# SolidWorks 2005
# What's New

Document Number: SWMWNENG0504

# Contents

## Chapter 2 Sketching

## Chapter 3 Features

## Chapter 4 Parts



## Chapter 5 Mold Tools



## Chapter 6 Weldments

## Chapter 7 Assemblies



## Chapter 8 Configurations



## Chapter 9 Drawings

## Chapter 10 Detailing



## Chapter 11 Library Features

# Introduction

## About this Book

This book highlights and helps you learn the new functionality in the SolidWorks® 2005 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 2005 software, refer to the *SolidWorks Online User's Guide* by clicking **Help**, **SolidWorks Help Topics**.

### Intended Audience

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

### Additional Resources

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

- **SolidWorks 2005 What's New Highlights**. This book provides the highlights of the new functionality in the SolidWorks software. This book is available in printed format for subscription customers.
- **Interactive What's New**. Click next to new menu items and the title of new and changed PropertyManagers to read what is new about the command. A help topic appears with the text from this manual.
- **Online Help**. The online help contains brief descriptions of the new functionality with links to the complete, related help topics.

## Late Changes

Due to software deadlines, this book may not include all of the enhancements in the SolidWorks 2005 software. For enhancements that are not in this book, refer to the *SolidWorks Release Notes* by clicking **Help**, **SolidWorks Release Notes**.

## Using This Book

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

#### *To use the example files:*

1. Install the SolidWorks 2005 software.
2. Be sure to select the option to install the **Example Files**.

   The example files are placed in the <*install_dir*>**samples****whatsnew** folder.
3. Open the example files from the **File Explorer** when instructed to do so.

   The **File Explorer** tab in the Task Pane duplicates Windows® Explorer® on your local computer. See **File Explorer** on page 1-3.

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

## Conventions Used in this Book

This book uses the following conventions:

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

# Moving to SolidWorks 2005

## Converting Older SolidWorks Files to SolidWorks 2005

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 2005 format. To access the Conversion Wizard, click the Microsoft® **Start** button, select **Programs**, **SolidWorks 2005**, **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 2005 software. To check for a new service pack, click **Help**, **Service Packs**, and click **Check**. Select the check box if you want the software to automatically check the SolidWorks Web site for a new service pack once a week.

1

# SolidWorks Fundamentals

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

- Task Pane
- Design Journal
- User interface
- Toolbars
- Repeat Last Command
- Selection
- Multi-user environment
- Copy Settings Wizard
- Options
- Documentation

## Task Pane

A new Task Pane appears when you open SolidWorks. It contains three tabs.

**SolidWorks Resources**

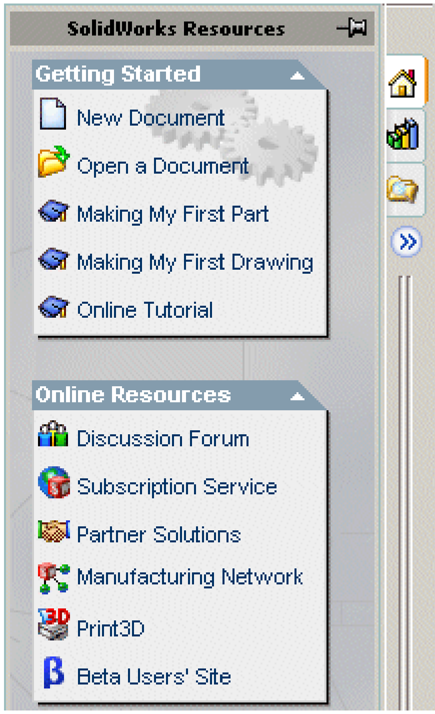
**Design Library**

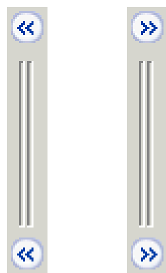
**File Explorer**

The Task Pane can be in the following states:

- **Visible or hidden**. Right-click in the border of the graphics area and select or clear **Task Pane**, or click **View**, **Task Pane**. To change tabs or open a pane, click a tab icon.
- **Expanded or collapsed**. Click one of the arrows or anywhere along the bar between the arrows. You cannot collapse the Task Pane if it is pinned.
- **Pinned or unpinned**. Click to pin the Task Pane. Click again to unpin it. If the Task Pane is unpinned, it collapses when you drag an item such as a document, a library feature, or an annotation into the graphics area, or when you open a new SolidWorks document.
- **Docked or floating**. To float the Task Pane, drag it by the bar between the arrows. To dock it again, drag it to a window border or click in the title bar.

To resize the Task Pane, drag its edge.



Expand Collapse

### SolidWorks Resources

The **SolidWorks Resources** tab in the Task Pane replaces the Welcome screen from previous releases. Groups include **Getting Started** (**New Document**, **Open a Document**, and so on), **Online Resources** (**Discussion Forum**, **Subscription Page**, and so on), and **Tip of the Day** (at the bottom of the tab).

You can customize **SolidWorks Resources** based on your work flow. See **Customize by Work Flow** on page 1-10.

## File Explorer

The **File Explorer** tab in the Task Pane duplicates Windows Explorer on your local computer. In addition, the following directories are displayed:

- **Recent documents**
- **Open in SolidWorks**
- **Samples** (*What's New* and Online Tutorial sample files)

In **Open in SolidWorks**:

- Files that appear in bold have been modified since they were last saved.
- Files that appear in orange indicate read-only status.
- The icon for files open in SolidWorks is solid, and the icon for referenced components that are not open in SolidWorks is transparent.
- To display or hide referenced components, right-click **Open in SolidWorks** and select or clear **Show hidden referenced documents**. You can also set this option in **Tools**, **Options**, **System Options**, **File Explorer**. In assemblies, selecting this option hides component files that are not open in SolidWorks.

You can:

- Drag documents from the **File Explorer** into the graphics area.
- Right-click to access shortcut menus.
- Display tooltips.

Tooltips display:

- File name, path, date modified, and size.
- Preview (for documents with models, such as SolidWorks and AutoCAD®).

You can specify whether to display the following folders:

- **My Documents**
- **My Computer**
- **My Network Places**
- **Recent Documents**
- **Samples**

See **File Explorer** on page 1-17.

