- Lofts
- Sweeps
- Filled Surface
- Ribs
- Planar Surface
- Trimmed Surface
- Projected Curve
- Split Line

Enhancements to these features enable you to do the following:

- Use 3D edges and sketch geometry with **Contour Select** .
- Select a single 3D profile with lofts and sweeps to use a single sketch as both profile and guide. See **Contour Select Tool** on page 3-7.

Surface Trim Options

Changes in the **Trimmed Surface**  tool on the Surfaces toolbar facilitate the trimming process. Improvements in the graphics display, along with new options produce more accurate results. Interface changes include:

- Renamed trim types
- Added selection options
- Added surface split options

Trim Types.

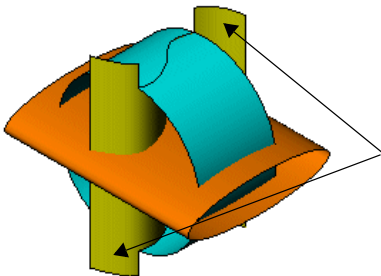
Trim Tool is now called **Standard**.

Mutual Trim is now called **Mutual**.

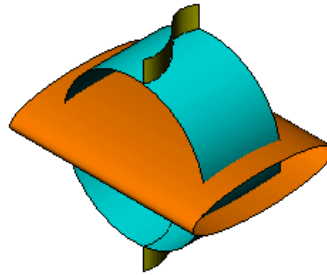
Selections

Under **Selections**, there are two new options to facilitate selection of which surface bodies to keep or discard.

- **Keep selections.** Indicates that you want to retain the surface bodies listed in the **Pieces to Keep** area.
- **Remove selections.** Indicates that you want to discard the surface bodies listed in the **Pieces to Remove** area.



Mutual - **Keep selections**



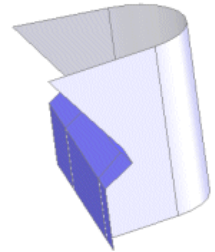
Mutual - **Remove selections**

Surface Split Options

The **Surface Split Options** allow you to control the shape of the boundary edges, which can be **Natural** or **Linear**. You can also select **Split All**.

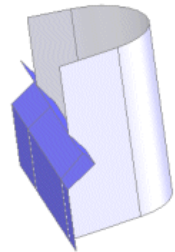
Natural. Trim and force the boundary edges to follow the shape of the surface.

Natural

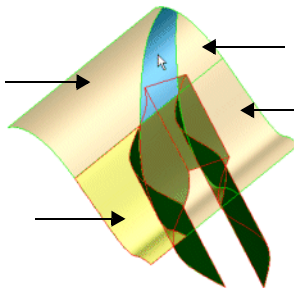


Linear. Trim and force the boundary edges follow a linear direction from the trim point.

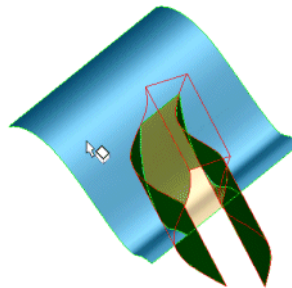
Linear



Split All. Display all possible splits in the surface.




Split All checked. All possible splits are displayed.



Split All cleared. A single surface is displayed.

Surface Fill

The **Filled Surface**  feature, similar to **Knit Surface**, can now create a solid. The surface fill is knit to adjacent surfaces. New options include:

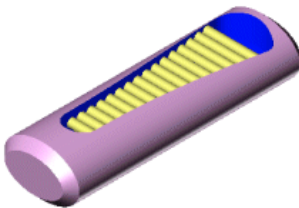
- **Merge result**
- **Try to form solid**
- **Reverse direction**

Merge result

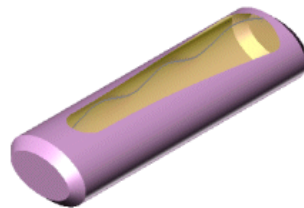
The behavior for this option depends on the boundaries.

- When all the boundaries belong to the same solid body, you can use the surface fill to patch the solid.
- If at least one of the edges is an open sheet edge and you select **Merge result**, then the **Surface Fill** knits with the surfaces to which the edges belong.

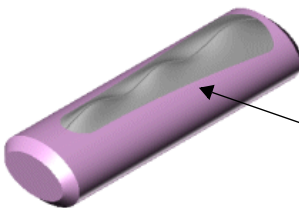
If all the boundary entities are open edges, then a solid may optionally be created.



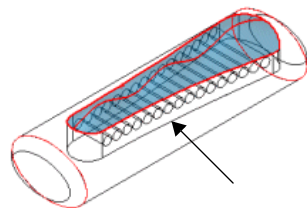
Model without **Filled Surface** applied. Note the features inside.



Model with **Filled Surface** applied and **Merge result** checked. The surface fill joins the solid, removing the features inside.



Model with **Filled Surface** applied and **Merge result** cleared. The surface fill creates a new surface, and the features inside remain.



The advantages of **Merge result** include:

- One less operation, as **Replace face** is eliminated.
- Hide solid details within the model.

Try to form solid

Forming a solid is possible if all the boundary entities are open surface edges. The option **Try to form solid** is cleared by default.






Reverse direction

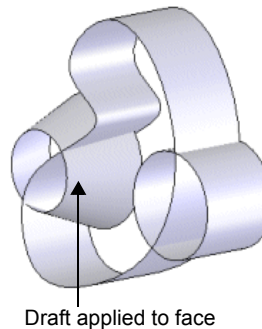
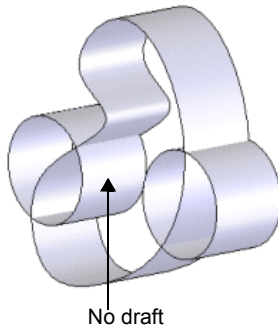
When patching a solid with a fill surface there are often two possible results. Click **Reverse direction** to correct if the fill surface displays in an unwanted direction.

Surface Drafts

You can now apply the draft feature to surfaces. Functionality is the same as applying draft to a solid model.

To apply surface draft:

- 1 Click **Draft**  on the Features toolbar, or click **Insert, Features, Draft**.
- 2 In the PropertyManager select:
 - **Type of Draft**
 - **Draft Angle** 
 - **Neutral Plane** 
 - **Faces to Draft** 
- 3 Click **OK** .

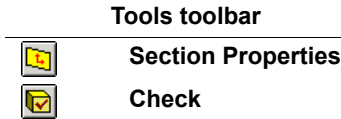


This chapter describes enhancements to parts in the following areas:

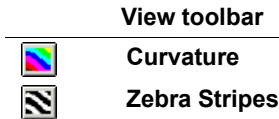
- ☐ Tools
- ☐ Materials interface
- ☐ Custom properties
- ☐ Hole Wizard
- ☐ Assemblies from multibody parts

Tools

The following new toolbar tools were added:



Previously, **Section Properties**  and **Check**  were only available through the Tools menu.




Previously, **Curvature**  and **Zebra Stripes**  were only available when you selected a face, right-clicked and selected the function.

Materials Interface

The ability to define a part's material for Mass Properties, CosmosXpress, or BOMs is now centralized through the **Materials Editor** PropertyManager. This tool uses a database in which the following attributes are stored:

- **Materials.** Includes the ability to create a new material.
- **Visual properties.** Includes **Advanced graphics**, **Part color**, and **Cross hatch**.
- **Physical properties.**

To apply a material to a part:

- 1 Open **materials_apply.sldprt**.
- 2 Click **Edit Material**  on the Standard toolbar.
- 3 In the PropertyManager, under **Materials**, select **Aluminum Alloy** to expand the materials category and choose **1060 Alloy**.




Visual Properties. If you have the appropriate graphics card on your system, **Real View** is selected when you have **Advanced Graphics** selected. Otherwise, **Standard** is selected. **Real View** creates a photo-realistic rendered image. If you do not have the appropriate graphics card, a color is applied to the model that simulates the material you selected (see **Materials** on page 2-3 for information about graphic cards).

- 4 Under **Visual Properties**, make sure **Advanced Graphics** and **Cross hatch** are selected. The **Cross hatch** pattern is assigned, based on the selected material.



You have the option to manipulate other factors such as:

- **Change the texture scale** .
- **Change the angle of the texture** .

- 5 Under **Physical Properties**, note that the **Property** and **Value** are assigned based on the material you selected.
- 6 Click **OK** .



If you have **Real View**, the image is photo-realistic, displaying how light is reflected off the selected material.

- 7 Click **Tools, Options, Document Properties, Material Properties** to see the material properties.

Custom Properties

Improvements to the **Custom** tab in the **Summary Information** dialog box include:

- When you type a **Name** and a **Value** to add a new **Property**, you can click **OK** to add the new property without first clicking **Add**.
- If you type a **Name** and a **Value** to add a new **Property**, and then select another name from the list without clicking **Add**, the first property is automatically added.
- When you modify an existing custom property, those changes are saved when you click **OK**. You no longer need to click **Modify** to apply the changes.

Hole Wizard

Enhancements to the **Hole Wizard** include:

- Custom standards
- Color Codes for non-default values
- Graphical threads

Custom Standards

You can now create your own standards for Hole Wizard holes by deriving from an existing standard. To customize existing standards, Hole Wizard uses the same database that is used by the SolidWorks Toolbox add-in application (see **Alternate Part Names** on page 10-15).



However, you do not need to have SolidWorks Toolbox installed on your system to create new standards for Hole Wizard holes.

To create your own Hole Wizard standards:

- 1 Click **Tools, Options, System Options** tab, **Data Options**.
- 2 Click **Edit Standards Data**.
The **Configure Data** dialog box appears.
- 3 On the **Standards** tab, type a name in the **New standard name** box.
- 4 Select an industrial or national standard from the **Create from list**, then click **Create**.
The new standard name appears in the **Derived Standards** list. Your new standard includes data from the selected industrial or national standard.
- 5 Click **Apply** to save your changes and continue editing, or click **OK** to save your changes and close the **Configure Browser** dialog box.
- 6 To edit your standard, switch to the **Edit Data** tab.

To delete one of your standards:

Select the standard from the **Derived Standards** list and click **Delete**.




To edit the data in a user-created catalog:

- 1 Click **Tools, Options, System Options** tab, **Data Options**.
- 2 Click **Edit Standards Data**.
- 3 On the **Edit Data** tab, select a user-created catalog from the tree.
- 4 Expand the catalog, then the chapters and the pages beneath the catalog to select the **Hole Wizard** hole that you want to edit.

Each row in the table refers to a **Hole Wizard** configuration.

- 5 To change a value, click in the table and edit the value.



For example, under **Bolts and Screws** , **Machine Screws** , **Hex Screw - Ansi** , you can change the **Size** name, the **Pitch**, the **Diameter**, and so on.

- 6 To add a new row of data, scroll to the bottom of the table and type values in the row with the asterisk.
- 7 To delete a row of data, click the button at the beginning of the row and press **Delete**.
- 8 Click **Apply** to save your changes and continue editing, or click **OK** to save your changes and close the **Configure Data** dialog box.

Color Codes with Changed Values

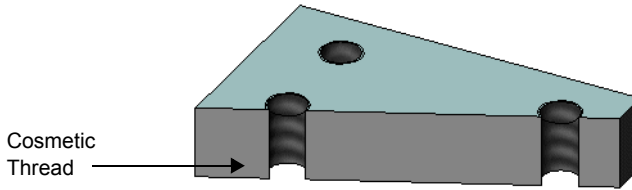
Each **Hole Wizard** hole type (except **Legacy**), includes standard, assigned values such as **C'bore Diameter & Depth** or **Head Clearance**. These default values are based on the hole type, **Standard** and **Size**. When you edit an assigned values, the area highlights with color to indicate that the value is not the originally assigned value.



You can reset any value you changed by re-selecting the **Size**.

Graphical Threads

You can display graphical threads, based on the attributes of the hole wizard hole or the cosmetic thread.



Assemblies from Multibody Parts

You can now select a multibody part comprised of solid bodies, and create an assembly as single step without inserting a split feature.

To create an assembly from multibody parts, click **Insert, Features, Create Assembly**. This opens the **Create Assembly** PropertyManager, and adds a new **Create Assembly** entry in the FeatureManager design tree.

Use the **Create Assembly** feature to do any of the following:

- Create additional features in the multibody part after implementing the **Create Assembly** feature. These additional features do not appear in the individual part files or as a part of the assembly document.
- Select previously created split features when those features resulted in bodies saved as part files. Those parts are included when you use **Create Assembly**.
- Specify names for each body and the assembly, as well as select the location of the part files.
- Include some or all of the bodies as part of the create assembly feature.
- Change the relative locations of the parts and automatically reflect this change in the assembly.



The parts always remain associative to the bodies in the multibody part. However, if you rename or delete the assembly, this data becomes obsolete.

- Add parts, assembly level features, or part level features to the assembly.




The additional features are saved as part of the assembly or as individual part files. The features are not written to the original multibody part.

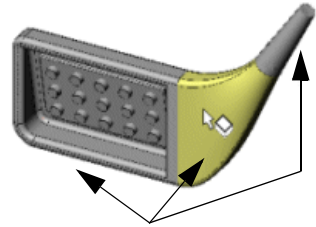
- Insert multiple **Create Assembly** features in the FeatureManager design tree.

To create an assembly from multi-body parts:

- 1 Open **Multibody_to_assembly.sldprt**.
- 2 Click **Insert, Features, Create Assembly**.

The pointer changes to  and the **Create Assembly** PropertyManager appears.

Select each of the three body parts to include in the assembly.




You can also select each part from the **File** list under **Resulting Parts**.



Callouts. Display the default path and file names.

Path. Location of your multibody part.

File names. Listed in the **Solid Bodies**  folder in the FeatureManager design tree.

- 3 Under **Resulting Parts**, in the **File** list, double-click each file to activate and open the **Save As** dialog box. You can select a new location and file name for each part.
- 4 Under **Create Assembly**, click **Browse**, and select a folder to save the assembly as **SplitAssembly** type (*.sldasm).
- 5 Click **OK** .

In the FeatureManager design tree, the following entities are listed:

- Under **Solid Bodies** , **Create Assembly1[1]**, **Create Assembly2[2]**, and **Create Assembly3[3]**.
- The assembly document, **Create Assembly** .

Weldments

This chapter describes functionality used to design weldments. Multibody parts allow the creation of a weldment model as a single part. Weldments, which are common to all machines, are composed of plates, tubes, and extruded profiles. Areas included:

- ☐ Design functionality
- ☐ Weldment toolbar
- ☐ Weldment feature
- ☐ Creating a weldment part
- ☐ Drawings of weldments

Design Functionality

Design functionality for weldments supports the following:

- Creating members of structural weldments.
- Creating plates by sketching on a profile face or a plane and extruding the sketch.
- Adding other features such as cuts, holes, and so on.
- Creating full length, intermittent, and staggered fillet weld bead features. Weld beads include system generated weld symbol annotations.
- Grouping solid bodies into cut list items to display on a cut list with a weldment drawing.
- Specifying a list of custom properties that are applied to cut list items.
- Breaking single weldments into groups of sub-weldments, and optionally saving each sub-weldment as a separate model.
- Creating drawings of weldments that include annotations with weld symbols and balloons.
- Creating a cut list in the drawings.

Weldment Toolbar

You can activate weldment commands by either using the menu (**Insert, Weldments**), or the new Weldment toolbar that includes:



Weldment Creates a weldment feature, designating the part as a weldment part.



Structural Member Creates a structural member by sweeping pre-defined profiles along user-defined paths.



Trim Trims or extends structural members using adjoining structural bodies/faces as the trim tools.



End Cap Creates an end cap feature using the end faces on open structural members.



Gusset Adds a gusset feature between two planar adjoining faces.



Fillet Bead Adds a fillet weld bead feature between two disjoint bodies.



Creating the first structural member feature inserts the weldment feature.

By default, the Weldment toolbar also includes the following tools:



3D Sketch



Extruded Boss/Base



Extruded Cut



Hole Wizard



The default Weldment toolbar also includes the Reference Geometry **Flyout Toolbar** (see **Flyout Toolbars** on page 2-18)

Weldment Feature

The weldment feature sets up a design functionality environment. It is not a feature in the same context as a fillet is a feature in the SolidWorks environment.

When you insert a weldment feature, it activates the multibody environment by clearing the **Merge result** check box in the PropertyManager of features that add material. The weldment feature also acts as a placeholder for common custom properties that are inherited by all cut list items.

Weldments include the following attributes:

- **Configurations.** The system creates two default configurations as soon as you create a weldment feature in a part. Inserting a weldment feature in a part creates two configurations: a parent configuration called (**Default <As Machined>**) and a derived configuration called (**Default<As Welded>**). *Default* is the name of the configuration.

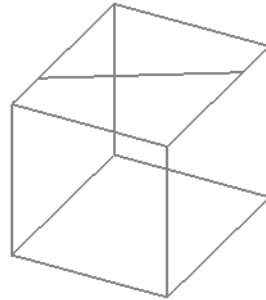
You can rename the **Machined State** and the **Welded State** configuration in the FeatureManager design tree. The attributes for both weldment configurations are identical to the configuration attributes that exist with any SolidWorks' configurations.

- **Custom properties.** A weldment feature defines a place holder for a default list of custom properties that are inherited by each body (see **Custom Properties** on page 6-14).