## Design Library

The **Design Library** tab in the Task Pane provides a central location for:

- **Design Library**
- **Toolbox**
- **3D ContentCentral**

---

To access SolidWorks Toolbox contents, you must install and add in the SolidWorks Toolbox Browser.

---

The **Design Library** contains reusable elements and recognizes file types for parts, assemblies, annotation favorites, blocks, library features, and DXF/DWG files. It does not recognize SolidWorks drawings, text files, or other non-SolidWorks files.

---

The contents of the Feature Palette™ window and Library Features have been moved from the **Tools** menu to the **Design Library**. See **Design Library** on page 11-2.

---

Tools available at the top of the **Design Library** window include:

| | |
|---|---|
| **Add File Location** | Adds a shortcut to a folder on disk. |
| **Create New Folder** | Creates a new folder on disk. |
| **Refresh** | Refreshes the display of the **Design Library** pane. |
| **Search** | Searches the **Design Library** (**Search local data** or **Search 3D ContentCentral**) for various types of files based on file or folder names or on custom properties. |

The **Design Library** folder contains subfolders, including:

- **annotations**
- **assemblies**
- **features**
- **forming tools**
- **parts**

The library folders are populated automatically, but you can add your own folders and contents as described below.

You can filter the files that are displayed for assemblies or forming tools. Right-click a folder the upper pane of the **Design Library**, then select:

- **Forming tools folder** - to mark a folder as a forming tools folder. The SolidWorks software handles forming tools differently than other parts.
- **Assemblies folder** - to display only the assembly documents in the folder.

Tooltips on the folders in the upper pane display the path and folder name. Tooltips on the files in the lower pane show the file names and previews.

**3D ContentCentral** includes:

- **Supplier Content** - contains links to supplier Web sites with SolidWorks models.
- **User Library** - contains models from individuals.

You must accept a license agreement when you first open **3D ContentCentral** before you can access the contents.

You can drag copies of parts, assemblies, features, annotations, and so on from:

- The **Design Library** into the graphics area.
- The graphics area into the lower pane of the **Design Library**.
- One folder to another in the **Design Library**.
- Microsoft Internet Explorer and Windows Explorer into the **Design Library**.

When you drag items into the **Design Library**, the **Save As** dialog box appears with the default file type for the type of item and the selected folder as the location.

- To copy a complete assembly or part into the **Design Library**, select it in the FeatureManager design tree and drag.
- To copy annotations or blocks into the **Design Library**, hold down **Shift** and drag from the graphics area. Annotations are saved with the extensions for favorites.
- To copy features, drag from either the FeatureManager design tree or the graphics area. Features are saved as Library Feature Parts.

If you modify an item that you inserted from the **Design Library**, the item is also modified in the library.

***To add an existing folder to the Design Library:***

1 Select the **Design Library** tab.

2 Select **Design Library** (or a subfolder in **Design Library**) and click **Add File Location**.

3 In the dialog box, browse to a folder, then click **OK**.

4 The selected folder name (or a folder named **Shortcut to** <*selected_folder*> if it is to be a subfolder) appears.

For top level folders, this procedure has the same effect as the next procedure using SolidWorks Options. If you delete the folder, you delete only the reference. For subfolders, if you delete the folder, you delete only the shortcut. In both cases, the folder remains on disk.

***To reference an existing folder in the Design Library:***

1 Click **Options** on the Standard toolbar, or click **Tools**, **Options**.

2 On the **System Options** tab, select **File Locations**.

3 Under **Show folders for**, select **Design Library**.

4 Click **Add**.

5 Browse to a <*install_dir*>**samples****whatsnew**\\, then click **OK**.

6 Click **OK** to close the options dialog box.

You can reference any number of folders. The folder names appear at the top level on the **Design Library** tab (that is, at the same level as **Design Library**, **Toolbox**, and **3DContentCentral**).

> If you delete a referenced folder from the **Design Library**, you delete only the *reference*. The reference is removed from **File Locations**.

***To create a new folder in the Design Library:***

1 Select the **Design Library** tab.

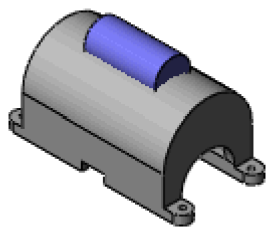
2 Select **Design Library** and click **Create New Folder**, or right-click **Design Library** and select **New Folder**.

3 Type **My Models**, then press **Enter**.

The folder appears under **Design Library** in alphabetical order.

> If you delete a folder created with **New Folder**, the folder and its contents are deleted from disk and moved to the **Recycle Bin**.

***To use a model from the Design Library:***

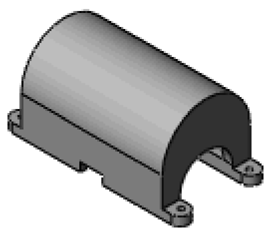


1 Click at the top of the Task Pane to pin the Task Pane open.

2 Open **indent.sldprt** from the **whatsnew** folder on the **Design Library** tab (drag the file into the graphics area, or right-click the file and select **Open**).

3 In the FeatureManager design tree, delete **Revolve** and **Sketch6**.

***To add a model to the Design Library:***



1 In the upper pane of the **Design Library** tab, select **My Models**.

2 Drag **Indent** from the FeatureManager design tree into the lower pane of the **Design Library**.

3 In the dialog box, type **My Part** for **Save As**, then click **Save**.

**My Part** appears in the lower pane of the **Design Library** and is saved on your computer in the **My Models** folder.

## Design Journal


A document called **Design Journal.doc** is embedded in SolidWorks documents. The document:

- Appears in the FeatureManager design tree in a new folder, **Design Binder**.
- Is a Microsoft Word® document.
- Resembles an engineering journal, with headings for **File Name**, **Description**, **Material**, and **Revision**.
- Initially appears as **<Empty>**, a designation that disappears when you activate the journal.
- Is added to the SolidWorks file when you activate it, so the SolidWorks file size is not affected unless you activate the journal.
- Can contain both text and embedded images. When you open the journal with Microsoft Office® 2002, the clipboard with images from the model is displayed in the graphics area. The file name is inserted automatically.
- Can be deleted, in which case it is again listed as **<Empty>**.
- Can be printed.

> You must have Microsoft Word on your computer to use the design journal.