Creating a Weldment Part


In this section you create a weldment part using the 3D layout sketch shown. The weldment part includes:

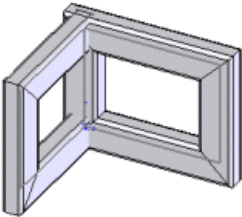
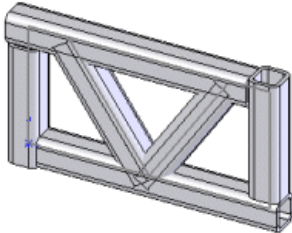
- Structural members
- Trim and extend feature
- End caps
- Gussets
- Weld beads



Adding Structural members

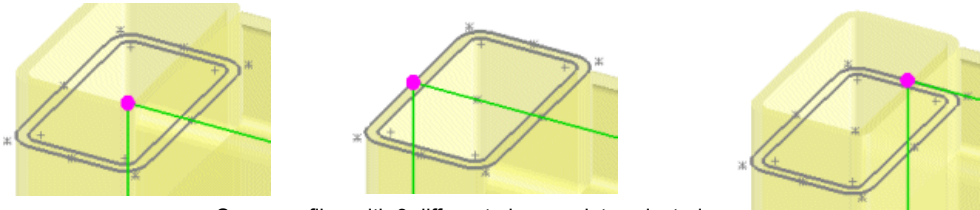
All structural members include the following attributes:

- Structural members use profiles, for example an **angle iron**.
- Each profile is identified by **Type** and by **Size**. Each profile **Type** includes multiple sizes. The **Size** is divided into **iso** and **ansi inch** designations.
- Structural members can include multiple segments, but all segments can use only one profile.
- Multiple structural members, each with a different profile, can belong to the same weldment part.
- Structural members appear in the FeatureManager design tree as **Structural Member1**, **Structural Member2**, and so on. Bodies created by the structural members appear under **Solid Bodies** .
- You can create your own profiles (**Type** and **Size**), and add them to the existing library of weldment profiles.
- Weldment profiles are located in: *installation dir\data\weldment profiles*.
- Structural members allow you to specify the pierce point of the profile, relative to the sketch segment used to create the structural member (see page 6-5).

Sample Structural Members	
	
rectangular tube (ansi inch) -50 x 30 x 2.6	square tube (ansi inch) 2 x 2 x 0.26

Pierce Points

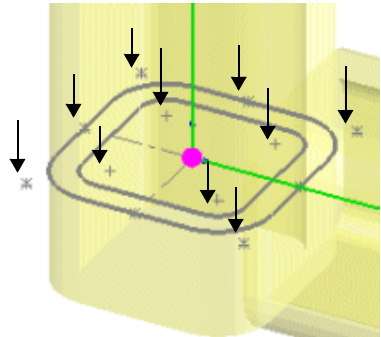
The pierce point defines the location of the profile, relative to the sketch segment used to create the structural member. The default pierce point is the sketch origin in the profile library feature part. Any vertex or sketch point specified in the library feature profile can also be used as a pierce point.



Same profiles with 3 different pierce points selected.

To change a pierce point:

- 1 Open an existing weldment part with structural members.
- 2 Select a structural member in the FeatureManager design tree, and click **Edit Feature**.
- 3 In the PropertyManager, under **Settings**, click **Locate Profile**.
The system zooms in to the appropriate area, with the last profile you attached joined using the default pierce point.
- 4 Select any of the pierce points on the profile, or a sketch point that you added.
- 5 Click **OK**



Sample **square tube** profile with 11 pierce points. Pierce point is located at the default.

End Conditions

You can change end conditions and combine multiple end conditions within the same structural member. End conditions define how two profiles are joined.

End Conditions

End Trim	End Miter	End Butt1	End Butt2

First, you create **Structural Member1** and **Structural Member2**.



To create a weldment part - Structural Member1:

- 1 Open **weldment_square.sldprt**.



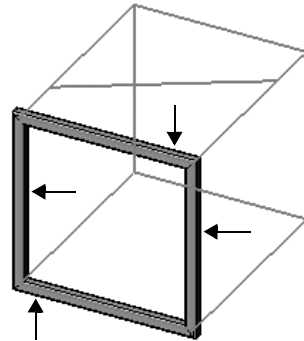
The model has a weldment feature, with the **(Default <As Machined>)** configuration displayed. To see the second configuration, **(Default<As Welded>)**, click **ConfigurationManager**.




- 2 Click **Structural Member**  on the Weldment toolbar.
- 3 In the PropertyManager click **Keep Visible** .
- 4 Under **Selections**:

- Select **square tube (iso)** in **Type** for **Profile Type**.
- Select **30 x 30 x 2.6** in **Size** for **Profile**.
- Select each edge as shown for **Path segments**.



- 5 Under **Settings**, select **Apply corner treatment** and click **End Miter** .



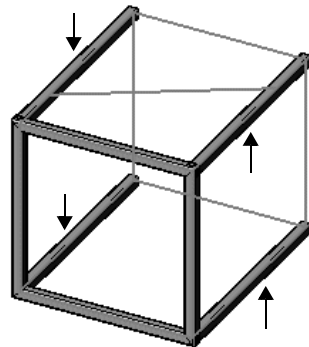
- 6 Click **OK** .

Structural Member1 appears in the graphics area and the FeatureManager design tree. **Path segments** is cleared for the next structural member.

To add Structural Member2:



- 1 Under **Selections**, use the same **Type - square tube (iso)** and **Size - 30 x 30 x 2.6**, as in the first procedure.
- 2 Select each edge as shown for **Path segments**.
- 3 Under **Settings**, click **End Butt1** .
- 4 Click **OK** .

Structural Member2 appears in the graphics area and the FeatureManager design tree. **Path segments** is cleared for the next structural member.






For **Structural Member3**, you create another structural member and change two of the end conditions.

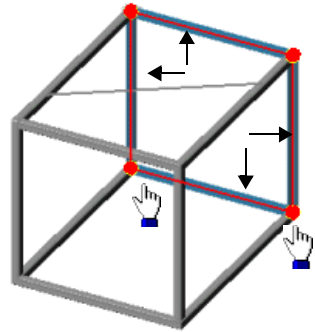
To add Structural Member3:

- 1 Under **Selections**, use the same **Type - square tube (iso)** and **Size - 30 x 30 x 2.6**, as in the first procedure.
- 2 Select each edge as shown for **Path Segments**.
- 3 Click **End Butt1** .
- 4 Select one of the two bottom points.
 - The pointer changes to , as shown.
 - The End condition pop-up toolbar appears.





- 5 Click **End Butt2** , and click  to apply the new end condition.
- 6 Repeat steps 4 and 5 to change the **End condition** for the second point.
- 7 Click **OK** .

Structural Member3 appears in the graphics area and the FeatureManager design tree. **Path segments** is cleared for the next structural member

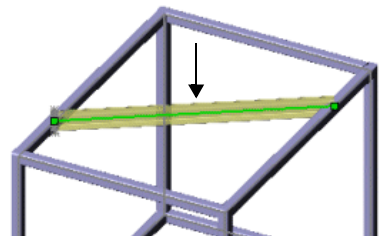


For **Structural Member4**, you add another structural member to the same model by specifying a different **Type** and **Size**.

To add Structural Member4:

- 1 Under **Selections**:
 - Select **rectangular tube (iso)** in **Type** for **Profile type**.
 - Select **50 x 30 x 2.6** in **Size** for **Profile**.
 - Select the sketch entity as shown for **Path Segments**.
- 2 Click **OK**  and click **Cancel**  to exit the PropertyManager.

Structural Member4 appears in the graphics area and the FeatureManager design tree.






Trim/Extend

You can trim structural members so they butt up correctly in the weldment part. To trim a structural member, select the structural member as the **Bodies to be Trimmed** and a **Face** or **Bodies** to act as the **Trimming Boundary**.

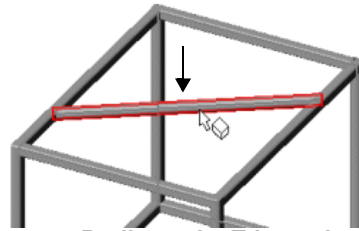


Selecting **Face** as the **Trimming Boundary** is normally more efficient and offers better performance.


To apply trim/extend - 1:

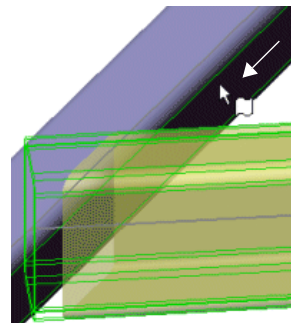
- 1 Click **Trim/Extend**  on the Weldment toolbar.
- 2 In the PropertyManager, click **Keep Visible** .
- 3 Under **Corner Type**, click **End Trim** .
- 4 Select the **rectangular tube (iso)** structural member created previously for the **Bodies to be Trimmed**.

Solid Bodies appears in the **Bodies List**.



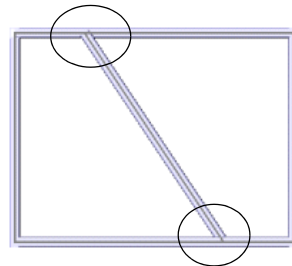
Select **Bodies to be Trimmed**

- 5 Under **Trimming Boundary**, select **Face**, and in the graphics area, select the **rectangular tube (iso)**, as shown.
- 6 Click **OK** .



Select **Trimming Boundary**



- 7 Repeat steps 4 through 6 to trim the **rectangular tube (iso)** structural member on the other side.

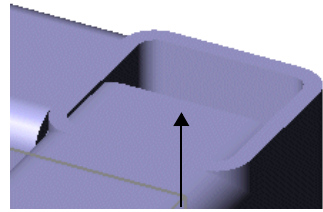


Trim/Extend applied to both sides

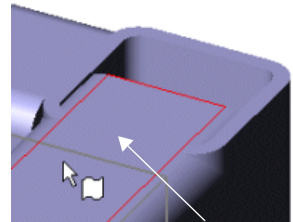
You also need to trim the three open profiles where the structural members extend into the adjoining profiles. Later, you add end caps to these structural members.

To apply trim/extend - 2:

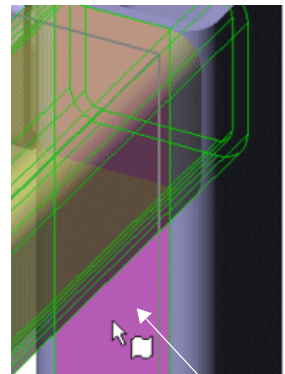
- 1 Zoom to the open profile of **Structural Member3**.
- 2 Select the **square tube (iso)** structural member for the **Bodies to be Trimmed**.
- 3 Under **Trimming Boundary**, select **Face**, and in the graphics area, select the right face on the **square tube (iso)** structural member, as shown.
- 4 Click **OK** .
- 5 Repeat steps 1 through 4 to trim the remaining two **square tube (iso)** structural members. The open structural members are located on the top and bottom left structural members.
- 6 Click **Cancel**  to exit the PropertyManager.



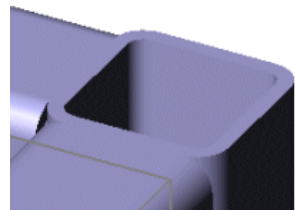
Trim structural member



Select face as **Bodies to be Trimmed**



Select face as **Trimming Boundary**



Trim/Extend applied









End Caps

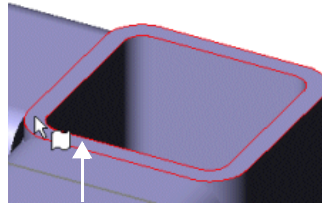
You add end caps to close off open structural members. When you add end caps, you can specify the offset distance of the end cap as a ratio of the thickness of the structural member that is being capped. If you do not specify **Use thickness ratio**, you specify an explicit offset dimension. You can also add chamfered corners.



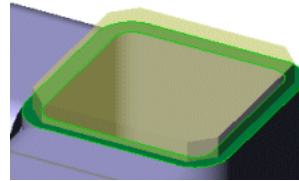
You can only apply end caps to profiles with linear edges.

To add end caps:

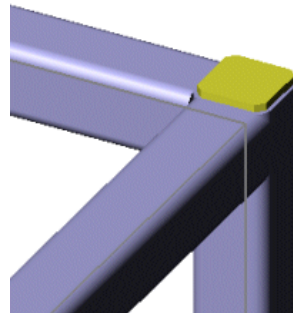
- 1 Click **End Cap**  on the Weldments toolbar.
- 2 In the PropertyManager, click **Keep Visible** .
- 3 Zoom to the top right open profile of **Structural Member3**.
- 4 Under **Parameters**:
 - Select the profile face as shown for **Face** .
 - Set **Thickness**  to 3.
- 5 Under **Offset**:
 - Select **Chamfer corners**.
 - Set **Thickness Ratio**  to 0.5.
 - Set **Chamfer Distance**  to 3.
- 6 Click **OK** .
- 7 Repeat steps 3 through 6 to cap the remaining 3 open profiles.
- 8 Click **Cancel**  to exit the **End Caps** PropertyManager.



Select profile face



End cap with chamfer preview

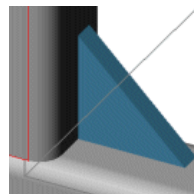


Gussets

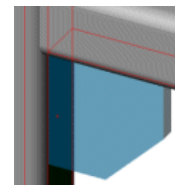
Gussets include **Triangular** or **Polygonal** profiles.

Triangular Profile. Set any of the following:

- **Profile.** Set the distance for each of the two edges adjacent to the structural member.
- **Thickness.** Add thickness to left, center or right side of the gusset.
- **Location.** Locate a profile relative to the left and right edges of the structural member. You can specify an offset distance from the edge to locate the gusset.



Triangular














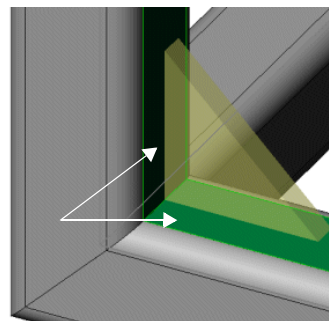
Polygonal

Polygonal Profile. Set all attributes listed for the **Triangular Profile**, plus the following:

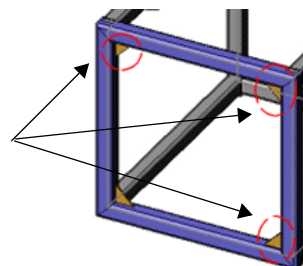
- **Profile.** Set the distance for the additional edges in the polygon.
- **Profile Angle.** Set the angle between the two edges in the polygon that are adjacent to the structural members.

To add gussets:

- 1 Click **Gusset**  on the Weldments toolbar.
- 2 In the PropertyManager, click **Keep Visible** .
- 3 Zoom to the front bottom left corner of **Structural Member1**.
- 4 Under **Supporting Faces** , select the two faces, for **Select Faces**, as shown.
- 5 Under **Profile**:
 - a) Click **Triangular Profile** .
 - b) Set **Profile Distance1**  **d1:** and **Profile Distance2**  **d2:** to **50**.
 - c) Click **Inner Side** , and set the **Gusset Thickness**  to **5**.
- 6 Under **Parameters**, click **Profile Locates at Mid Point** .
- 7 Click **OK** .
- 8 Repeat steps 4 through 7 to add a gusset to the other three corners of **Structural Member1**, as shown.
- 9 Click **Cancel**  to exit the PropertyManager.








You can add gussets only between planar faces.

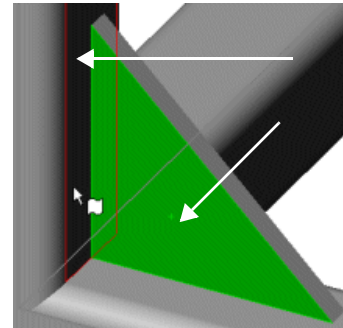


Fillet Beads



You can add full length, intermittent, or staggered fillet weld beads to your model. You can add the fillet weld bead on the **Arrow Side** or on the **Other Side**.

To add fillet weld beads:




- 1 Click **Weld Bead**  on the Weldment toolbar.
- 2 Zoom to the bottom left corner of **Structural Member1**.
- 3 In the PropertyManager, click **Keep Visible** .
- 4 Under **Arrow Side**:
 - a) Select **Full length** for **Bead type**.
 - b) Under **Fillet size**, set **Bead size**  to **5**.
 - c) Select the gusset face, as shown for **Face Set1** .
 - d) Click **Face Set2** , and select the flat, structural member face, as shown.

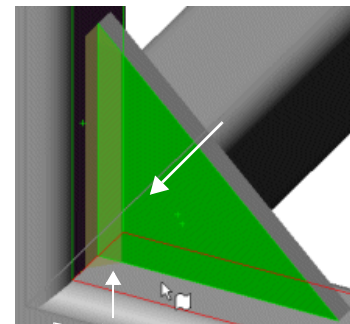


Arrow Side selections



The system assigns the **Intersecting edges** based on your selections for **Face Set1**  and **Face Set2** .

A fillet weld bead preview appears along the edges between the gusset and the structural member.



- 5 Select **Other Side**:
 - a) Use the same **Bead Type** and **Fillet size**  as in **Arrow Side**.
 - b) Select the gusset face, as shown for **Face Set1** .
 - c) Click **Face Set2** , and select the flat, structural member face, as shown.