***To edit the design journal:***

1 Open **indent.sldprt** from the **File Explorer**.

2 In the FeatureManager design tree, double-click **Design Journal.doc** in the **Design Binder** folder.

3 Type text in the journal and add images by:
   - In Microsoft Office 2002, drag images from the clipboard.
   - In other versions of Microsoft Word, use the **Paste** command to paste in the image from the active window in SolidWorks.

4 Close the journal.

   Your changes are saved automatically.

You can attach additional documents in the **Design Binder** folder and embed them in or link them to the model document.

***To insert an attachment to the journal folder:***

1. In the FeatureManager design tree, right-click **Design Binder** and select **Add Attachment**.
2. In the dialog box, browse to a file, such as a Microsoft Word or Adobe® Acrobat® file.
3. To link rather than embed the file, select **Link**.
4. Click **OK**.

   If you drag a document into the folder, it is linked rather than embedded.

You can specify the file location folder for both **SolidWorks Journal File** and **Design Journal Template** in **Tools**, **Options**, **System Options**, **File Locations**.

## Comments

You can add text and voice comments to items in the FeatureManager design tree (assemblies, components, features, sketches, and so on) in the manner of Post-It® notes. Text comments are displayed in tooltips, and a **Comments** folder is created in the FeatureManager design tree.

***To add a text comment:***

1. Right-click an item in the FeatureManager design tree and select **Comment**, **Add Comment**.
2. In the message box, type text, click **Date/Time Stamp** to add the current date and time, then click **Save and Close**.
3. Hold the pointer over the item in the FeatureManager design tree to view the comment, or expand the **Comments** folder to view all the comments in the document.

***To add a voice comment:***

To record sounds, your computer must have a microphone. You can edit the sounds, add effects, and specify audio devices.

1. Right-click an item in the FeatureManager design tree and select **Comment**, **Add Voice Comment**.
2. Record sounds in the **Sound Object in** <*filename*> dialog box.
3. Click **File**, **Exit & Return to** <*filename*>.

***To edit a comment:***

1. Right-click an item in the FeatureManager design tree that has a comment and select **Comment**, **Edit Comment** or **Play / Edit Voice Comment**.
2. Edit the comment, then click **Save and Close**.

***To delete a comment:***

Right-click an item in the FeatureManager design tree that has a comment and select **Comment**, **Delete Comment** or **Delete Voice Comment**, then click **Yes** to confirm.

# User Interface

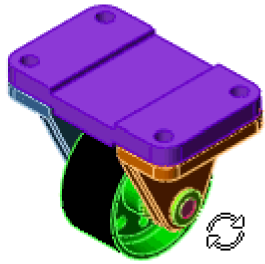
## User Interface Style

Open

2004 2005

Many aspects of the user interface (icons on toolbar buttons and in the FeatureManager design tree, scroll bars, check boxes, tabs, pointers, and so on) now follow the Windows XP style in Windows XP. In Windows 2000, the icons display new images.

## Highlighting



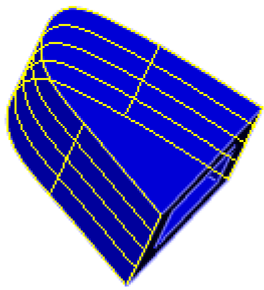
If an entity is highlighted, it remains highlighted when you manipulate a view with tools such as zoom, pan, and rotate.

## Materials

In part documents, the **Materials Editor** PropertyManager contains a list of the ten most recently-used materials.

This list also appears when you right-click **Material** in the FeatureManager design tree.

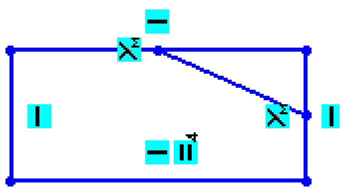
## What's Wrong



The **What's Wrong** dialog box has been redesigned.

- Messages are displayed in a table with icons that indicate **Error** or **Warning** .
- When the icon appears in the **Preview** column, select the message to highlight the geometry in the graphics area. For example, a fillet feature error is highlighted as shown.
- Click column headers to reorder messages.
- When the icon appears in the **Help** column, click to open help topics about the errors or the features.

## Sketch Relations



To display symbols of sketch relations in sketches and drawings, click **View**, **Sketch Relations**. See **Sketch Relations Display** on page 2-28.

## Customize User Interface

The SolidWorks application now recognizes toolbar customization from one release of SolidWorks to the next. Settings are saved in the registry. Other registry settings continue to be recognized from one release to the next.

Keyboard shortcut and menu customization done through **Tools**, **Customize** have been stored in **.cus** files in releases before SolidWorks 2005. If you have performed this customization in previous releases, move the **.cus** file to the corresponding folder in SolidWorks 2005 to propagate the settings for this release. From SolidWorks 2005 on, settings previously handled by **.cus** files are included in the registry and the **.cus** file is no longer needed.

To copy registry settings, use the Copy Settings Wizard. See **Copy Settings Wizard** on page 1-16.

## Customize by Work Flow

You can customize the user interface (toolbars, menus, and links on the Task Pane **SolidWorks Resources** tab) by your area of expertise. For example, in part documents, the CommandManager, in addition to the Sketch and Features toolbars, contains the following toolbars for the work flow area:

- **Machine Design** - Sheet Metal, Weldments
- **Mold Design** - Surfaces, Molds
- **Consumer Product Design** - Surfaces


Resources such as tutorials, **Partner Solutions**, **Manufacturing Network**, and **Print3D** are included in the Task Pane **SolidWorks Resources** for all industries. Resources specific to particular work areas include **CosmosXpress™** for **Machine Design**, **MoldflowXpress** for **Mold Design** and **Consumer Product Design**, and **Import File** for **Mold Design**.

***To customize the user interface by work flow:***

1 Click **Tools**, **Customize**.

2 On the **Options** tab, under **Work flow customization**, select one or more check boxes, then click **OK**.

Menus and shortcut menus that are generally not used in an industry are hidden. For example, when you select **Consumer product design**, the **Sheet Metal**, **Weldments**, and **Molds** items on the **Insert** menu are hidden.

To display hidden items, click **Customize Menu** on individual menus. You can also override the customization in **Tools**, **Customize**.

When you open SolidWorks 2005 for the first time on a computer, the **Welcome to SolidWorks** dialog box gives you the opportunity to customize the user interface by work flow.

## Toolbars

The **Select** tool now appears on the Standard toolbar as well as the Sketch toolbar. The following new tools have been added to the Standard toolbar.

| | |
|---|---|
| **Save All** | Saves all open documents |
| **Close** | Closes the active document |
| **Reload** | Reloads the current document from disk |
| **Check Read-only Files** | Checks if read-only files are out of date or available for write access |
| **Print 3D** | Places an online order for a rapid prototype part |
| **Options** | Accesses the SolidWorks **Options** dialog box |
| **Properties** | Accesses the **Properties** dialog box for the current selection |
| **New Window** | Opens another window for the active document |
| **Tile Horizontally** | Arranges windows as non-overlapping horizontal tiles |
| **Tile Vertically** | Arranges windows as non-overlapping vertical tiles |

**Redraw**, which redraws the active window, is a new tool on the View toolbar.

## Repeat Last Command

To repeat the last command, press **Enter** or click **Edit**, **Repeat Last Command**.

The last ten unique commands are displayed in a command history. The list is specific to the active document and to part, assembly, and drawing documents and sketch mode. The most recent command is at the top of the list. You can select any command from the list for your next command.

***To view recent commands or repeat a recent command:***

Right-click in the graphics area and select **Recent Commands**.

# Selection

## Cross Select

When you box select from left to right, only items completely within the box are selected. Now you can cross select from right to left to select items *crossing* the box boundaries in addition to items within the box.

The system gives you feedback as which type of selection you are implementing. When you select from left to right, the box is a solid line. When you select from right to left, the box is a dotted line.

Entities that you could select with the left-to-right box selection in previous releases now are also selected with the right-to-left cross selection, including:

- **Sketches** - sketch entities and dimensions.
- **Drawings** - sketch entities, dimensions, and annotations.

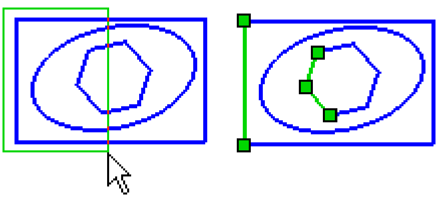
***To select sketch entities:***

1 Open **select\\cross_select_sketch.sldprt** from the **File Explorer** .

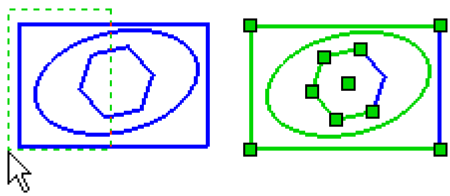
2 Box select from left to right, part way across the sketch entities, and note which entities are selected.

3 Cross select from right to left, crossing the same entities as in step 2, and again note which entities are selected.

In the example, only the left side of the rectangle and two sides of the polygon are selected with the left-to-right box. Three sides of the rectangle, four sides of the polygon, and the ellipse are selected with the cross select.



Box select (left to right)



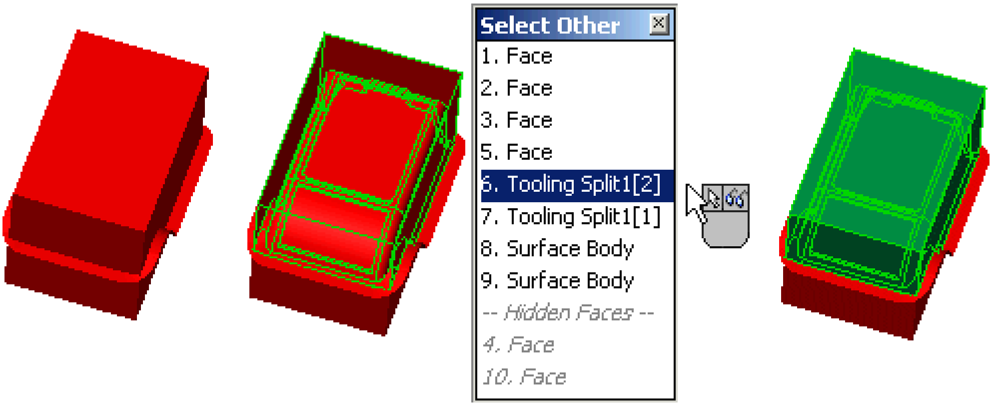
Cross select (right to left)

## Select Other

When you right-click a face and choose **Select Other**, the face disappears and a list of items underneath the pointer is displayed. While in the **Select Other** mode, you can right-click additional faces to remove them. To toggle visibility of a face without completing the operation, hold down **Shift** and right-click the face.

When you hover over an item in the list, the geometry is highlighted. Press **Tab** to cycle through the list. You can select items from the list or in the graphics area.

To exit the command, either select an entity or click in the graphics area outside the model. Selected entities are visible through the faces.




## Quick Snap Selection

When a sketch tool is active, you can specify Quick Snaps to focus the tool on a particular type of snap: **Point Snap**, **Center Point Snap**, and so on. See **Sketch and Quick Snaps** on page 2-5.

## Smart Selection

A new method of selecting edges for Sweep and Loft features allows you to select continuing edges so that the result is considered to be one group of edges. See **Smart Selection** on page 3-2.

## Selection Filters

**Filter Weld Beads**  has been added to the Selection Filter toolbar.

## Multi-user Environment

You can manage write access for shared documents in a multi-user environment.

- The multi-user environment provides read/write access control and tracking for two or more users working with the same files concurrently. Users share a file when they have loaded the same file from the same location.
- Tooltips in the FeatureManager design tree inform you if an assembly, sub-assembly, or component is read-only. If another user with write access has the document open, the tooltip names the user.
- Files that are read-only appear in orange under **Open in SolidWorks** in the Task Pane **File Explorer**.
- If you start to make changes to a read-only document, a message warns you that the document is read-only.
- The system checks the document for changes in status at the interval you specify. When the system detects a change, a tooltip in the lower right of graphics area points to an icon on the status bar. Click the icon to display the **Reload** dialog box. See **Reload** on page 1-15.

***To enable a multi-user environment:***

1 Click **Options** on the Standard toolbar, or click **Tools**, **Options**.

2 On the **System Options** tab, select **Collaboration**.

3 Select **Enable multi-user environment**.

4 Select **Check if files opened read-only have been modified by other users** and set the time interval for checking files. (You can also check the file status manually by clicking **Check Read-only Files** on the Standard toolbar.