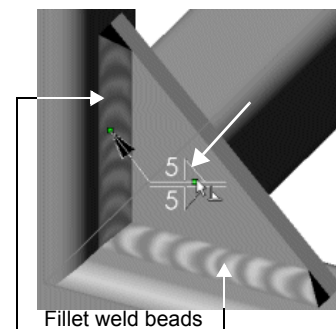


Other Side selections

The system assigns the **Intersecting edges** based on your selections for **Face Set1**  and **Face Set2** .

A fillet weld bead preview appears along the edges between the gusset and the structural member.

- 6 Click **OK** .
- The annotation appears, as shown.
- 7 Repeat steps 4 through 6 to apply fillet weld beads to the remaining three gussets.
- 8 Click **Cancel**  to exit the PropertyManager
- 9 Save the model.



Fillet weld beads

Sub-Weldments

You create sub-weldments to break a weldment into sub-weldments.


To create a sub-weldment:

- 1 In the FeatureManager design tree, click **Cut-List-Item1(12)**, press **Shift** and select all structural members.
- 2 Right-click and select **Create sub-weldment**.
This creates **Sub-weldment1(12)**.
You can optionally save the sub-weldment to a separate file.
- 3 Right-click **Sub-weldment1(12)** and select **Save to File**.
- 4 The **Save As** dialog box appears. You can:
 - Create a new part file that includes the 12 structural members under **Sub-weldment1(12)**.
 - Add **Sub-weldment1** to the part name.
 - Rename the part.
 - Save the referenced documents.
- 5 Save as **weldment_square-Sub-weldment1.sldprt**.

Cut Lists

You can create a weldment cut list that you can display on the drawing sheet. See **Weldment Cut Lists** on page 6-15 for more details on drawings.

To create cut lists:

- 1 In the FeatureManager design tree, click **Solid Bodies** , press **Shift** and select **Structural member1[1]** through **Structural Member3[4]**.
- 2 Right-click and select **Create cut list item**.
This creates **Cut-List-Item1** with all the structural members in a sub-folder.
- 3 Right-click and select **Create cut list item**.
This creates **Cut-List-Item2(1)** with only the trim-extend item.
- 4 Repeat step 3 for the gusset features.
- 5 Save the model.

Custom Properties

In a weldment feature, the custom properties on the cut list and sub-weldments represent a different list from the custom properties that are stored at the document level. In a weldment feature, you can manipulate the following custom properties' attributes:

- **List maintenance.** Locate the custom properties' list in the following location:
lang<language>\weldments\weldmentproperties.txt.
- **Dimensions.** Embed dimensions in the values of custom properties.
- **Mass property parameters.** Embed the mass property parameters of a body in the values of its custom properties.

Custom properties are required to generate a cut list with associative balloons for a multibody part (see **Weldment Cut Lists** on page 6-15).

To add custom properties:

- 1 Right-click **Cut-List-Item1(12)** and select **Properties**.
- 2 In the **Cut-List Item1 Custom Properties** dialog box:
 - Type **square tube iso** for **Name**.
 - Type **30 x 30 x 2.6** for **Value**.
- 3 Click **OK**.



You no longer need to click **Add** to save a custom property. When you click **OK**, the custom property is added (see **Custom Properties** on page 5-3).

Drawings of Weldments




Drawings of weldments support:

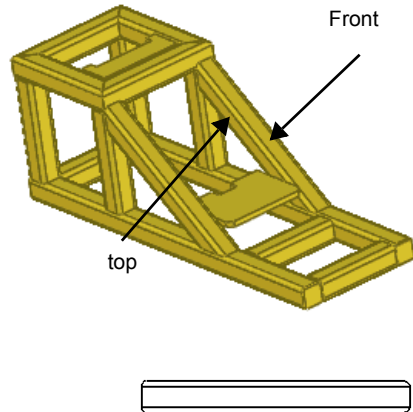
- Views of individual bodies of weldment parts
- Balloons
- Auto Balloon (see **AutoBalloons** on page 8-8)
- Weldment cut lists
- Alternating crosshatch for section views

Views of Single Bodies


You can create views of single bodies from multibody parts, including weldments.


To create a drawing view of a single weldment body:

- 1 Open <weldments\weldment_cut_list.sldprt> and <weldments\weldment_cut_list.slddrw> and tile the windows.
- 2 In the drawing document, click **Relative View**  on the Drawing toolbar.
- 3 In the part document, select the face labeled **Front** for the **First orientation** in the **Relative View** PropertyManager.
- 4 Under **Second orientation**, select **Top** from the list, then in the graphics area select the face labeled **Top** (rotate the model if necessary).
- 5 Click **OK** , then place the view at lower left of the drawing sheet and set the **Scale** to **1:5**.
- 6 Click **OK**  and leave the documents open for the next example.



Weldment Cut Lists

Weldment Cut List  is available on the Annotations toolbar and on the **Insert, Tables** menu. The cut list updates when the model or values of custom properties change.

In weldment part documents, you specify the bodies to include in a cut list item. You can group similar bodies, and sub-weldments, into one cut list item. Cut list items appear in the FeatureManager design tree under **Solid Bodies** .




You can save weldment cut list templates. Standard cut list templates are stored in the same folder as Bill of Materials templates.

All configurations reference the same cut list for creating balloons. Only top level items (bodies and sub-weldments) are listed in the cut list. You can split the cut list, and you can also sort the cut list in ascending or descending order.

Balloon numbers remain associative to the cut list, even if they are created in another view. Balloon numbers are the same for a body in a drawing that shows the entire weldment and for the same body in a view showing a single body.


If you insert a balloon into a drawing view of a weldment and the drawing does not contain a cut list, you are prompted on whether you want to create a cut list. If you delete a cut list, then all balloons related to the cut list change their item number to 1.

To insert a weldment cut list:

- 1 In the part document opened in the previous example, in the FeatureManager design tree, expand **Solid Bodies** .
- Several **Cut-List** items are listed with the number of bodies in the item in parentheses.
- 2 In the drawing document, select the drawing view of the complete part.
- 3 Click **Weldment Cut List**  on the Annotation toolbar.
- 4 In the PropertyManager, click **OK** , then click in the graphics area to place the upper-left corner of the cut list.

ITEM NO.	QTY.
1	2
2	4
3	3
4	2
5	4
6	4
7	1
8	1



To add custom properties to the cut list:

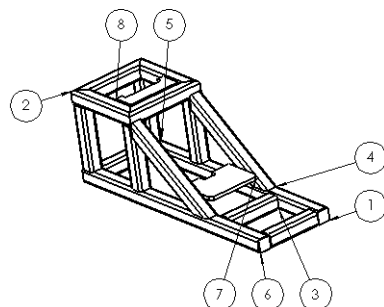
- 1 In the part document, in the FeatureManager design tree, right-click **Cut-List1** and select **Properties**.
- 2 Under **Name**, select **PartNo**, type **SM5556** for **Value**, click **Add**, then click **OK**.
- 3 Repeat steps 1 and 2 for **Cut-List2** and **Cut-List3**, entering values of **SM5557** and **SM5558**.
- 4 In the drawing document, right-click in the right column of the cut list and select **Insert, Column Right**.
- 5 In the PropertyManager, select **Cut List Item Property** and type **PART NO** for **Title**. The property selected is **PartNo**.
- 6 Click **OK** .
- 7 Drag the right border of the new column to size the column to the data.

ITEM NO.	QTY.	PART NO
1	2	SM5556
2	4	SM5557
3	3	SM5558
4	2	
5	4	
6	4	
7	1	
8	1	

See **Tables** on page 8-9

To add balloons to the drawing view:

- 1 In the drawing document, select the drawing view of the complete part.
- 2 Click **AutoBalloon**  on the Annotation toolbar.
- 3 In the **AutoBalloon** PropertyManager, select a **Layout** (the square layout is shown) and click **OK** .





Assemblies

This chapter describes enhancements to assemblies in the following areas:


- ☐ Insert components
- ☐ Move and rotate components
- ☐ Mates
- ☐ Assembly performance
- ☐ User interface
- ☐ Component patterns
- ☐ General enhancements

Insert Components into Assembly



Enhancements to inserting a component into an assembly include:

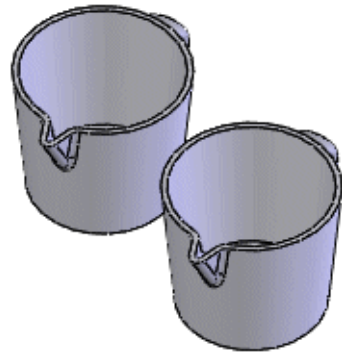
- **New Insert Component PropertyManager.** Click **Insert Components**  on the Assembly toolbar, or click **Insert, Component, Existing Part/Assembly**.
- **From file or an open document.** From the PropertyManager, you can insert an active document (open in the assembly or in another window) or click **Browse** to open a document.
- **Multiple components.** Click **Keep Visible**  in the PropertyManager to insert multiple components.
- **Active in a new assembly.** Select **Start command when creating new assembly** in the PropertyManager to open this PropertyManager when you create a new assembly.

To insert components:



- 1 Open **measuring_cup.sldprt**.
- 2 Click **Make Assembly from Part/Assembly**  on the Standard toolbar. See **New Assembly** on page 7-13.
- 3 If necessary, select an assembly template in the **New SolidWorks Document** dialog box and click **OK**.

An assembly opens with the **Insert Component** PropertyManager open. Note that **measuring_cup** is in the **Open documents** list.




- 4 Click **Keep Visible**  to insert multiple components.
- 5 Click in the graphics area to add the first instance, click again to add a second instance, then click **OK** .





Move and Rotate Components

You can move and rotate one or more components anytime in an assembly, without the need for a dialog box or PropertyManager. You can drag a component in the graphics area to move it without clicking **Move Component**  or **Rotate Component**  first.

You can also move or rotate in a specific direction. Right-click a component and select **Move with Triad**. A triad appears that offers the following move and rotate options:

Item to Click	Left-drag	Right-drag	Right-click
Arm 	Moves along axis.	Rotates about axis.	N/A
Wing 	Moves along plane.	N/A	N/A
Center Sphere 	Drag to a linear or planar entity to align the triad with the entity.	N/A	Pop-up dialogs appear: <ul style="list-style-type: none">• Moves to specific XYZ coordinate.• Moves by a specific amount.• Aligns the triad's axes to the component's or assembly's axes.• Rotates about axis by a specific amount.




The **Move Component**  and **Rotate Component**  tools are still available if you want to use the PropertyManager. To use Collision Detection, Dynamic Clearance, or Physical Dynamics, you must use the PropertyManager.

To move and rotate a sub-assembly:

- 1 Open **\\wise\\wise.sldasm**.

- 2 Click the handle sub-assembly in the graphics area and drag to the left.

The sub-assembly moves. Note you did not have to click **Move Component** .

- 3 Right-click the connector component and select **Move with Triad**.

A triad appears on the component.

- 4 Right-click the arm of the triad, hold down the right mouse button and drag the sub-assembly.

The sub-assembly rotates about the selected arm.

- 5 Left-click the arm of the triad, hold down the left mouse button and drag the sub-assembly.

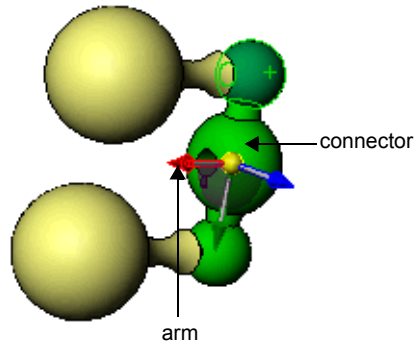
The sub-assembly moves along the selected arm.

- 6 Right-click the center sphere of the triad and select **Show Translate Delta XYZ Box**.

- 7 Type **3** for **DeltaY** and click **OK**.

The sub-assembly moves 3 inches along its Y axis.

- 8 Close the assembly without saving.

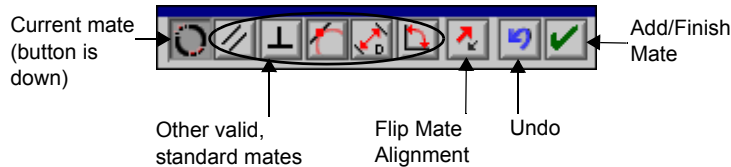


Mates


Mate Pop-up Toolbar


The **Mate** pop-up toolbar minimizes your interaction with the **Mate** PropertyManager for common mates and functions. For example, you can add a distance mate, change the distance value, flip the dimension direction, and finish the mate using the **Mate** pop-up toolbar.

The following shows the **Mate** pop-up toolbar after you select two circular faces:




To use the Mate pop-up toolbar:

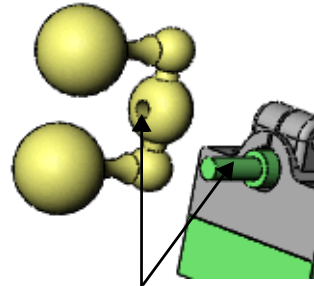
- 1 Open `\vise\vise.sldasm`.
- 2 Click **Mate**  on the Assembly toolbar.
- 3 Select the two circular faces shown.

The **Mate** pop-up toolbar appears with **Concentric**  selected. Also, the handle moves into position in the graphics area to preview the mate.

- 4 Click **Add/Finish Mate**  on the pop-up toolbar to add the mate.

A concentric mate appears under **Mates** in the PropertyManager. See **Mates Box** on page 7-6.

- 5 Click **OK**  in the PropertyManager to finish adding mates.
- 6 Save the assembly to use in another section.



Mate PropertyManager

Standard Mates

Angle, **Coincident**, **Concentric**, **Distance**, **Parallel**, **Perpendicular**, and **Tangent** mates are under **Standard Mates**. All the mate types are always shown in the PropertyManager, but only the mates that are applicable to the current selections are available.





Mates under **Standard Mates** also appear on the **Mate** pop-up toolbar. Mates under **Advanced Mates** appear only in the **Mate** PropertyManager.

Advanced Mates

Cam, **Gear**, **Limit**, and **Symmetric** mates are under **Advanced Mates**. See **Gear Mates** on page 7-8 and **Limit Mates** on page 7-9.

Mate Alignment

There are now buttons to set the aligned  and anti-aligned  property.

Mates Box

The **Mates** box contains all the mates added while the PropertyManager is open or all of the mates that you are editing. When there are multiple mates in the **Mates** box, you can select one to edit that mate.



To edit more than one mate at a time, select multiple mates in the FeatureManager design tree, then right-click and select **Edit Feature**. All the mates appear in the **Mates** box.

Options

The new options in the PropertyManager are:

- **Add to new folder.** When selected, new mates appear in a folder in the mategroup in the FeatureManager design tree. When cleared, new mates appear in the mategroup.
- **Show pop-up toolbar.** When selected, the **Mate** pop-up toolbar appears when you add standard mates. When cleared, you add standard mates in the PropertyManager.
- **Show preview.** When selected, a preview of a mate occurs when you make enough selections for a valid mate.
- **Use for positioning only.** When selected, components move to the position defined by the mate, but a mate is not added to the FeatureManager design tree. A mate appears in the **Mates** box so you can edit and position the components, but nothing appears in the FeatureManager design tree when you close the **Mate** PropertyManager.



This check box is an alternative to adding many mates, then later deleting those mates in the FeatureManager design tree.

Conical Mates

SolidWorks 2004 includes new mate combinations for conical faces:

- Angle - cone, cylinder, extrusion, and line
- Coincident - circular edge or point
- Concentric - circular edge
- Parallel - cone, cylinder, extrusion, and line
- Perpendicular - cone, cylinder, extrusion, and line
- Tangent - plane






An extrusion refers to a single face of an extruded solid or surface feature. The direction of extrusion is used in the mate.

Gear Mates

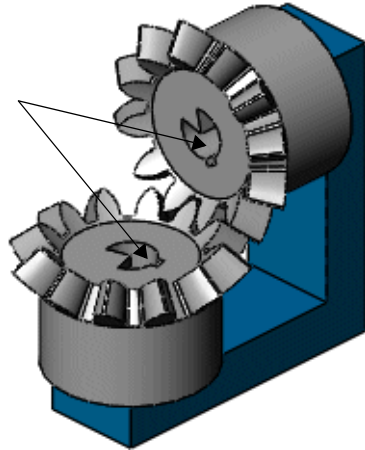
Gear mates force two components to rotate relative to one another about selected axes. Valid selections for the axis of rotation for gear mates include cylindrical and conical faces, axes, and linear edges. Other notes include:

- You can set the gear ratio between the two components.
- You can flip the direction of rotation so one component moves clockwise and the other moves counter-clockwise.
- Similar to other mate types, gear mates do not prevent interference or collisions between components. To prevent interference, use Collision Detection or Interference Detection.

To add a gear mate:

- 1 Open `\gear_mates\gears.sldasm`.
- 2 Click **Mate**  on the Assembly toolbar.
- 3 Under **Advanced Mates**, click **Gear** .
- 4 Select the two circular faces shown.
- 5 Click **OK**  twice to add the mate and close the PropertyManager.
- 6 Drag a gear in the graphics area.







The two gears rotate with respect to one another.

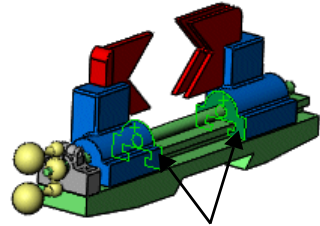


Limit Mates


Limit mates allow components to move within a range of values for distance and angle mates. Limit mates are located under **Advanced Mates** in the **Mate** PropertyManager. You specify a starting **Distance** or **Angle** as well as a **Maximum Variation** and **Minimum Variation**.

To add a limit mate:

- 1 Open `\\wise\\wise.sldasm`.
- 2 Click **Mate**  on the Assembly toolbar.
- 3 Select the two faces shown.
The components move together as a **Coincident** mate is inferred.
- 4 Under **Advanced Mates**, click **Distance** .
The components separate to the specified distance.
- 5 Type **4** for the **Distance** , **8** for the **Maximum Variation** , and **0** for the **Minimum Variation** .
- 6 Select **Flip Dimension** if the components move to an intersecting position.
- 7 Click **OK**  twice to add the mate and close the PropertyManager.
- 8 Drag one of the components.
Both components move within the limits specified by the mate.




SmartMates

The **SmartMates**  tool has been removed from the Assembly toolbar because of the **Mate** pop-up toolbar but the functionality to quickly add a mate remains as follows:

- Hold down **Alt** and drag a component by the entity that you want to mate. Drop the component on an entity that you want to mate to. The **Mate** pop-up toolbar appears and you can add the mate.



You can do this whether or not the **Mate** PropertyManager is open.

- The **Move Component** PropertyManager keeps its **SmartMates**  button and the functionality to add a SmartMate is unchanged. The **Mate** pop-up toolbar appears to confirm the mate type and alignment.

Mate Diagnostics

Enhancements to **Mate Diagnostics** include:

- **Component names.** The names of the mated components appear in the **Diagnostics** PropertyManager and in a tooltip when you hover over the mate.
- **Shortcut menu.** You can right-click mates in the PropertyManager to suppress or edit them. You can also toggle the alignment on some mates via the shortcut menu, depending on the error type.
- **Multiple sets of mates.** More than one set of mates with errors can appear in the PropertyManager at one time. Expand **Subset** under **Analyze Problem** to see each set of mates.
- **Transparent components.** Components that are unrelated to the current subset become transparent to help you identify the problematic areas.
- **Font improvement.** Mates that appear under both **Analyze Problem** and **Not Satisfied Mates** are in bold.

To use mate diagnostics:

- 1 Open `\mate_diagnostics\mates.sldasm`.
- 2 Click **Close** in the **Mates - Rebuild Errors** dialog box.
- 3 Right-click **Mates** in the FeatureManager design tree and select **Mate Diagnostics**.
- 4 In the PropertyManager, under **Analyze Problem**, click **Diagnose**.


Note that one subset of mate problems is found. The three other bolts and the second base become transparent because they are unrelated to this subset.

- 5 Expand **Subset 1**.

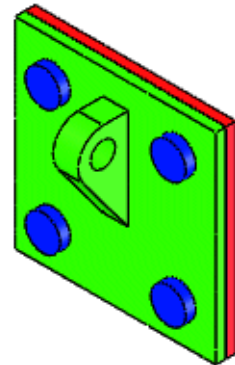
Note that **Concentric1** is in bold because it is under both **Analyze Problem** and **Not Satisfied Mates**.

- 6 Right-click **Concentric1** and select **Toggle alignment**.

The concentric mate's alignment is fixed and the PropertyManager reports that the **Mates are now satisfied**. Also, all components are transparent because there are no more mate errors.


- 7 Click **OK** .

Keep this assembly open for the next section.




Mate References


Enhancements to mate references include:

- **New access to PropertyManager.** Click **Mate Reference**  on the Reference Geometry toolbar, or click **Insert, Reference Geometry, Mate Reference** to access the PropertyManager.
- **More ways to add mates.** Now you can add mates via mate references if you hold down **Alt** and drag one component in an assembly and drop it over a valid mate reference partner. Additionally, when the **SmartMates** PropertyManager is open, the software adds mates via mate references before it adds geometric SmartMates.
- **Capture mate references.** You can use any existing mate on a component to define a mate reference for the component.

To capture a mate reference from an existing mate:

- 1 Open `\mate_diagnostics\mates.sldasm` if it is not already open.
- 2 Edit **bolt<4>** in the context of the assembly.
- 3 Click **Mate Reference**  on the Reference Geometry toolbar.
- 4 In the PropertyManager, under **References to Capture**, select **Edge of bolt-4** or **Coincident6**.

The entity, mate, and alignment of the mate appear under **Primary Reference Entity**.

- 5 Type a name under **Reference Name** and click **OK**  to add the mate reference to the component.

Assembly Performance

Lightweight Sub-assemblies

You can now open a sub-assembly as lightweight. In earlier versions, only part documents opened as lightweight in assemblies. Lightweight sub-assemblies behave similarly to lightweight parts.


Memory Allocation

A Microsoft Windows NT-based operating system allows a maximum of 4GB of RAM. By default, 2GB is allocated to the operating system and 2GB is allocated to the applications. You can now allocate 3GB of RAM to the SolidWorks application by decreasing the amount allocated to the operating system. The ability to increase the amount of RAM allocated to the SolidWorks application might benefit users with very large assemblies.

For detailed instructions, see:

- <http://support.microsoft.com/default.aspx?scid=kb;EN-US;q171793>
- http://msdn.microsoft.com/library/default.asp?url=/library/en-us/memory/base/4gt_ram_tuning.asp


Pointers in Lightweight Mode

The pointer for lightweight mode now indicates if it is over a face, edge, or vertex. For example, the pointer changes to  when over a face of a lightweight component.

User Interface

New Assembly

You can start a new assembly from a part or assembly document.

- 1 Click **Make Assembly from Part/Assembly**  on the Standard toolbar, or click **File, Make Assembly from Part**, or click **File, Make Assembly from Assembly**.
- 2 If necessary, select an assembly template in the **New SolidWorks Document** dialog box and click **OK**.



If you last used this dialog box in **Advanced** mode, you must select a template. If you last used this dialog box in **Novice** mode, a template is chosen for you. See **New SolidWorks Documents** on page 2-15.

An assembly opens with the **Insert Component** PropertyManager open.

- 3 Click in the graphics area to insert the component.

Assembly Toolbar

New Tools

The following buttons have been added:



Change Transparency

Toggles the transparency of the selected component between 0% and 75%.



Insert Components

Adds a part or sub-assembly to an assembly.



Interference Detection

Detects interference between components.



New Part

Adds a new part to an assembly.



New Assembly

Adds a new sub-assembly to an assembly.

Updated Tools

The images on the following tools are new. All previous functionality is retained.



Edit Component



Explode Line Sketch



Exploded View



Move Component




Replace Mate Entities




Rotate Component

Removed Tool

The **SmartMates**  tool has been removed. See **SmartMates** on page 7-9 for the replacement functionality.


Assembly Features

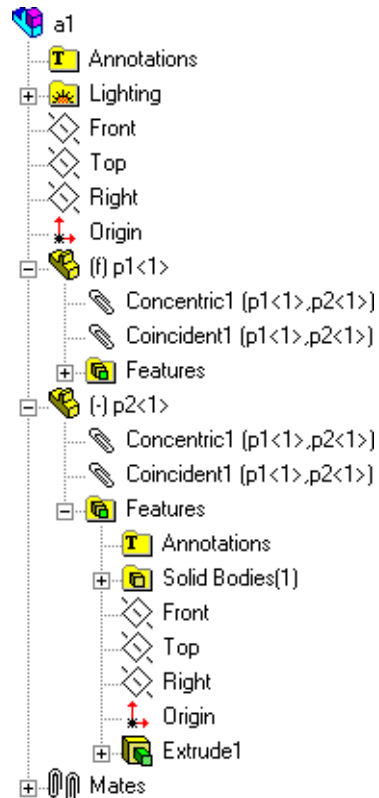
Enhancements to assembly features include:

- **Post-selection.** You can now select the following features without pre-selecting a sketch, a face, or a plane: Extruded Cut, Revolved Cut, Simple Hole. See **Post Selection** on page 4-4.
- **Revolved Cut** . This tool on the Features toolbar is now available in assemblies.

FeatureManager Design Tree

Enhancements to the display of the FeatureManager design tree using the **View Dependencies** tool include:

- **New menu item.** Right-click the top-level icon in the FeatureManager design tree and select **Tree Display, View Mates and Dependencies**.
- **New folder.** A **Features**  folder appears under each part in the assembly. This folder contains planes, features, and other items unique to the part. This simplifies the view by hiding many items by default, as shown for **p1** at right. You can open the folder to edit the features or mate to the planes, as shown for **p2** at right.



PropertyManagers

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

- Cavity
- Component Pattern
- Interference Detection



The menu item for the cavity feature is now **Insert, Molds, Cavity**. See **Mold Core and Cavity Functions** on page 4-12.

Component Patterns

Dissolve

You can dissolve a component pattern to make its components independent. One benefit is that you can individually move and rotate components that were in the pattern. One drawback is that you cannot hide and suppress all the components in the pattern because the pattern feature is gone.

Equal Spacing

Equal spacing is available for circular patterns of components.

Menu Items

Linear, circular, and feature driven component patterns are now available as separate menu items. Click **Insert, Component Pattern** and select from the menu.

PropertyManager

You create component patterns via the **Linear Pattern**, **Circular Pattern**, or **Feature Driven** PropertyManagers. All previous dialog box functionality is retained in the PropertyManagers.

General Enhancements

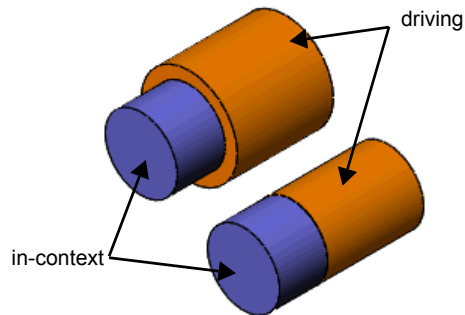
Configurations and In-Context Components

You can have more than one instance of an in-context component that is geometrically different in an assembly. In earlier versions, all instances were the same regardless of the component's configuration or the geometry of the driving component.

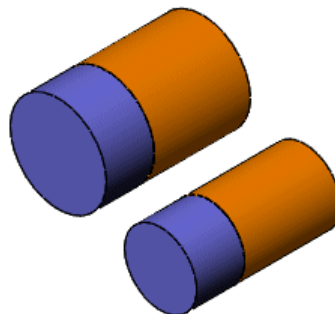
For example:

- There are two components in an assembly.
- The driving component has two configurations with different diameters.
- The in-context component is created in the context of an assembly. There is a relation to keep it the same diameter as the driving component. There are two configurations of the in-context component, one for each size of the driving component.
- There are two instances of this assembly as sub-assemblies in a main assembly.

In earlier versions, the in-context component was incorrect in the main assembly because all instances were the same size regardless of the driving component.



In SolidWorks 2004, the in-context component is the correct size because it matches the different sizes of the driving component.



Component Previews

To help you place a component, you now see a preview of a component when you add a component using any of the following methods:

- Drag and drop from Windows Explorer
- Drag and drop from the Feature Palette window
- Drag and drop from a part or sub-assembly window
- Click **Insert, Component, Existing Part/Assembly**
- Add a component pattern

Print and Copy References

The **Search Results** dialog box that you access by clicking **File, Find References** has been enhanced as follows:

- **Print.** Click this button to print the contents of the dialog box.
- **Copy list.** Click this button to copy the contents of the dialog box to the Windows clipboard so you can paste the list into a text editor.


Drawings and Detailing

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

- ☐ Drawing views
- ☐ Detached drawings
- ☐ Lightweight drawings
- ☐ Dimensions marked for drawings
- ☐ Annotations
- ☐ AutoBalloons
- ☐ Tables
- ☐ Options

Drawing Views

Model View

Named View is now called **Model View** . The **Model View** PropertyManager appears when you open a new drawing. You can select a model from a list of open documents or browse for a file.

To prevent PropertyManager from appearing automatically, under **Options** in the PropertyManager, clear **Start command when creating new drawing**.

When you place the model view in the graphics area, the view is still selected and the **Projected View** PropertyManager opens automatically.

Display Style is now available in the **Model View** PropertyManager. Choose a view:


- **Wireframe**
- **Hidden Lines Visible**
- **Hidden Lines Removed**
- **Shaded with Edges**
- **Shaded**

You can also select:

- **High Quality**
- **Draft Quality** (previously called **Fast HLR/HLV**)

Drawing documents open in **High Quality**, except for drawings in **Large Assembly Mode**, which open in **Draft Quality**.

New Drawings

You can create a new drawing from a saved part or assembly document. Click **Make Drawing from Part/Assembly**  on the Standard toolbar. A new drawing document appears with the **Model View** PropertyManager active even if you have previously cleared **Start command when creating new drawing**.

Drawing Sheet Formats

In previous releases, when you opened a new drawing, the **Sheet Format to Use** dialog box appeared. When you added a sheet to a drawing or accessed **Properties** for a drawing sheet, the **Sheet Setup** dialog box appeared.

In SolidWorks 2004, both dialog boxes are reorganized and include a **Preview** window. **Sheet Format to Use** is now called **Sheet Format/Size**, and **Sheet Setup** is called **Sheet Properties**.

When you open a new drawing, add a new sheet, or edit the sheet properties, you can display the sheet format or use the size and orientation of a standard sheet without displaying the format.

Colors in Drawings

You can specify the colors of the drawing sheet properties in **Tools, Options, System Options, Colors**:

- **Drawings, Paper Color**
- **Drawings, Background**
- **Drawings, Sheet Border** (previously **Paper Border**)
- **Drawings, View Border**
- **Drawings, Locked Focus**

If you specify a gradient background, the gradient color is used in the drawing sheet background. The dynamic drawing view border activation is a thinner, dashed font.

Moving Drawing Views

Child views move relative to parent views based on the alignment. Previously, all views moved in a fixed position relative to the parent view. For the previous behavior, press **Alt** while you move the views.

View Scales

You can choose to use the parent view scale for dependent views such as projected views, auxiliary views, and so on. The choices for parent scale, sheet scale, or custom scale appear in the drawing view PropertyManagers.

Crosshatch

Section views of multibody parts (including weldments) alternate crosshatch between the different bodies.

Crosshatch identifiers are now language-independent. If you save a drawing in one language and open it in a different language, the crosshatch patterns are recognized correctly.

The **Remove crosshatch** check box is now named **Material crosshatch**.

An **Auto hatching** option has been added to the **Section View** PropertyManager.

Dimension Type in Drawing Views

The selection of **Dimension Type (Projected or True)** now appears in the drawing view PropertyManagers. Previously the type of dimension was shown in the **Drawing View Properties** dialog box.

Section Views

Creation of section views in assembly drawings is now faster.

Relative Views

The procedure for relative views has been revised. See **Views of Single Bodies** on page 6-15.

Detached Drawings

The term RapidDraft has been replaced by Detached. Detached drawings are those that do not need their associated models when opened.

Detached drawings can be in Draft Quality, but they cannot be Lightweight drawings.

In previous releases, RapidDraft drawings were created when opening a regular drawing. Now you save drawings as type **Drawing (*.slddrw)** or **Detached Drawing (*.slddrw)** in the **Save As** dialog box, so you can save Detached drawings as regular drawings and vice versa.

In the FeatureManager design tree, the icons for Detached drawings display a broken link:



Drawing icon for Detached drawing



Drawing view icon for Detached drawing with model not loaded

When you select **Display drawing view borders** in **Tools, Options, System Options, Drawings**, view borders for Detached drawings are dashed blue lines.

As with RapidDraft drawings, right-click a drawing view and select **Load Model** to load the model. If the model needs to be loaded, a message prompts you.

Lightweight Drawings

Lightweight drawings are analogous to lightweight assemblies. When a drawing is lightweight, only a subset of its model data is loaded in memory. The remaining model data is loaded as needed.

Performance of drawings of large assemblies is improved significantly with lightweight drawings. Loading a lightweight drawing is faster than loading the same drawing with fully resolved parts.

Lightweight drawings are efficient because the full model data is loaded only as it is needed. Only parts that you select, and parts that are affected by changes that you make in the current editing session, become fully resolved.

To load a drawing as lightweight, select **Lightweight** in the **Open** dialog box. **Set Lightweight to Resolved** and **Set Resolved to Lightweight** are available in the drawing sheet shortcut menu.

Lightweight drawings use **Draft Quality** views. You can specify new views to be **Draft Quality** in **Tools, Options, System Options, Display Style**. If you insert a view into a lightweight drawing as **High Quality**, you are prompted to convert the view to **Draft Quality**.

Lightweight drawings cannot be Detached drawings.

A feather is displayed to indicate lightweight parts and views:



Pointer for a lightweight part in a drawing view



Drawing view icon in the FeatureManager design tree for a lightweight view

With lightweight drawings, you can:

- Create all types of drawing views
- Attach annotations to models in views
- Specify edge properties
- Highlight and dynamically select edges and vertices
- Set drawings of sub-assemblies to lightweight or resolved

Dimensions Marked for Drawings

You can specify which dimensions in a part or assembly to insert in a drawing with **Insert Model Item** or auto insert. Right-click a dimension and select **Mark For Drawings**.