You might also want to select options in **Tools**, **Options**, **System Options**, **External References**. For example, select **Open referenced documents with read-only access** so other users can work on components while you have an assembly open, and select **Don't prompt to save read-only referenced documents (discard changes)** to prevent a message box appearing whenever you make a document read-only.

***To set your status for a part or assembly document:***

Click **File**, **Make Read-Only** or **File**, **Get Write Access**.

***To set your status for a component:***

Right-click a component in the FeatureManager design tree or in the graphics area and select **Make Read-Only** or **Get Write Access**.

## Reload

Two new tools on the Standard toolbar access the new dialog box for managing files in a shared environment.

**Reload** - displays the active document in the **Reload** dialog box.

**Check Read-only Files** - checks if files have been made available for write access or have changed on disk since the last reload. A message appears if the files did not change, and the **Reload** dialog box appears if any files changed.

You can also access the dialog box for a component from the FeatureManager design tree or the graphics area. Right-click a component and select **Reload**.

Options in the dialog box include:

- **Show full paths**
- **Show references** (for assemblies and sub-assemblies). The components are indented and listed in the order they appear in the FeatureManager design tree.

The dialog box displays the following columns:

- **Reload**. Select to reload all components.
- **Filename**. Displays the name of the active document and any references (if **Show references** is selected).
- **Read Only**. Shows if files are read-only, and allows you to specify files to become read-only.
- **User With Write Access**. Displays the name of the user with write access if you have read-only status for this document.
- **Needs Save?** If a component has changed and needs to be saved, **Yes** is displayed.
- **Newer Version on Disk?** If the version on disk is newer than the version open in memory, **Yes** is displayed.
- **?** A question mark appears if a context-sensitive Quick Tip message regarding potential loss of data is available.

Once the references are displayed, you cannot hide them again. With references displayed, you can sort the files by **Default** (FeatureManager design tree order), **Filename**, **User with write access**, **Needs save**, or **Newer version on disk**. The file names remain indented when sorted.

Rows displayed in green indicate changes to **Reload** or **Read Only** that will not result in a potential loss of data.

Rows displayed in red indicate that you have selected to reload a file that has changed since it was last saved, or you have selected to make a file read-only when it has changed since it was last saved. Hover the pointer over the question mark to display a Quick Tip explanation.

## Copy Settings Wizard

SolidWorks 2005 automatically retains your settings from earlier releases of SolidWorks. However, to distribute your settings to other computers, you can use the Copy Settings Wizard. The wizard (previously called Copy Options Wizard) now has options to **Save** or **Restore** system settings for:

- **Keyboard shortcuts**
- **Menu customization**
- **System options**
- **Toolbar layout** (**All toolbars** or **Macro toolbar only**)

You can restore settings as follows:

| **Profile** | **Registry** |
|---|---|
| **Current user** | **CURRENT_USER** of current user |
| **One or more network computers** | **LOCAL_MACHINE** of selected computers |
| **One or more roaming user profiles** | **CURRENT_USER** of selected users |

Only system administrators should copy settings to network computers or roaming user profiles. When you restore settings to network computers, the settings apply to *new* SolidWorks users on the specified computers. You can restore settings to roaming user profiles only if your company uses roaming user profiles.

***To save system settings:***

1 In Windows, click **Start**, **Programs**, **SolidWorks 2005**, **SolidWorks Tools**, **Copy Settings Wizard** .

2 In the dialog box, select **Save Settings**, then click **Next**.

3 Browse to a location and file name, select the settings, then click **Finish**.

The settings files have a default extension of **.sldreg**. If you double-click a file with this extension, the Copy Settings Wizard appears.

The wizard confirms that the settings have been written to the specified file.

4 Click **OK**.

***To restore system settings:***

1 In Windows, click **Start**, **Programs**, **SolidWorks 2005**, **SolidWorks Tools**, **Copy Settings Wizard** .

2 In the dialog box, select **Restore Settings**, then click **Next**.

3 Browse to a file from which to restore the settings, select the settings to restore, then click **Next**.

4 Select the destination (**Current user**, **One or more network computers**, or **One or more roaming user profiles**), then click **Next**.

5 If you selected:

- **One or more network computers** - a list of available computers is displayed.
- **One or more roaming user profiles** - a list of available users is displayed.

Select computers or users and click **Add** until you have specified all computers or users, then click **Next**.

6 Select or clear **Create backup of current settings for** <*each_computer_or_user*>, then click **Finish**.

The wizard confirms that the settings have been copied successfully.

7 Click **OK**.

# Options

## File Explorer

In **Tools**, **Options**, **System Options**, **File Explorer**, you can choose whether to show the following folders in the Task Pane **File Explorer**:

- **My Documents**
- **My Computer**
- **My Network Places**
- **Recent Documents**
- **Hidden referenced documents**
- **Samples**

If you select **Hidden referenced documents**, referenced documents that are in memory but not open in SolidWorks are not shown in the Task Pane **File Explorer**. For example, if you open an assembly, any components not open in a SolidWorks window do not appear in the list of documents under **Open in SolidWorks**.

## Collaboration

In **Tools**, **Options**, **System Options**, **Collaboration**, you can enable a multi-user environment and specify an interval to check if read-only files have been modified by other users. See **Multi-user Environment** on page 1-14.

## Macro Editing

In **Tools**, **Options**, **System Options**, **General**, the option **Automatically edit macro after recording** specifies that the macro editor will open after you have recorded and saved a macro.

# Documentation

## Welcome to SolidWorks

When you open SolidWorks 2005 for the first time on a computer, the **Welcome to SolidWorks** dialog box gives you the opportunity to customize dynamic help. You can select:

- **Quick Tips** - to aid new users in getting started.
- **Interactive What's New** - to point out new functionality for experienced users.
- **No dynamic help** - neither Quick Tips nor Interactive What's New.

You can also customize the visibility of toolbars and menus on the basis of work flow in this dialog box. See **Customize by Work Flow** on page 1-10.

You can change these options at any time in the SolidWorks software.

## What's New

The *SolidWorks 2005 What's New* book is available both as an Adobe Acrobat file and as an online HTML file. You can view the online version and the SolidWorks software simultaneously on screen. You can also click next to new menu items and the titles of new and changed PropertyManagers to read what is new about the command.