Dimensions are marked for drawings by default. Dimensions not marked for drawings appear in a different color. Specify the color for **Dimensions, Not Marked for Drawings** in **Tools, Options, System Options, Colors**.

To import only the marked dimensions into a drawing, in the **Insert Model Items** dialog box, select **Marked for drawings** under **Dimensions**.

To auto insert marked dimensions into new views, select **Dimensions marked for drawings** under **Auto insert on view creation** in **Tools, Options, Document Properties, Detailing**.

For more new functionality in dimensions, see **Dimensions** on page 3-2.

Annotations

When you select an annotation, the geometry to which the annotation is attached is now highlighted.

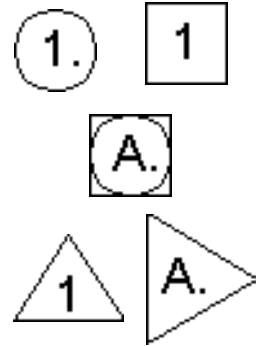
You can select dimensions and annotations by selecting their lines or leaders. Previously you had to select the dimension value or the annotation itself.

On-screen editing has been improved so that text, symbols, and dimensions appear in the editor as they appear on screen.

Hole Callouts now display the number of instances if the hole is part of a pattern.





New Flag symbol libraries are now available in the **Symbol** dialog box for Notes, Geometric Tolerancing, and so on. The library names begin with the word **Flag** and the symbols contain:

- A letter or number
- A period or no period
- A circle, a square, both a circle and square, a triangle, or a rotated triangle



PropertyManagers and Dialog Boxes

The following annotations are now handled in PropertyManagers rather than dialog boxes:

-  Cosmetic Thread
-  Datum Target
-  Datum Feature Symbol
-  Surface Finish

The **Weld Symbol** dialog boxes have been reorganized. Weld symbol sizes for ANSI standard reflect up-to-date ASME specifications. Weld symbols have been added for the JIS standard.

The **Cosmetic Thread** PropertyManager now has an option for **Thread texture**. See **Graphical Threads** on page 5-6.

The **Move/Copy** PropertyManager (now called **Move or Copy**), previously available only from sketch entity PropertyManagers, is now available in drawings (for annotations) as well as sketches from the **Sketch Tools** menu (see **Move or Copy** on page 3-4) and Sketch toolbar.

Favorites

Favorites have been added for the following annotations:

- Notes
- Geometric Tolerance Symbols
- Surface Finish Symbols
- Weld Symbols

Favorites for these annotations are similar to favorites for dimensions. Notes have two types of favorites:

- **With text.** If you type text in a note and save it as a favorite, the text is saved with the note properties. When you create a new note, select the favorite, and place the note in the graphics area, the note appears with the text. If you select text in a drawing and then select a favorite, the properties of the favorite are applied without changing the selected text.
- **Without text.** If you create a note without text and save it as a favorite, only the note properties are saved.

The file name extensions for the new favorites are:

Notes	.sldnotfvt
Geometric Tolerance Symbols	.sldgtolfvt
Surface Finish Symbols	.sldsffvt
Weld Symbols	.sldweldfvt

The extension for dimension favorites remains **.sldfvt**.

Blocks

Block definitions now keep layer information for the sketch entities, dimensions, and notes that make up the definition. These are used when you:

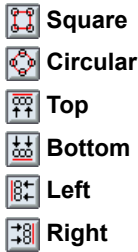
- Display block instances
- Explode block instances
- Paste blocks into new drawings
- Save and insert blocks from **.sldbldk** files

During **Insert Block** operations, blocks snap to sketch geometry and other block placement points.

AutoBalloons

AutoBalloon inserts a set of balloons for a drawing view or multiple views. Balloons are inserted in the appropriate views without duplicates.



Balloon layouts include:

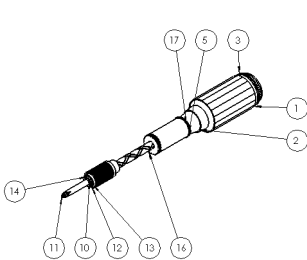
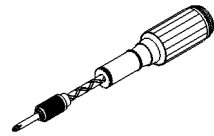


The balloons are placed outside the view boundary and leaders do not intersect. If the number of balloons does not fit at the view boundary, then the boundary size is increased. You can move a group of balloons or a single balloon manually.

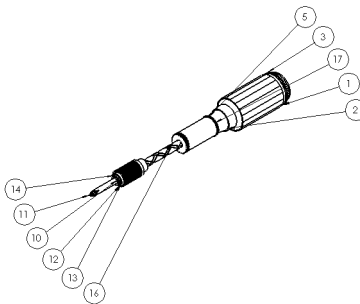
Specify the default AutoBalloon layout in **Tools, Options, Document Properties, Balloons**. In **Tools, Options, Document Properties, Detailing, Balloons** for **Auto insert on view creation**.

To insert balloons:

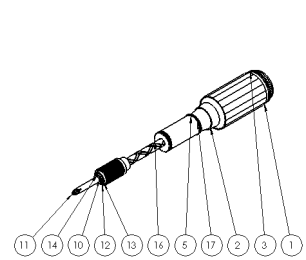
- 1 Open `\drawings\drw_auto_balloon.slddrw`.
- 2 Select the drawing view.
- 3 Click **AutoBalloon**  on the Annotation toolbar.
- 4 Under **Balloon Layout**, click the various layout styles to see their effect.
- 5 Click **OK**  to place the balloons.



Square



Circular



Bottom

Tables

Previously, Bill of Materials tables used Microsoft Excel. Now the SolidWorks software provides its own table functionality for:



Hole Table



Bill of Materials



Revision Table



Weldment Cut List (see **Drawings of Weldments** on page 6-14)

The tools appear on the Annotation toolbar and the new **Insert, Tables** menu.

If you edit the contents of tables, the changes remain after you update the drawing or rebuild the model. However, if you edit a value that is parametrically linked to a model, the link breaks.

Specify options for Hole Tables, Revision Tables, and Bills of Materials in **Tools, Options, Document Properties, Tables**. See **Options** on page 8-14.

Characteristics of the new tables include:

- Move a table by either holding down **Alt**, selecting anywhere in the table, and dragging, or by dragging the title bar
- Drag to resize row and column borders
- Zoom to selection
- Snap to horizontal and vertical lines and points in sheet formats

Selections in the table PropertyManager include:

- Headers and borders
- Column properties
- Cell text format
- Table format and table properties
- Layers

From the shortcut menu:

- Insert or delete columns or rows
- Merge or split cells
- Sort columns or rows in ascending or descending order

Each type of table has its own anchor point in the sheet format.

Standard table templates are available, and you can create and save custom templates. Custom templates contain headings and formatting, but not table contents. The PropertyManager remembers the last used template. File name extensions for the table templates are: **.sldholtbt** for hole tables, **.sldrevtbt** for revision tables, **.sldbomtbt** for Bills of Materials, and **.sldwldtbt** for weldment cut lists.

Hole Tables

Hole Tables measure the positions of selected holes from a specified origin datum. You can select a face to include all the holes on the face in the hole table.

The software labels each hole with a tag that corresponds to a row in the table. The columns are labeled **Tag**, **XLoc**, **YLoc**, and **Size**.




Non-circular holes such as rectangles and slots are recognized as holes. The table lists the X and Y coordinates of the entities, but you must enter the size.

In addition to general table properties, you can specify:

- Tags to be alphabetic or numeric
- Cells with the same tags to be combined
- Cells for holes of the same size to be combined
- Precision of hole locations

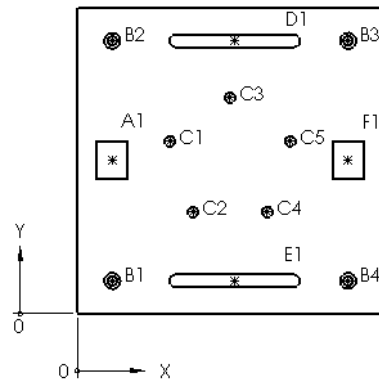
To create a hole table:



- 1 Open `\drawings\drw_hole_table.slddrw`.
- 2 Click **Hole Table**  on the Annotation toolbar.
- 3 Select the lower-left corner of the model face to specify the origin datum.

Vertex <1> appears in **Origin** under **Datum**.



You can select an X axis and Y axis rather than a vertex. You can also drag the origin datum into position.



- 4 Select the model face to select all holes on the face for the hole table.
Face <1> appears in **Edges/Faces** under **Holes**.
- 5 Click **OK**  and place the table in the graphics area.
Sizes for the rectangles (**A1** and **F1**) and slots (**D1** and **E1**) are not given.
- 6 Right-click in the table (anywhere except in the **Size** column) and select **Properties**.
- 7 In the PropertyManager, under **Scheme**, select **Combine same sizes** and click **OK** .
- 8 Drag the right border to size the column.

Tag	XLoc	YLoc	Size
A1	33	145.60	
B1	33.16	31.18	$\varnothing 9$ THRU ALL $\varnothing 15 \nabla 4.40$
B2	33.16	258.82	
B3	256.84	258.82	
B4	256.84	31.18	
C1	87.94	163.54	$\varnothing 10$ THRU
C2	109.73	96.46	
C3	145	205	
C4	180.27	96.46	
C5	202.06	163.54	
D1	149.27	258.82	
E1	149.27	31.18	
F1	257	145.60	

Bills of Materials



In addition to using the new SolidWorks table functionality, properties for a Bill of Materials have been moved from dialog boxes into the PropertyManager. You can specify:

- Template
- BOM type (**Top level only**, **Parts only**, or **Indented assemblies**)
- Configurations
- Whether to keep missing items
- Zero quantity display (dash, zero, or blank)
- Item numbers (value at which to start, whether to follow assembly order)
- Column properties (**Item Number**, **Part Number**, **Quantity**, or **User Defined**) and the column position
- BOM contents (reorder rows, group item numbers, view which components have balloons)

Tables built in previous releases using Excel are supported but can no longer be created.

You can now exclude assembly components from a Bill of Materials. In the assembly document, right-click the component and select **Properties, General**. In the **Component Properties** dialog box, select **Exclude from bill of materials**.

To insert a Bill of Materials:

- 1 Open `\drawings\drw_BOM.slddrw`.
- 2 Select the drawing view.
- 3 Click **Bill of Materials**  on the Annotation toolbar.
- 4 In the PropertyManager, under **Table Anchor**, select **Attach to anchor**.
- 5 Under **Configurations**, select all configurations, then click **OK** .

The Bill of Materials appears with its upper-left corner attached to the anchor at the top-left corner of the drawing sheet.


- 6 Right-click in the table and select **Zoom/Pan/Rotate, Zoom to Selection**.

ITEM NO.	PART NUMBER	DESCRIPTION	C0/ QTY.	C1/ QTY.	C2/ QTY.
1	8112156	Handle	1	1	-
2	8112174	Handle-shoulder	1	1	-
3	8113199	End cap	1	1	-
4	9113155	Embedded bolt	1	1	1
5	8112992	Housing	1	1	1
6	8116170	Gear shaft screw	1	-	1
7	112-135	Crescent washer	1	1	1
8	112-139	Shaft screw	1	1	1
9	113-144	Shaft spring	1	1	1
10	8112188	Shaft	1	1	1
11	403-112	Phillips	1	-	-
12	8114175	Collar	1	1	1
13	Purchased	Clip	3	2	3
14	8112001-1	Turn gear	1	1	1
15	8112001-2	Turn gear	1	1	1
16	8115777	Switch cover nut	1	1	1
17	8115142	Switch casing	1	1	1
18	8117199	Turn selector plate	2	2	2
19	9581-12	Selector switch	1	1	1




The Bill of Materials lists quantities for each configuration.

To edit the Bill of Materials:

- 1 Right-click the table and select **Properties**.
- 2 In the PropertyManager, under **Item Numbers**, click **Do not change item numbers** .
- 3 Click **BOM Contents**.
- 4 On the **Contents** tab, select a row, click **Row down** several times, and click **Apply**.



Because you selected **Do not change item numbers**, the item number remains with the row.

- 5 Click **OK**.
- 6 To restore the Bill of Materials to its original order, click **Follow assembly order**  under **Item Numbers**.
- 7 Drag the borders to resize the columns.
- 8 Click outside the table and leave the drawing open for the next exercise.





Revision Tables

In addition to the functionality for all tables, for a Revision Table you can select:

- Revision symbol shapes (circle, square, triangle, or hexagon)
- Alphabetic or numeric sequence
- Column properties (**Zone**, **Revision**, **Description**, **Date**, **Approved**, or **Custom Properties**) and the column position
- **Add Revision**
- **Add Symbol**

Revision tables snap to horizontal lines, vertical lines, points, and vertices.

To insert a revision table:

- 1 With the drawing from the previous exercise open, click **Zoom to Fit**  on the View toolbar.
- 2 Click **Revision Table**  on the Annotation toolbar.
- 3 In the PropertyManager, under **Revision Symbol Shapes**, select **Triangle**  and click **OK** .


The table appears at the upper-right corner of the drawing sheet, attached to the **Revision Table Anchor**.


- 4 Right-click in the table and select **Revisions, Add Revision**.

A row is added with revision **A** and the current date.




The revision letter also appears under **REV** in the lower-right corner of the sheet format. This field always displays the latest revision.

- Click in the graphics area to place a revision symbol, then click **OK**  to close the PropertyManager.



REVISIONS				
ZONE	REV.	DESCRIPTION	DATE	APPROVED
	A		01/15/2003	

To edit the table:

- Right-click in the table and select **Properties**.
- Under **Options**, clear **Enable symbol when adding new revision**, then click **OK** .



Now the pointer is *not* ready to insert revision symbols when you add revisions to the table.

- Right-click in the table and select **Zoom/Pan/Rotate, Zoom to Selection**.
- Add two more rows to the revision table, then double-click **B** and change it to **BB**.
- Click **Tools, Options, Document Properties, Tables**.
- Under **Revision Table, Alpha/Numerical Control**, select **1, 2, 3...** (**Change all** is selected) and click **OK**.

The revision letters change to numbers, except for **BB**.



Automatic controls do not change items that you edit.

- Click **Tools, Options, Document Properties, Tables** again.
- Select **A, B, C...** and **Start from where user left**, then click **OK**.
- Add another revision to the table.

The previous numbering does not change, while the new revision starts at the next letter.

REVISIONS				
ZONE	REV.	DESCRIPTION	DATE	APPROVED
	1		01/15/2003	
	BB		02/15/2003	
	3		03/15/2003	
	D		05/15/2003	

Options

System Options

Drawings

Automatic scaling of 3 view drawings is now **Automatically scale new drawing views**.

Optimize drawing file size for shaded and draft quality views has been added for assembly drawings. Normally, display data for drawings containing shaded or draft quality views is stored with the drawing. When this option is selected, the display data is not stored with the drawing, so the drawing opens faster. When the drawing is loaded later, the display data is read from the assembly document.

Custom property used as Revision has been added. In the **Summary Information** dialog box (click **File, Properties**), **Revision** is one of the custom properties on the **Custom** tab. However, you can specify any custom property to be used for **Revision**. This property is used in Revision Tables and appears in drawing sheet formats (in the title block at the lower right, and, for D and E size drawing formats, in the right margin).

Display Style

Previously, the **Display Style** options were called **Default Display Type**.

The **Fast HLR/HLV** check box is replaced by **Display quality for new views**:

- **Draft quality** - default for **Large Assembly Mode**
- **High quality** - default for all other drawing documents

Large Assembly Mode

HLR edges when shaded has been removed.

Default display quality for new views (**Draft Quality** or **High Quality**) has been added.

Document Properties

Detailing

Under **Dimensioning standard**, an option for **Leading zeroes** has been added. You can choose to show or remove leading zeroes, or follow the standard selected for the drawing.

Under **Auto insert on view creation**, you can now designate for **Balloons** and **Dimensions marked for drawings** to be inserted into new drawing views.

Dimensions

Click **Precision**. You can now specify the precision for tolerance to be the same as for the dimension.

In the **Dimension PropertyManager**, you can specify the precision of individual dimensions to be different from the precision of the document, and the precision of the tolerance to be same or different from the precision of the dimension.

Balloons

Auto Balloon Layout has been added. You can choose square, circular, or top, bottom, left, or right as the layout for inserting auto balloons.

Tables

Options have been added for the following types of tables:

- **Hole Tables**. Display of the origin indicator, tag angle and offset from profile center, alphabetic or numeric control, and schemes for combining cells with the same tags or with holes of the same size.
- **Revision Tables**. Symbol shapes and alphabetic or numeric control.
- **Bill of Materials Tables**. Display of zero quantities and missing components.

View Labels

You can change the name, label display, scale, delimiter, and text display for Detail, Section, and Auxiliary views.

If you modify the labels in this options dialog box and later change to a different dimensioning standard, the labels do not update to the new standard.

Other Functionality

This chapter describes enhancements to the following applications:

- ☐ Application Programming Interfaces and macros
- ☐ Configurations
- ☐ COSMOSXpress
- ☐ Import/Export
- ☐ Reference Geometry
- ☐ SolidWorks Routing
- ☐ SolidWorks Task Scheduler

Application Programming Interfaces and Macros

Application Programming Interfaces

This release includes the following new APIs (Application Programming Interfaces). These APIs are in addition to the existing SolidWorks, PDMWorks, and Toolbox APIs.

- FeatureWorks API
- SolidWorks Utilities API

The major enhancements made to the SolidWorks API in SolidWorks 2004 are as follows. Click **Help, SolidWorks API and Add-Ins Help Topics**, and read the *SolidWorks 2004 API Release Notes* for details.

- **Old-style MFC (Microsoft Foundation Class) extension add-ins end-of-life.** This is the last release that old style MFC extension add-ins are supported. Use the ATL (Active Template Library) Object Wizard to rewrite existing add-ins.
- **Persistent reference IDs for selectable objects.** Applications can use persistent reference IDs to locate SolidWorks objects at runtime.
- **Support for .NET add-ins**
- **Project Wizards for Visual Studio .NET.** Available in the SolidWorks API SDK (Software Development Kit).
- **Macro feature support for:**
 - Configurable dimensions
 - User-defined macro feature icons
 - Creation of reference curve features
- **Support for BOM, hole, and revision tables and annotations**
- **Drawing view components.** Use the DrawingComponent interface to manipulate the drawing-related settings of components, without preselecting components in the drawing views.
- **Interface for selecting objects.** Use the **SelectData** interface to select features, components, and sketch segments in the context of a drawing view. Allows the SolidWorks software to determine which values to set.
- **Multiselect objects.** Select groups of objects at the same time instead of individually selecting each object.
- **Mate-related interfaces.** Allow better access to assembly mates:
 - Mate2
 - MateEntity2
 - MateInPlace
- **User-defined bitmap buttons on PropertyManager pages.** Define your own bitmap buttons for PropertyManager pages.

- **Pushpins for PropertyManager pages.** Add pushpins to PropertyManager pages.
- **Macro recorder.** Application and document user preferences are recorded by the Macro recorder.
- **New type library for SolidWorks constant enumerators.** The **swconst.h** and **swconst.bas** files have been merged into a common type library, **swconst.tlb**. Applications that used either **swconst.h** or **swconst.bas** should now reference **swconst.tlb** for the SolidWorks constant enumerations.
- **Third-party storage, SolidWorks template file, and SolidWorks part file.** Third-party storage is now copied from the SolidWorks template file to the SolidWorks part file. Therefore, the stream exists in the SolidWorks part document and holds the data that was in the document template. This can affect other applications if those applications assumed that the storage was empty for a new part.
- **Context-sensitive Help.** Use context-sensitive Help in the Microsoft Visual Basic Object Browser. Press **F1** to display the Help topic for the item selected in the **Classes** or **Members of** list.

Macros

After you edit a macro, click **Tools, Macro** to see a list of the most recently edited macros. This list contains up to the last nine macros you edited. Select a macro from the list to edit it.

Configurations

Design Tables

Column headers in design tables are now case-insensitive.

Previews

You no longer have to save a model in each configuration to see its preview in the PropertyManager.

COSMOSXpress

Materials

COSMOSXpress uses the material assigned to the part in **Tools, Options, Document Properties, Material Properties** by default. You can change the material assigned to the part from COSMOSXpress. To update the results after changing the material assigned to a part, you must re-run the analysis.



Assigning a material to a part from COSMOSWorks or PhotoWorks 2 does *not* update the material for COSMOSXpress.

Automatic Mesh Trials

COSMOSXpress automatically tries to mesh the model using 3 different element sizes before reporting a mesh failure. If meshing using the specified element size (E) fails, COSMOSXpress tries to mesh the part using an element size of $(0.80 * E)$ in the second trial, and $(0.64 * E)$ in the third trial.

Showing Areas Where Meshing Failed

If meshing fails, COSMOSXpress lists the faces and edges that caused the mesh failure and makes suggestions to mesh the part successfully. Select an edge or face from the list to highlight it in the graphics area.

Options Page Instead of Dialog Box

The **Options** dialog box in earlier releases of COSMOSXpress is replaced by the **Options** page for consistency.

Ctrl Key Not Required for Selecting Multiple Items

COSMOSXpress does not require pressing the **Ctrl** key to select multiple faces when applying restraints and loads.

Show Locations of Extreme Values on Plots

COSMOSXpress offers an option to show the locations of maximum and minimum values on result plots.

To show locations of extreme values:

- 1 Start COSMOSXpress.
- 2 On the **Welcome** tab, click **Options**.
The **Options** screen opens.
- 3 Select **Always show annotation for maximum and minimum in the result plots**.

Exporting COSMOSXpress Studies to COSMOSWorks

When opening a part with a COSMOSXpress study in COSMOSWorks, the study automatically appears as COSMOSXpress_Study in the COSMOSWorks Manager tree. You can view the results of the study directly.

Import/Export

Adobe Portable Document Format (PDF) Files

You can save SolidWorks drawing documents as Adobe® Portable Document Format (PDF) version 1.4 files using the PDF translator. The PDF files can be read using Acrobat® Reader® version 5.0 and above. In the **Save As** dialog box, select **Adobe Portable Document Format (*.pdf)** in **Save as type**.

The PDF translator supports bookmarks for drawing sheets and views. The SolidWorks document information, such as file name, title, subject, author, and version used is available when you view the document summary information in the PDF file.

DXF/DWG Files

Export


When you export SolidWorks drawings, the DXF™/DWG translator supports line thickness, hidden sketches, and auto-centerlines.

Hole Tables

The DXF/DWG translator supports the import and export of hole tables. See **Hole Tables** on page 8-10.

Import Diagnosis

The **Import Diagnosis** PropertyManager has a new design and improved functionality. Under **Gaps**, click **Diagnose** to search for gaps to fix.

Under **Faces**, the **Geometry** section (including the **Simplify Geometry** tool) has been removed. Click **Diagnose** to list faces that need simplification in the **Face selection**  list. Tooltips describe what is wrong with each face when you hover the pointer over items in the list.

With the new design, you can fix faces at any time. If you try to fix gaps but all the gaps cannot be healed, try fixing the faces to fix the gaps. You can now also fix surface bodies. Previously, you could only fix solid bodies.

To view and fix gaps:

- 1 Open **Gaps.x_t**.
- 2 In the FeatureManager design tree, right-click **Surface-Imported1** and select **Diagnosis**.
- 3 In the PropertyManager, under **Gaps**, click **Diagnose**.
The PropertyManager reports **4 gap(s) found**.
- 4 Click **Next** repeatedly to view each gap.


Each gap is highlighted and an arrow shows its location.

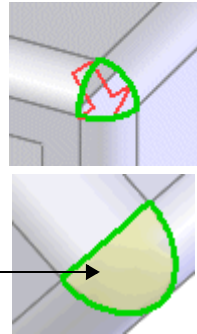
- 5 Select **Healing Preview** then click **Next**.

The model zooms to the next gap and displays a preview of the proposed fix.

- 6 Click **Accept**.


The gap is fixed in the graphics area and the PropertyManager updates to report **3 gap(s) found**.

- 7 Click **Next** to zoom to the next gap and the healing preview.
- 8 Click **Close All** to fix all gaps.
- 9 Click **OK** .



To diagnose and simplify or merge faces:

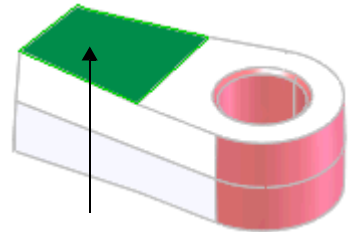
- 1 Open **Simplification.x_b**.
- 2 In the FeatureManager design tree, right-click **Imported1** and select **Diagnosis**.
- 3 In the PropertyManager, under **Faces**, click **Diagnose**.

One face that requires simplification highlights in the graphics area. **Face <1>** appears in the **Faces to be fixed/removed**  list.

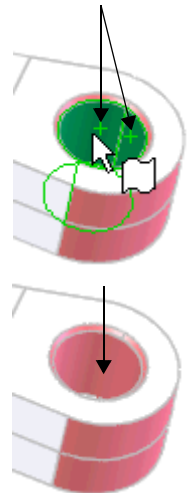
- 4 Move the pointer over **Face <1>** in the list.

The tooltip reports there is a **General geometry problem** with the face.

- 5 Click **Fix** to simplify the face.
- 6 Select the two faces to merge inside the cylinder.
Face <2> and **Face <3>** appear in the list.



- 7 Click **Fix** to merge the two faces.
- 8 Click **OK** .



Parasolid Files

The Parasolid® translator supports all Parasolid versions up to and including version 15.0.

Print 3D

You can contact rapid part and prototype vendors to request price quotes and place orders for the current part document through a web portal that is linked to the SolidWorks software. Print 3D saves time by automating the prototype request process, eliminating the need to save parts as STL files, FTP parts to vendors, and perform other operations.

Click **File**, **Print 3D** to access the web portal.

Pro/ENGINEER Files

Assembly Import Options

When you import the geometry of a Pro/ENGINEER® assembly file, you can choose to use BREP or knitting. Previously, knitting was the only option.

In the **Pro/ENGINEER to SolidWorks Converter** dialog box, under **Component Import Options**, select **BREP** or **Knitting**.

Import Improvements

The Pro/ENGINEER translator offers improved:

- Import of features (such as extrudes, holes, and cuts) and complex curved shapes (such as helical springs) contained in encrypted files.
- Treatment of reference planes so sketches are imported to the proper planes and redundant reference planes are not created.
- Resolution of complex features and geometry.

Import Options

You have the option to import material properties or sketch/curve entities. These options are useful if you want to import the geometry of large part or assembly files but not the face colors and other sketches. Previously material properties and sketch/curve entities were automatically imported.

In the **Pro/ENGINEER to SolidWorks Converter** dialog box, select **Import material properties** or **Import sketch/curve entities**.

Solid Edge Files

You can import the Parasolid information from Solid Edge® assembly files into SolidWorks assembly documents. In the **Open** dialog box, select **Solid Edge Assembly (*.asm)** in **Files of type**.

Reference Geometry

New PropertyManagers

The **Helix Curve**, **Coordinate System**, and **Reference Axis** dialog boxes have moved to the **Helix/Spiral**, **Coordinate System**, and **Axis** PropertyManagers. All previous dialog box functionality is retained in the PropertyManagers.



Reference Datum Points

You can create several types of reference datum points to use as construction objects:

- Center of arcs or faces
- Intersection of entities
- Projection on a face




You can also create multiple reference points on a curve and set the distance between the points.

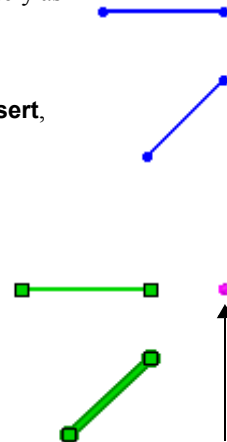
To create reference datum points:

- 1 Open a sketch and sketch two non-intersecting lines approximately as shown.
- 2 Exit the sketch.
- 3 Click **Point**  on the Reference Geometry toolbar, or click **Insert, Reference Geometry, Point**.
- 4 In the PropertyManager, click **Intersection** .
- 5 Select the two lines in the graphics area.

A preview of the reference point appears at the virtual intersection of the two lines.

- 6 Click **OK** .

The reference point changes to  in the graphics area followed by the point number, for example,  Point1. The point entity  **Point1** also appears in the FeatureManager design tree.



Sketch Plane References

When you delete a parent feature that has child features built from sketches, you have the option to keep the child features and have the sketch plane references dangle. You can dangle a sketch plane reference by editing the sketch plane definition and removing the plane definition.

To create dangling sketch plane references:



- 1 Open **ParentChild.sldprt**.

Chamfer1 is the parent feature and **Extrude2** is its child.

- 2 Select **Chamfer1** in the FeatureManager design tree and press **Delete**.
- 3 In the dialog box, clear **Also delete all child features**, then click **Yes**.

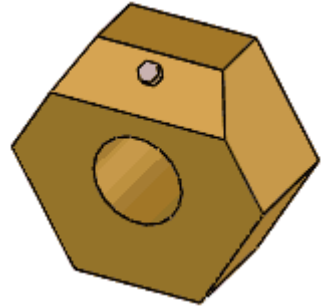
The dialog box warns that the reference plane is missing for the sketch used to create the **Extrude2** feature.

- 4 Click **Close**.

In the FeatureManager design tree, the part icon, **Extrude2** feature, and **Sketch3** display  and  yellow warning icons that indicate the sketch plane is dangling.



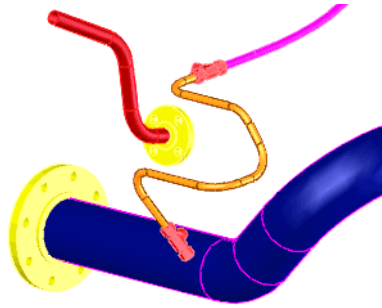
You can use the **Edit Sketch Plane** command to add a reference plane.



SolidWorks Routing

The SolidWorks Routing add-in now supports both tubing and piping design. This product was formerly called SolidWorks Piping.

You can now add different connection points for tubing or piping on a fitting or equipment connection, so when you start a route, the type of route is known. You can have a mix of tubing and piping in an assembly, each type with its own route.



PropertyManagers control properties for:

- **Connection Point**
- **Route Point**
- **Route Properties**

The **Route Properties** PropertyManager handles bends and elbows and you can choose options such as:

- **Use flexible**
- **Bend radius:** **Inside** or **Center line** (changes the default bend radius)
- **Create custom fittings**
- **Create pipes on open lines**
- **Automatically create fillets**

Previously, any bend in a route was filled with a standard elbow if possible, or a custom elbow if not. Now you can bend the pipe and create routes with both elbows and bends, and also flexible tubes and hoses.

If you choose **Use flexible** (available only for tubing), you can include splines and fit splines in route sketches to create flexible tubing, hoses, and so on.

You can output tube and pipe bend information to files for driving bending machines.

New library content for tubing has been added.

SolidWorks Routing sub-assemblies (tubing and pipe based) can be inserted from the FeaturePalette™ in the same way that a T or reducer is inserted. The tube or pipe is trimmed and its length adjusted on both sides of the inserted assembly.

File locations have been streamlined. Part files are saved in the directory with the assembly file.

You can now control the display of connection points and route points, including size of label text and whether the label is displayed, which aids in selection of entities.

Route details can be included in drawing Bills of Materials.

SolidWorks Task Scheduler

SolidWorks Task Scheduler is a new utility that enables you to set up tasks to be performed at a future time. For example, if you need to perform a resource-intensive task such as rebuilding a large assembly, you can use Task Scheduler to perform the job at off-peak hours. You can schedule a task to be performed only once or on a daily, weekly, or monthly basis. The tasks you can schedule are:



Update Files - Rebuilds specified files.



Update Associated Files - Rebuilds the specified file, plus any associated files located in specified folders.



Print Files - Prints specified files using default document printer settings or selected options.



Import Files - Imports STEP or IGES files and saves them as SolidWorks documents.



Export Files - Exports SolidWorks documents to another file format (DXF, DWG, IGES, STEP AP203, or STEP AP214).



Update Custom Properties - Adds or modifies custom properties in the specified files.



Create Drawings - Creates drawings for parts or assemblies, using the specified drawing document template.



Convert to High Quality Views - Converts drawing views from **Draft Quality** to **High Quality**.



Run Custom Task - Runs any application, using macros or command line arguments that you specify.



Create eDrawings - Creates eDrawings, using selected export options.

Scheduling Tasks


Task Scheduler runs separately from the main SolidWorks application. In the following example, you schedule a task to update a part file.

To open Task Scheduler:

- From the Windows Start menu, click **Programs, SolidWorks 2004, SolidWorks Task Scheduler**.

The **SolidWorks Task Scheduler** window appears.

To update a file using Task Scheduler:

- 1 Click **Update Files**  on the sidebar, or click **Tasks, Update Files**.
- 2 In the **Update Files** dialog box, click **Add File**.
- 3 Browse to <install_dir>\samples\whatsnew\ and open **bolt.sldprt**.
The path appears in the **Folder** column and the file name appears in the **File Name or Type** column.
- 4 Click **Advanced**.
- 5 In the **Advanced Options** dialog box:
 - Type **c:\temp** for **Tasks working folder**.
 - Type **3600** for **Time out (seconds)**. Task Scheduler ends the task after the task runs for this amount of time, regardless of whether or not the task is complete.
 - Select **Backup task files** to save a backup file to the **Tasks working folder** before the task is performed.
- 6 Click **OK**.
- 7 Under **Task Schedule**:
 - Select **Once** in **Running Mode**.
 - Set **Start Time** to approximately 2 minutes in the future.
 - Select today's date in **Start Date**.
- 8 Click **Finish**.

The task and its title, scheduled time, scheduled date, and status appear in the **Tasks** pane. The status of the task is **Scheduled**.



To run a scheduled task, your computer must be on at the scheduled start time. The **SolidWorks Task Scheduler** window does not need to be open.

At the scheduled start time, the task is initiated. A backup copy of **bolt.sldprt** is saved to a zip file in **c:\temp**, then **bolt.sldprt** is updated and saved in the original folder. When the task is complete, the status changes to **Completed**.



After the task is completed, click **Completed** to view a **SolidWorks Task Scheduler Report**.