## Online Tutorials

New lessons on Molds and Weldments have been added to the online tutorial.

When you click a tool in the online tutorial, the corresponding tool in the SolidWorks software flashes. The flashing tools appear with a blue border. When you hover the pointer over the tool image, the border changes to dark blue.

## Overview Topics

Overview topics on Mold Design and Machine Design have been added to the online help. The topics lead you through industry procedures and link those procedures to the SolidWorks software functionality.

## 2D Topics

Topics on sketch relations, inferencing, and 2D drawings in the SolidWorks software have been enhanced in Moving from AutoCAD and in the SolidWorks online help.

2

# Sketching

This chapter describes enhancements to sketching in the following areas:

- General sketch entities
- Offset entities
- Circles
- Sketch and Quick Snaps
- Splines
- Splines on surfaces
- Offset splines
- Infinite lines
- Insert line
- Trims
- Mirrors
- Minimum/Maximum dimensions with arcs
- Sketch relations displays

# General Sketch Entities

Enhancements were added to the following sketch entities:

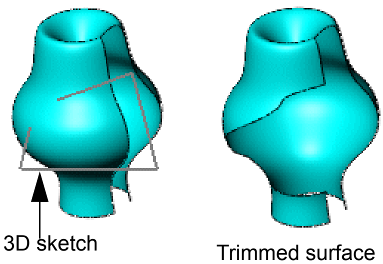
- Autodimension
- Surface trim with 3D sketches

## Autodimension

The **Autodimension** tool on the Dimensions/Relations toolbar allows you to autodimension drawings as well as sketches (see **Autodimension** on page 10-15).

## Surface Trim with 3D Sketch

You can trim surfaces using 3D sketches and the **Trim Surface** tool.




# Offset Entities

The **Offset Entities** tool on the Sketch toolbar includes new options in the PropertyManager under **Parameters**.

> You can apply these options to offset any sketch entity, including splines (see **Offset Splines** on page 2-17).

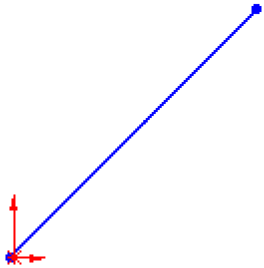
New options include:

### Add dimensions

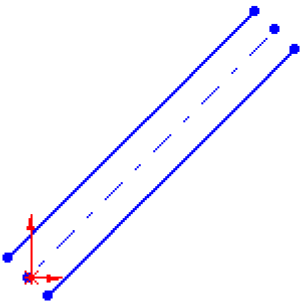
Select **Add dimensions** to include the **Offset Distance** in the sketch. This does not affect any dimensions included with the original sketch entity.

## Make Base Construction

Select **Make base construction** to convert the original sketch entity to a construction line.



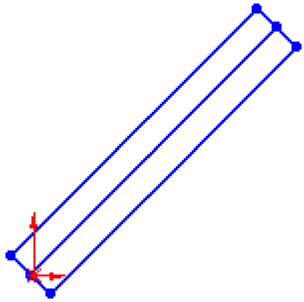
Original sketch



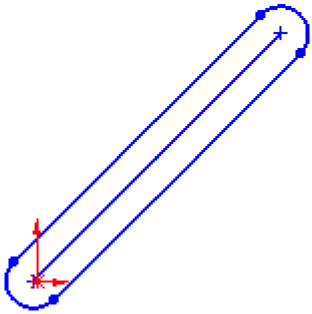
Select Bi-directional and Make base construction

## Cap Ends

Select **Cap ends** and **Bi-directional** to extend the original non-intersecting sketch entities by adding a cap. You can create **Arcs** or **Lines** as extension cap types.



Offset entities with Line caps

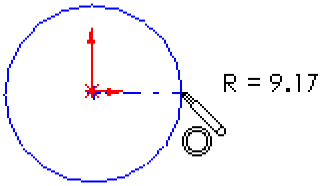


Offset entities with Arc caps

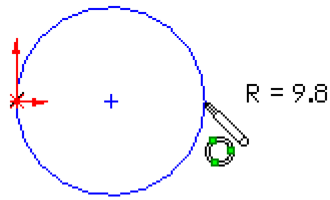
## Circles

You can sketch circles that originate from the perimeter with the **Perimeter Circle** tool. Perimeter circles extend the border of the circle from the perimeter outwards. The **Circle** tool still originates the sketch from the center.

Both circle tools allow you to make the circle tangent to other sketch entities. Both types of circles open the same PropertyManager.



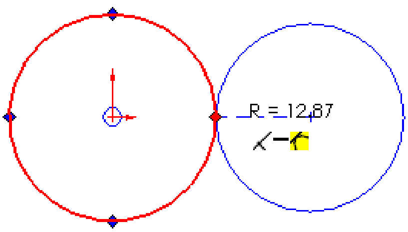

Center creation




Perimeter creation

### Quick Snaps Quadrant Points

When you use the **Circle** tool to originate the sketch from the center, quadrant points are added at 0/360°, 90°, 180°, and 270°. These four quadrant points are Quick Snaps (see **Sketch and Quick Snaps** on page 2-5) that facilitate sketching new circles that are tangent at that point. You can also use the Quick Snaps as the center points for other circles.




Quadrants with second circle tangent to first at 90 degree marker.

Quick Snaps quadrant points also appear on arcs, fillets, parabolas, ellipses, and partial ellipses.

Quick Snaps quadrant points behave as follows:

**Display**. When sketching multiple circles, only the current circle displays the Quick Snaps quadrant points.

**Wake up**. You can wake up any previously sketched circle to display the Quick Snaps quadrant points by using any sketch tool to hover over the circle. Hovering over the circle:

- Displays the Quick Snaps quadrant points.
- Highlights the circle in a different color.
- Displays the pointer with the appropriate inference information.

## Sketch and Quick Snaps

You can enable sketch snaps which behave similar to O-snaps in AutoCAD. Each sketch snap allows you to automatically snap to selected entities as you create the sketch. By default, all **Sketch Snaps** except **Grid** are enabled. Access to **Sketch Snaps** is through the **Quick Snaps** toolbar, the **Quick Snaps** flyout on the Sketch toolbar, or through the shortcut menu.

### Sketch Snaps

***To set Sketch Snaps options:***

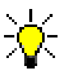
1. Click **Options** on the Standard toolbar, or click **Tools**, **Options**.
2. On the **System Options** tab, click **Relations/Snaps**.
3. Select or clear **Enable snapping** to affect all sketch snaps.
4. Under **Sketch Snaps**, select or clear the appropriate sketch snap to affect only that item.

Sketch snaps and Quick Snaps are related, but not identical. Sketch snaps are global sketch settings that apply to all sketch commands. Quick Snaps are single, independent selections (see **Quick Snaps** on page 2-7).

Sketch snaps include the following:

| Sketch Snaps | Tools | Description |
|---|---|---|
| **Endpoints and sketch points** | | Includes endpoints and sketch points. Snaps to the end of the following sketch entities: lines, polygons, rectangles, parallelograms, fillets, arcs, parabolas, partial ellipses, splines, points, chamfers, and centerlines. |
| **Center Points** | | Snaps to the center of the following sketch entities: circles, arcs, fillets, parabolas, and partial ellipses. |
| **Mid-points** | | Snaps to the midpoints of lines, polygons, rectangles, parallelograms, fillets, arcs, parabolas, partial ellipses, splines, points, chamfers, and centerlines. |
| **Quadrant Points** | | Snaps to the quadrants of circles, arcs, fillets, parabolas, ellipses, and partial ellipses. |
| **Intersections** | | Snaps to the intersections of entities that meet or entities that intersect. |

| Sketch Snaps | Tools | Description |
|---|---|---|
| **Nearest** |  | Supports all entities. When you clear **Nearest**, and all snaps are enabled, your pointer does not need to be in the immediate vicinity of another sketch entity to show inference or snap to that point. When you select **Nearest**, snaps are enabled only when the pointer is in the vicinity of the snap point. |
| **Tangent** |  | Snaps to tangents on circles, arcs, fillets, parabolas, ellipses, partial ellipses, and splines. |
| **Perpendicular** |  | Snaps so the pointer indicates perpendicular when sketching a line to another line. |
| **Parallel** |  | Snaps to lines, arcs, and splines. |
| **Horizontal/vertical lines** |  | Snaps a line sketch vertically to an existing horizontal sketch line, and horizontally to an existing vertical line sketch. |
| **Horizontal/vertical to points** |  | Snaps a line sketch vertically or horizontally to an existing sketch point. |
| **Length** |  | Snaps lines to the increments that are set by the grid, without requiring display of the grid. |
| **Grid** |  | Snaps sketch entities snap to the grid's vertical and horizontal divisions. This is the only sketch snap that is not active by default. |
| **Angle** |  | Snaps to angles. To set the degrees, click **Tools**, **Options**, **System Options**, **Sketch**, select **Relations/Snaps**, and set a value for **Snap angle**. |



All **Sketch Snaps** are available for the sketch entities listed as well as model edges in parts, assemblies, and drawings.

On sketches with multiple entities, only the current sketch entity displays snaps. You can wake up any snap points such as quadrant points, mid-points, and so on, for previously created sketch entities.

### *To wake up Sketch Snaps:*

1. Create any sketch that includes multiple entities.
2. With the any sketch tool selected, hover over the previously sketched entity.
   - The sketch entity changes color.
   - The appropriate Quick Snaps appears on the sketch entity.
   - The pointer changes, adding the appropriate inference information.

## Quick Snaps

Quick Snaps are instantaneous, single operation Sketch Snaps. For example, sketching a line from start to finish, is a single operation.

You can double-click any Quick Snaps on the toolbar to retain the snap capability for multiple instances of the same sketch entity, or until you select a different Quick Snaps.

### *To control Quick Snaps:*

1 In sketch mode, with any sketch tool selected:

- **Menu**. Click **Tools**, **Relations**, **Quick Snaps**.
- **Shortcut menu**. Right-click, select **Quick Snaps.**

Right-click in a sketch, and you can also select **Relations/Snaps Options** to display **System Options**, **Sketch**, **Relations/Snaps**.

- **Toolbar**. Display the Quick Snaps toolbar.

2 Select the appropriate Quick Snaps to shift the focus.

Selecting Quick Snaps from either the toolbar or the shortcut menu filters out other Quick Snaps, enabling you to focus on a particular capability. The **Enable snapping** option under **Tools**, **Options**, **System Options**, **Relations/Snaps**, can be cleared when using Quick Snaps.

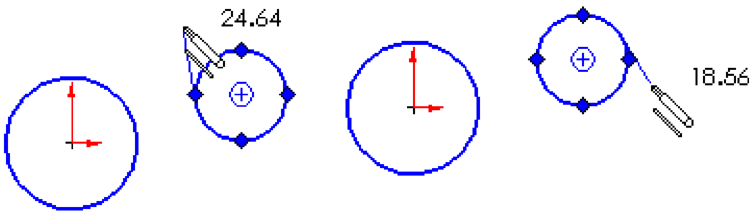
### *To use Quick Snaps:*

1 Click **Circle** on the Sketch toolbar, or click **Tools**, **Sketch Entities**, **Circle** and sketch two, separate circles of any diameter.

2 Click **Line** on the Sketch toolbar, or click **Tools**, **Sketch Entities**, **Line**.

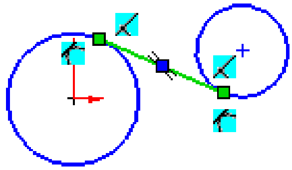
3 Right-click, select **Quick Snaps**, **Tangent Snap**.

4 Starting at any point along the perimeter of either circle, sketch a line and drag it along the perimeter. The line remains tangent at any point along the perimeter of the circle.




5 As you drag the line, extend it to any quadrant point along the perimeter of the second circle.

The line snaps, and is tangent to the perimeter of the second circle.



6 Close the sketch.

# Splines

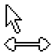
Enhancements to splines include the following areas:

- Sketching a spline
- Controlling the shape
- Adding relations

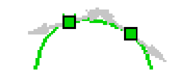
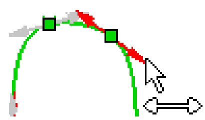
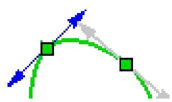
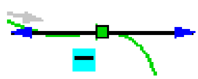
## Sketching a Spline

When you sketch a spline, each spline point displays spline handles. The handles include vectors originating from the spline point.