Managing Tasks

The tasks you create appear in the **Tasks** pane. You can filter which tasks to display, such as the tasks for **Today**, **All Future Dates**, **All**, and so on, by using the **Show Tasks** box on the **Tasks** toolbar. From the **Tasks** pane you can edit, copy, or delete any listed tasks and print the tasks list.

To edit a task:

- 1 Click the task's **Title**.

The **Properties** dialog box for that task appears.

- 2 Make the changes and click **OK**.



You can edit a task only when its status is **Scheduled**.

To copy a task:

- 1 Click the task's index number in the leftmost column of the task list to select the task.
- 2 Click **Copy** on the **Tasks** toolbar.

A **Properties** dialog box for the new task appears with information from the copied task.

- 3 Make the changes and click **OK**.

To delete a task:

- 1 Click the task's index number in the leftmost column of the task list to select the task.
- 2 Click **Delete** on the **Tasks** toolbar.

To print the tasks list:

Click **Print** on the **Tasks** toolbar.



To preview the printed list, click **File, Print Preview**. To change printer settings, click **File, Print Setup**.

SolidWorks Office Add-Ins

This chapter describes enhancements to the following SolidWorks Office add-ins:

- ☐ eDrawings
- ☐ eDrawings Professional
- ☐ FeatureWorks
- ☐ PDMWorks
- ☐ PhotoWorks release 2
- ☐ SolidWorks Animator
- ☐ SolidWorks Toolbox
- ☐ SolidWorks Utilities

eDrawings


Animation Customization

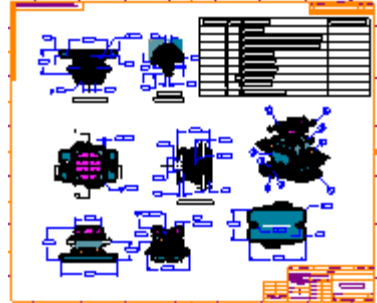
You can customize the sequence of views in animations for eDrawings part, assembly, and drawing documents using the **Animation** tab.


To customize animations:






- 1 Open **ToolVise.edrw** in the eDrawings Viewer.
- 2 Click **Animation, Reorder**.

On the **Animation** tab, the current view sequence appears in a list with the first view in the animation on top and the last view on the bottom.

- 3 Select **Drawing View2** from the list, then click **Up**  until this view is first in the list.






For parts and assemblies, select **Show Preview**  to display the view in the graphics area as you reorder it in the list.

- 4 Select the following views, then click **Up**  and **Down**  to make them the next two views in the sequence:
 - Section View A-A
 - Section View B-B
- 5 Click **OK** .
- 6 Click **Home**  to return the animation to the start of the new sequence.
- 7 Click **Continuous Play**  to display the customized animation.

The first four views displayed match the reordered sequence.



For drawings, on the **Sheets** tab, the currently displayed view displays this icon  during the animation. All other drawing views on the same sheet change to an uncolored drawing view icon .

- 8 Click **Stop** .

Configuration Names

In assemblies, eDrawings displays component configuration names next to part names for components that have multiple configurations.

Enable Performance Email

You can enable the performance email tool when you install eDrawings, or when you open the first eDrawings executable or HTML file. This performance tool sends email to SolidWorks regarding the performance and usage of the software. You can change this setting under **Tools, Options, General, Enable performance email**.


Export Formats

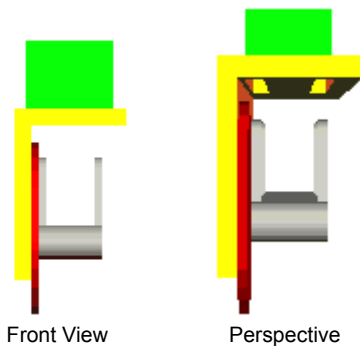
You can export files as STL (*.stl), BMP (*.bmp), TIFF (*.tif), and JPEG (*.jpeg) files. Click **File, Save As**, and set **Save as type** to the desired format.

For STL files, you can export eDrawings part and assembly files using binary or ASCII format. The exported file uses the units from the current model. If cross sections are shown in the model, they are not shown in the STL file.

For BMP, TIFF, and JPEG, you can export eDrawings part, assembly, and drawing documents. The image in the exported file matches the size, color depth, and resolution of the screen display. The image includes entities such as cross sections, markup comments, measured dimensions, and so on.

Perspective View

You can display a perspective view of models. A perspective view is the view normally seen by the eye. Parallel lines recede into the distance to a vanishing point. Click **Perspective**  or **View, Perspective** to display a perspective view of the model.



Send or Save as Zip Files

You can send or save eDrawings files as Zip (*.zip) files. The Zip file contains the eDrawings file and the eDrawings executable.

When you send a Zip file, the recipient must unzip the executable to view the eDrawings file. The **Send as Zip** option replaces the **Send as masked executable** option from previous versions.







To save eDrawings files as Zip files, in the **Save As** dialog box, select **eDrawing Zip Files (*.zip)** in the **Save as type** list.

Standard View Orientations

eDrawings imports the standard views from SolidWorks documents. If you change the standard view definitions in SolidWorks, eDrawings retains the modified definitions when you publish the eDrawings file. eDrawings supports the **Front**, **Back**, **Left**, **Right**, **Top**, **Bottom**, and **Isometric** standard views.

Tool Icons

Most tool icons are brighter and more colorful. The table below shows a few samples of the old and new icons. The tools' functionality has not changed.

Tool	Old Icon	New eDrawings 2004 Icon
Open		
Home		
Rotate		

UI Mode

The eDrawings Viewer has two user interface (UI) modes: simplified and complete. In the simplified UI mode:

- The eDrawing Manager is not shown.
- The graphics area expands to the full width of the eDrawings Viewer.
- Under **Tools**, all markup-related items, such as **Move**, **Measure**, **Cross Section**, **Comments**, and all markup tools, are unavailable.
- The markup toolbar is not shown.



If you open an eDrawings file that contains markup data, the eDrawings Viewer initially displays the complete UI.

Toggling Between UI Modes

In the eDrawings Viewer, you can toggle between the simplified and complete UI.

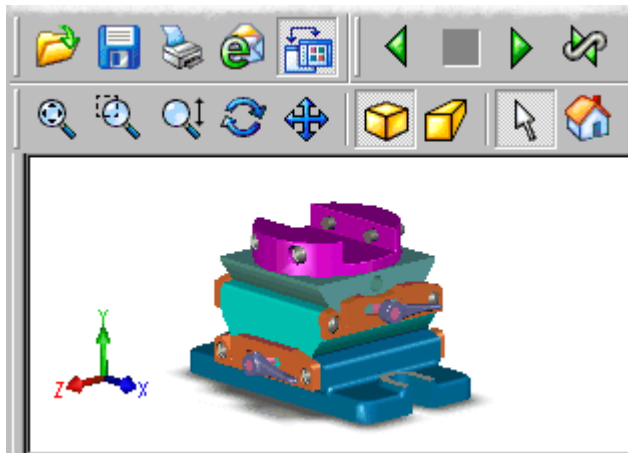
To toggle between UI modes:


- 1 Open an eDrawings file.

The eDrawings Viewer displays the file with the complete UI.

- 2 Click **Toggle between Simple and Complete UI mode**  or select **View, UI Mode, Simple**.

The simplified UI appears.




- 3 Click **Toggle between Simple and Complete UI mode**  again to return to the complete UI mode.

Sending and Saving Files With UI Modes

When you send or save eDrawings files as Zip, HTML, or Executable files, you can choose the UI mode for the eDrawings file.

To choose a UI mode when sending eDrawings files:

- 1 With an eDrawings file open, click **Send**  or **File, Send**.
- 2 In the **Send As** dialog box, select **Zip (.zip)**, **HTML page (.htm)**, or **Executable (.exe)**.



You do not have the option to choose the UI when you send files as eDrawings files.

- 3 Select **Send with Simplified UI** to have the eDrawings file open with the simplified UI in the recipient's email program.
- 4 Click **OK**.
- 5 In your default email application, enter the recipient's address and click **Send**.

To choose a UI mode when saving eDrawings files:

- 1 With an eDrawings file open, click **File, Save As**.
- 2 In the **Save As** dialog box, under **Save as type**, select **eDrawing Zip Files (*.zip)**, **eDrawing HTML (*.html) Files**, or **eDrawing Executable Files (*.exe) Files**.




You do not have the option to choose the UI when you save files as eDrawings, STL, BMP, TIFF, or JPEG files.

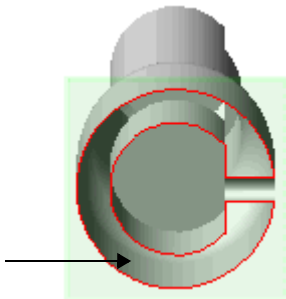
- 3 Select **Save with Simplified UI**.
- 4 Click **Save**.

eDrawings Professional

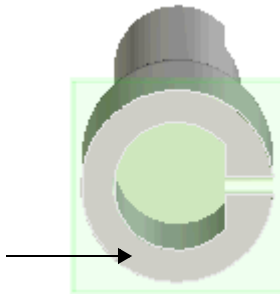
This section describes new features and enhancements that are available only in the eDrawings Professional version.

Cross Sections


Cross sections are capped with an end face to present a more realistic view of sectioned models. Previously, uncapped cross sections made solids appear hollow. You can select or clear **Show Cap**  on the **Cross Section** tab to display or clear capped cross sections.



OLD - Uncapped cross section with hollow end face



NEW - Capped cross section with solid capped end face

You can show or hide the cross section plane when viewing cross sections. Select or clear **Show Plane**  on the **Cross Section** tab to show or hide the cross section plane.



Show cross section plane





Hide cross section plane

Automatic Extrude and Revolve Recognition

FeatureWorks independently recognizes extrude and revolve features using automatic feature recognition. Previously, you had to select the **Basic features** option, which recognized both extrudes and revolves.

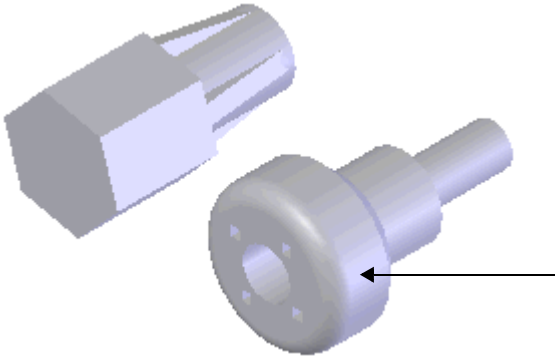
To independently recognize extrudes and revolves:

- 1 Open **MultibodyFW.x_t**.
- 2 When prompted, click **Yes** to automatically recognize features.
- 3 In the **FeatureWorks** dialog box, select **Imported1** as the body to recognize, then click **OK**.
- 4 In the PropertyManager:
 - Under **Recognition Mode**, select **Automatic**.
 - Under **Feature Group**, select **Standard features**.
 - Under **Automatic Features**, select **Revolves**. Clear all other check boxes.
- 5 Click **Recognize**.

In the FeatureWorks FeatureManager design tree , note that FeatureWorks recognized the  **Cut-Revolve1** feature.
- 6 In the **Feature Manager** dialog box, click **Continue**.
- 7 In the PropertyManager, under **Automatic Features**, select **Extrudes** and **Fillets/Chamfers**. Clear all other **Automatic Features** check boxes.
- 8 Click **Recognize**.

FeatureWorks automatically recognizes the extrudes and other features, which appear in the FeatureWorks FeatureManager design tree.
- 9 In the **Feature Manager** dialog box, click **Map Features**.

FeatureWorks recognizes the entire **Imported1** body. All recognized features appear in the SolidWorks FeatureManager design tree.




Keep this document open for the next item, **Multibody Recognition**.


Multibody Recognition

FeatureWorks recognizes features from multiple imported bodies in one session. Previously, FeatureWorks recognized only one imported body per session and deleted any other imported bodies.

To see an example of the enhanced multibody recognition:

- 1 Complete the previous section, **Automatic Extrude and Revolve Recognition**, if not already done.

You recognized one of two imported bodies. The unrecognized  **Imported3** body is available for recognition in the SolidWorks FeatureManager design tree.

- 2 Right-click  **Imported3** and select **Recognize Features**.
- 3 In the PropertyManager, under **Automatic Features**, select all check boxes.
- 4 Click **Recognize**.

In the FeatureWorks FeatureManager design tree, FeatureWorks recognizes extrude, revolve, hole, rib, and chamfer features.

- 5 In the **Feature Manager** dialog box, click **Map Features**.

FeatureWorks recognizes all features in the **Imported3** body. These features replace the imported body icon in the SolidWorks FeatureManager design tree.

Step-by-Step Recognition

You can recognize some imported body features from a part, save the part, then recognize more features from the same imported body at a later time. You can also recognize features of partially recognized parts (that contain imported bodies and recognized features). Previously, you had to completely recognize an imported body to retain the recognized features, and you could not recognize features in partially recognized parts.

To recognize imported body features step-by-step:

- 1 Open **StepByStep.sldprt**.

The imported body in the part document contains hole, extrude, and revolve features. Note the loft feature in the FeatureManager design tree.

- 2 When prompted, click **Yes** to automatically recognize features.


The **FeatureWorks** dialog box warns you that loft features are not supported by FeatureWorks. You can delete the feature or map it as a thicken feature.

- 3 Select **Map unsupported features as thicken features**, then click **OK**.

- 4 In the PropertyManager:

- Under **Recognition Mode**, select **Automatic**.
- Under **Feature Group**, select **Standard features**.
- Under **Automatic Features**, select **Holes**.
Clear all other check boxes.

- 5 Click **Recognize**.


In the FeatureWorks FeatureManager design tree, note that FeatureWorks recognized only the  **Hole** feature.

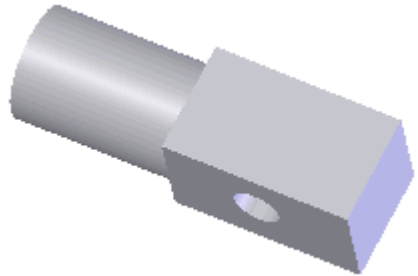
- 6 In the **Feature Manager** dialog box, click **Map Features**.

The hole and thicken features appear in the SolidWorks FeatureManager design tree.

- 7 Save the document as **Holes.sldprt**.

Now you can recognize other features contained in the partially recognized imported body.

- 1 In the SolidWorks FeatureManager design tree, right-click  **Imported1** and select **Recognize Features**.
- 2 In the PropertyManager, under **Automatic Features**, select **Revolves**, and clear all other check boxes.



3 Click **Recognize**.

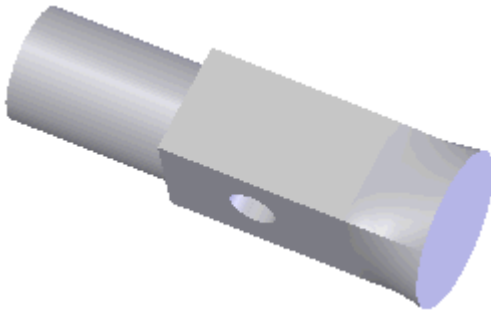
FeatureWorks automatically recognizes the revolve features, which appear in the FeatureWorks FeatureManager design tree.

4 In the **Feature Manager** dialog box, click **Map Features** to map the revolve features into the SolidWorks FeatureManager design tree.

5 Save the document as **RevolveHoles.sldprt**.

You can save the partially recognized document to preserve its various stages of recognition.

6 Repeat steps 1 through 4 for extrude features to complete recognition of the imported body features.



PDMWorks

Installation

The installation procedure has been streamlined. Now you select **Client** or **Server** installation.

If you install a **Client**, you can choose a path in **Custom** or let PDMWorks use its defaults in **Typical** or **Complete** versions.

New Vault folders are now named **c:\VaultData**. Existing vaults are not affected.

At the end of the **Server** installation, you can go directly to a **Client** installation.

PDMWorks supports Microsoft Windows Server 2004.

Documentation

A PDMWorks lesson has been added to the online tutorial.

Best Practices and **Frequently Asked Questions** topics have been added to the PDMWorks online help.

VaultAdmin

The **Save** button on the VaultAdmin tabs is now named **Apply**.

Vault Settings. New global settings:

- Allow users to create a reference to a document owned by another user when they are checking in a document type other than SolidWorks.
- Allow the **.prt**, **.asm**, and **.drw** extensions to be associated with applications other than SolidWorks.
- Restrict users to open documents read-only if they are not the owner.
- Allow users to rename documents in the Vault.

Projects. New functionality:

- **Bulk check-in files**. You can select the revision number, add a note to the documents, and generate a report. The check-in procedure recognizes whether **.prt**, **.asm**, and **.drw** files are SolidWorks files.
- **Archive and Restore document revisions**. You can select revisions for archiving or restoring manually or based on primary, secondary, and tertiary criteria. The file date and path appear with the file name in a window. You can browse to an archive directory.

Users and Groups. In addition to the user name and password, you can enter **Display name**, **Email**, and **Comment** when adding a user.

Lifecycle. You can assign access rights based on lifecycle status and prevent users from returning to a specified lifecycle status.

Properties. You can map SolidWorks document properties to PDMWorks properties; for example, map SolidWorks property **PartNo** to PDMWorks property **Number**.

Clients

The **Preview** window and **Document Information** dialog box now show configuration-specific previews. The new **View** tab in the **Document Information** window uses eDrawings to view SolidWorks and AutoCAD documents.

You can rename files in the Vault if given permission by the vault administrator.

You you save revision information in the file properties, you can use it during check-in.

You can archive or restore document revisions if given permission by the vault administrator.

Toolbox

Toolbox parts now contain an internal flag identifying them as such, which is useful when identifying Toolbox parts on a path other than the path specified by the administrator. A utility `<installation_directory>\toolbox\slidsetdocprop.exe` is available to add the flag to old Toolbox parts.

AutoCAD Support

On the VaultAdmin Properties tab, you can map properties; for example, map AutoCAD block attributes to PDMWorks properties.



Bulk check-in (see **VaultAdmin** on page 10-11) is particularly useful for legacy AutoCAD documents.

A PDMWorks menu is available from within AutoCAD.

You can view AutoCAD documents in the PDMWorks Web Portal.

Web Portal

eDrawings has been added so you no longer need to download the SolidWorks or AutoCAD viewers.

The vault administrator can restrict the ability of read-only users to open documents in the Web Portal. Security has been enhanced to require user login when accessing the Vault by means of a URL path.

For projects with large numbers of files, ranges break the projects into smaller chunks for faster manipulation.

PhotoWorks release 2

PhotoWorks release 2 adds enhancements to fog lighting controls, file management, and depth-fading contour rendering. See the PhotoWorks release 2 Online Help for a complete list of enhancements.


PhotoWorks release 2, based on mental ray® rendering technology, has superseded previous versions of the PhotoWorks product. All subscription service customers of PhotoWorks have been upgraded to PhotoWorks release 2. Previous versions of PhotoWorks cannot be added-in to SolidWorks 2004.

SolidWorks Animator

A default pop-up playback toolbar with all functions enabled is deployed when you do one of the following:

- Create an animation with SolidWorks Animator and select playback.
- Create an assembly animation with Physical Simulation and select playback.



To select playback:

Click **Animator**, **Animation**, **Play**, or click **Play**  on the Animation Controller toolbar.

The pop-up playback toolbar appears.






You can animate an exploded view using the right-mouse button.









To animate an exploded view:

- 1 Create an exploded view.
- 2 Click **ConfigurationManager** .
- 3 Right-click the exploded view .
- 4 Select **Animate Collapse**.

The Animation Controller toolbar appears, and an animated collapse of the assembly plays in the graphics area.

- 5 Repeat step 3 and select **Animation Explode** to reverse the process.

SolidWorks Animator Tools		
	Start	Start the animation.
	Rewind	Rewind the animation.
	Play	Play the animation.
	Fast	Forward. Play the animation ahead fast.
	End	Go to the end of the animation.

SolidWorks Animator Tools		
	Pause	Temporarily pause the animation.
	Stop	Stop the animation.
	Record	Record the animation (only with SolidWorks Animator).
	Normal	Play the animation at normal speed.
	Loop	Play the animation in a continuous loop.
	Reciprocate	Play the animation forward, then reverse in a continuous loop.
	Slow Play	Play the animation at half-speed.
	Fast Play	Play the animation at twice the normal speed.

The pop-up toolbar is now available when you stop an animation, so you no longer need to exit the animation mode to replay the animation sequence.

As you play the animation, the pop-up toolbar displays the following:

- Visual progress slider
- **Total Time** readout

The Animation Controller bar is now simplified, and includes the following functions:


Animation Controller	
	Animation Wizard
	Create Path
	Traverse Path
	Edit Path
	Play
	Record Animation
	Turn off screen capture
	Turn on screen capture

SolidWorks Toolbox

Alternate Part Names

You can add an alternate name for the parts supplied with SolidWorks Toolbox. The alternate name appears in the FeatureManager design tree and applies to the file name in your Toolbox library of parts.



Existing assemblies do not update with the new name. To update existing assemblies, use SolidWorks Explorer or **Replace**  in an assembly.

To create an alternate file name:

- 1 In a part or assembly document, click **Toolbox, Browser configuration**.
- 2 In the dialog box, on the **Browser** tab, expand **Document Properties** and select **File Names**.
- 3 Select an item in **Catalogs**, **Chapters**, and **Pages** to set the names.
- 4 Locate the part in the table, click in **Alternate File Name**, and type the new name.
- 5 Repeat steps 3 and 4 for other file names.
- 6 Click **OK**.

Hole Wizard Interaction

SolidWorks Toolbox and the Hole Wizard feature share the same database and user-defined standards. See **Hole Wizard** on page 5-4.

SolidWorks Utilities

Compare Documents

The **Enable face selection** option has been removed from the following dialog boxes:

- **Compare Geometry: Results**
- **Compare Documents: Results**

Compare Features

After you run Compare Features, click individual nodes in the **Compare Features: Results** dialog box to see callouts in the graphics area.



Geometry Analysis

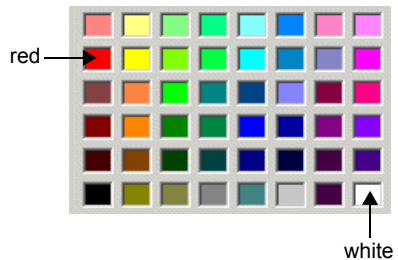
The Geometry Analysis utility has moved from a dialog box to a PropertyManager. All previous dialog box functionality is retained in the PropertyManager.

Thickness Analysis

Use the Thickness Analysis utility to determine different thicknesses of a part. Thickness Analysis identifies thin and thick regions, and determines the portions of a part that are equal to a specified target thickness. This utility is helpful with thin-walled plastic parts.

To determine thin regions of a part using a single color range:

- 1 Open **bucket.sldprt**.
- 2 Click **Thickness Analysis**  on the Utilities toolbar, or click **Utilities, Thickness Analysis**.
- 3 In the PropertyManager, under **Analysis Parameters**:
 - a) Type **3** for **Target thickness** . This is the value to which you want the part to conform.
 - b) Select **Show thin regions**.
- 4 Under **Color Settings**:
 - a) Clear **Full color range**.
 - b) Click **Edit Color** under **Target thickness color** and select red as shown.
 - c) Make sure **Minimum thickness color** is set to white as shown.

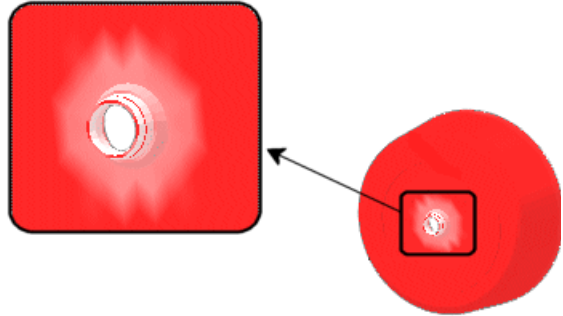


The **Thickness scale** updates to show the shades of red that will be used to display the thickness ranges. The brightest shade illustrates the target thickness (3mm), and the palest shade illustrates the thinnest regions of the part.

- 5 Under **Analysis Parameters**, click **Calculate**.

The **Thickness scale** updates to show the values for each shade of red.

The bucket is highlighted in various shades of red to indicate the different thicknesses. Move the pointer over the part to see the tooltip display thickness values.



- 6 Keep the PropertyManager open for the next procedure.

To determine thin regions of a part using a full color range:

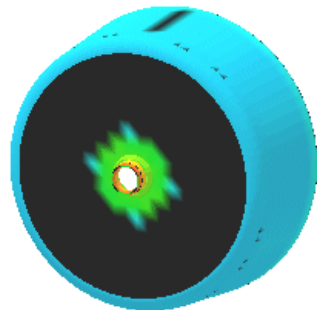
- 1 In the PropertyManager, under **Color Settings**, select **Full color range**.
- 2 Click **Edit Color** under **Target thickness color** and select black.

The **Thickness scale** updates to show the values of each shade in the color range. Blue indicates values closest to the target thickness and red indicates the thinnest regions of the part. Black indicates thicknesses equal to 3mm.

- 3 Click **Calculate**.


The part is displayed in various colors to indicate the different thicknesses. Because you did not change any other values in the PropertyManager, the calculation updates the colors immediately.

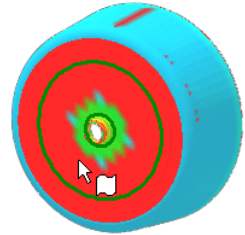
- 4 Keep the PropertyManager open for the next procedure.



To determine thin regions of a part with a high resolution:

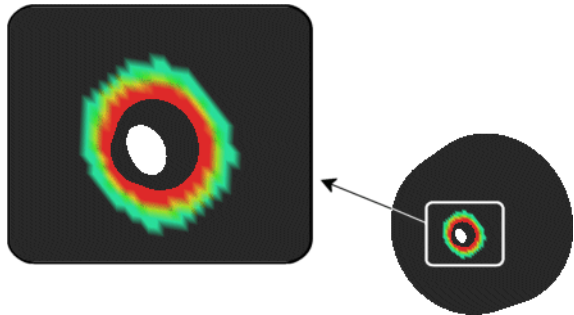
When you increase the resolution for Thickness Analysis, the utility takes longer to calculate the values. However, you can use the utility on selected faces to decrease calculation time. An increased resolution gives more accurate thickness results.

- 1 In the PropertyManager, under **Performance/Accuracy**:
 - a) Select the face as shown for **Face for local analysis** .
 - b) Move the **Resolution**  slider to **High**.
- 2 Click **Calculate**.





Only the selected face is displayed in various colors to indicate the different thicknesses.

- 3 Keep the PropertyManager open for the next procedure.



To determine the thickness of a part within a range:

- 1 Under **Analysis Parameters**:
 - a) Select **Show thick regions**.
 - b) Type **6** for **Thick region limit**. The utility will find the part's range of thicknesses between 3mm and 6mm.
 - c) Select **Treat corners as zero thickness**. When Thickness Analysis is run, the utility disregards sharp corners as thin. All corners are treated as having uniform thickness.
- 2 Under **Performance/Accuracy**:
 - a) Clear **Face <1>** in **Face for local analysis** .
 - b) Move the **Resolution**  slider to **Low**.

- 3 Click **Calculate**.


The **Thickness scale** updates to show the values of each shade in the color range.

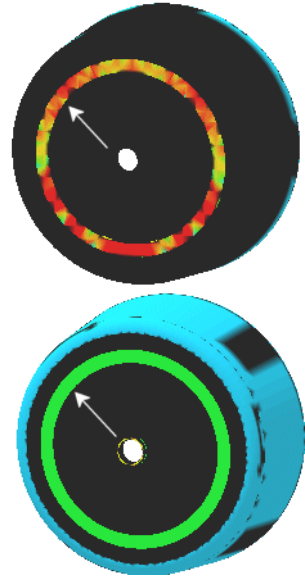
The inside ring around the base of the part is displayed in various colors to indicate the different thicknesses.

- 4 Under **Analysis Parameters**, clear **Treat corners as zero thickness**.

- 5 Click **Calculate**.

The inside ring around the base of the part is displayed with uniform thickness of 4mm.

- 6 Click  to close the PropertyManager.



SolidWorks 2003 Service Pack Enhancements

This appendix contains information about new and changed functionality introduced in the Service Packs between the release of SolidWorks 2003 and the release of SolidWorks 2004.

Assemblies

Mass Properties of a Component

When you calculate the mass properties of a component, SolidWorks now accounts for assembly feature cuts or holes through the component. The mass properties of an assembly do not change; these calculations previously accounted for assembly feature cuts or holes.

Suppressing the Simulation Folder

You can suppress and configure the suppression state of the **Simulation** folder and its simulation elements if you are using **Physical Simulation**. Suppressing the folder also suppresses its elements.

Design Tables

You can now control the configuration of a split part with a design table.

Detailing

For Hole Wizard Tapped Holes, you must rebuild any holes created before SolidWorks 2003 for the correct information to appear in a Hole Callout. In the part document, right-click the hole in the FeatureManager design tree and select **Edit Definition**. In the **Hole Definition** dialog box, click **Next** and **Finish** without making any changes.

Drawings

Tangent Edge Display Dialog Box

In new installations, when you insert a view into a drawing document, a dialog box appears so you can specify the Tangent Edge Display for the current view and future views. The default that appears in the dialog box is set under **Default display of tangent edges in new drawing views** in **Tools, Options, System Options, Drawings, Default Display Type**. You can select a check box, **Don't ask me again**, to prevent the dialog box from appearing again. Changes in this **Tangent Edge Display** dialog box are reflected in the **Default Display Type Options**.

Unexpected Dimensions

When opening drawings created prior to SolidWorks 98, you may see dimensions appear unexpectedly. This results from improvements in the SolidWorks 2003 software. These dimensions were not hidden; they should have appeared during an operation such as **Insert Model Items** in the earlier release of SolidWorks. You can hide or delete these dimensions.

Replacing Referenced Model

The mechanism to replace the model referenced in a drawing view using **File, Reload** has changed. The functionality has been moved to the **File, Open** dialog box as follows:

- 1 Click **File, Open**.
- 2 In the **Open** dialog box, select the drawing, but do not open it.
- 3 Click **References**.
- 4 In the **Edit Reference File Locations** dialog box, edit the references under **New pathname**.
- 5 Click **OK**, then **Open**.

Dimension Types

SolidWorks now selects **True** type dimensions for isometric, dimetric, and trimetric views. Previously you were prompted for dimension type when inserting these types of named views. You are still prompted when inserting views in other orientations, such as custom views that are not orthogonal.

eDrawings

- When you save a SolidWorks drawing document as an eDrawing file, you have the option to save the shaded data with the published eDrawing file.
- In the eDrawings Viewer, you can open any SolidWorks document from SolidWorks 97Plus and higher, even if the document does not have eDrawings data saved in it.

Fundamentals

Macros

When you assign a macro to a keyboard key or to a menu, you can specify which method within the macro you want to run.

Use Software OpenGL

When you select the **Use Software OpenGL** check box in **Tools, Options, System Options, Performance**, some of your selected system options will change. This is done for optimum display performance. For a list of the options that change, see Performance Options in the *SolidWorks Online User's Guide*.

Manufacturing Network

Click **Tools, Manufacturing Network** to access an online directory of SolidWorks-enabled service vendors. The Manufacturing Network lists contract design and manufacturing companies who can help reduce your product's time to market.

Shortcut Menus

A new option, **System Options, General, Always show full shortcut menus**, expands all shortcut menus so they are never truncated.

Import/Export

Autodesk Inventor

You can now set import options when you import Autodesk Inventor files into the SolidWorks application.

CATIA Graphics (CGR) Files

To import CGR files, you need Windows NT or Windows 2000 or above.

AutoCAD 2004 DXF or DWG

To import an AutoCAD 2004 DXF or DWG file, you must save the file in AutoCAD 2004 as “AutoCAD 2000/LT2000.” SolidWorks warns you if you try to import AutoCAD 2004 DXF or DWG files that are not saved in this format.

PDMWorks

Login user names and passwords are now case-sensitive.

PhotoWorks release 2

PhotoWorks release 2 is a major revision of the PhotoWorks software. See the PhotoWorks release 2 Online Help for more information. Items to note include:

- The Archive Converter is not be available in SolidWorks 2004. You need to convert your material and scene archives with SolidWorks 2003.
- PhotoWorks legacy data **will be lost if you do not translate** into PhotoWorks release 2 format. You can choose to not translate the data. But if you do not, and you save the part or assembly, you will not be prompted to translate the legacy data again and that data is lost.
- For an InstallShield client-server installation, you need to patch each client via a CD install.

Sketching

To improve performance, sketches no longer inference to temporary axes unless the temporary axes have been made visible in the **View** menu.

SolidWorks Piping

3D Content Central

A Piping and Tubing section has been added to 3D Content Central and includes Flanges, Piping Components, and Tubing Components. A library of copper piping and tubing parts is designed per American standards. The parts include connection points, route points, and so on for SolidWorks Piping.

Flanges

The bolt holes in flanges in the Feature Palette now match American industry convention. The bolt holes are now staggered about the centerline of the flange. Previously one hole was always on the vertical centerline.

Forged Threaded (NPT) Pipe Fittings

The Feature Palette now includes NPT fittings per American standard ASME B16.11.

Other Enhancements

- It is now possible to control alignment of pipe fittings when the sketch contains multiple construction lines.
- If a reducer contains an axis called **Vertical** and the 3D sketch has a construction line at the end for controlling the angular alignment of the reducer, the reducer is now inserted correctly.
- Alignment of the eccentric reducer using construction lines now works when the first connection pipe in the series is connection point2.
- The pipe route connection point now connects correctly in a custom Y fitting.
- The eccentric reducer part shipped with Piping now has its rotation axis named as **Vertical** to facilitate control of the reducer alignment.
- The connection point direction is now calculated using Face/Plane normal, so the display shows the connection point direction correctly. If the direction of the pipe coming out of a connection point is incorrect, you can now edit the direction.
- **Undo** for fitting-to-fitting operations is now supported. For example, when you delete a route between two fittings, the two fittings move close together. If you then click **Undo**, now the route is added back and the fittings move apart.
- Both the directional arrow of the connection point (**CPOINT1**) for custom flange and the connected pipe now come out from the same direction.
- The pipe length with a 10x8 eccentric reducer is now displayed correctly relative to the dimensioned length.

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