When you select the spline handle:

- The handle changes color.
- The pointer changes to 
- The appropriate **X Coordinate**  or **Y Coordinate**  highlights in the PropertyManager.

The color of the spline handle depends on its state. Handles can be one color, or the shaft and the arrowheads can be different colors. Some spline handle states include:

| Colors | State of Spline Handles | Example |
|---|---|---|
| Gray | Spline handle is not selected. |  |
| Red | Hover over spline handle with the pointer. |  |
| Green | Select the spline handle. |  |
| Blue | Spline handle is active, with no constraints. You can drag or rotate the spline handle. |  |
| Black shaft with blue arrowheads | Spline handle is active, with some constraints. You cannot rotate the handle, due to the horizontal relation, but you can still drag the handle left and right. |  |

## Splines PropertyManager

New options to control splines are available from the PropertyManager and include the following:

### Tangent Driving

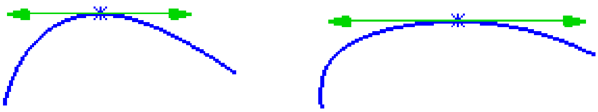
When you drag an arrow on a spline handle, it selects **Tangent Driving** in the PropertyManager. With **Tangent Driving** selected, you control splines using:

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

### Tangent Magnitude

**Tangent Magnitude**  controls the tangency vector by modifying the spline's degree of curvature at the spline point.

Select one of the arrowheads and drag the handles left or right, while maintaining the same direction vector, to diminish the angle. As the angle diminishes, the entities straighten on either side of the through point (the point through which the handles pass).

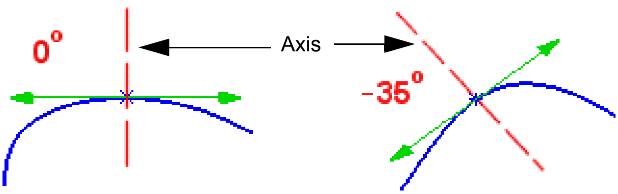


Although the through point and the tangent direction remain constant, the rest of spline may change when you modify the tangent magnitude.

### Tangent Radial Direction

**Tangent Radial Direction**  controls the tangency direction by modifying the spline's angle of inclination relative to the X, Y, or Z axis.

Select one of the arrowheads, and rotate either handle by dragging up or down while minimizing the effect on the tangent magnitude. The spline's angle of inclination increases or decreases accordingly, relative to the X, Y, or Z axis, as shown below.




Although the through point and the tangent direction remain constant, the rest the spline may change when you modify the tangent radial direction.

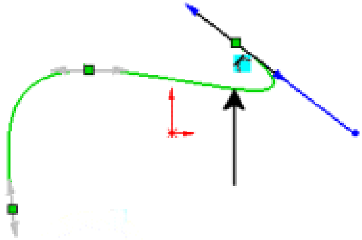
## Flip Relation

You can flip relations in the PropertyManager for tangency and equal curvature relations between the following entities:

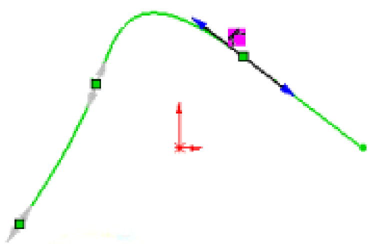
- Splines and lines
- Splines and arcs
- Multiple splines

***To flip relations:***

1. Open **sketching\flip.sldprt** from the **File Explorer**.
2. Right-click **Sketch1** and select **Edit Sketch**.
3. Select the spline from the top sketch (spline and line), as shown.

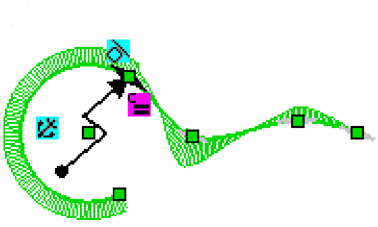
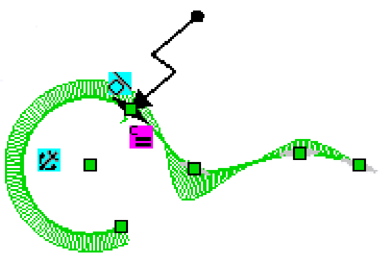


4. In the PropertyManager, under **Existing Relations**, right-click **Tangent0** and select **Flip Relation**.



5. Select the spline from the bottom sketch (spline and arc).
6. Under **Existing Relations**, right-click **Equal Curvature1**, and select **Flip Relation**.

   The curvature comb on the spline flips.
7. Click **OK**.

## Spline Tools and Shortcut Menu Controls

Use the Spline Tools toolbar or right-click on the spline to display the following new options:

### Add Tangency Control

Select **Add Tangency Control** to change the tangency vector by modifying the spline's degree of curvature. New tangency control handles also create a new spline point with an active spline handle.

***To add tangency control:***

1 Select a spline in the graphics area.

2 Select **Add Tangency Control** from the Spline Tools toolbar, or right-click and select **Add Tangency Control**.

An active spline handle drops on to the spline adjacent to the position of your pointer.

3 Drag the pointer following the shape of the spline to position the spline handle.

4 Click the pointer to place the spline handle.

A new spline point appears with an active tangency control handle.

---

To remove the tangency control handle, select the handle and press **Delete**.

---

Adding tangency controls at an existing spline point using the **Add Tangency Control** tool or the shortcut menu, is identical to activating an existing tangency vector control handle in the graphics area. Adding a new tangency control handle selects **Tangent Driving** in the PropertyManager, allowing you to:

- Drag the handles to change the value of **Tangent Magnitude** .
- Rotate the handles to change the value of **Tangent Radial Direction** .

### Add Curvature Control

Select **Add Curvature Control** to manipulate the radius of curvature at any spline point. The curvature control pointer can be on either side of the spline, depending on negative or positive curvature.

***To add curvature control:***

1 Select the spline in the graphics area.

2 Select **Add Curvature Control** from the Spline Tools toolbar, or right-click and select **Add Curvature Control**.

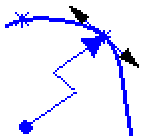
An add curvature pointer drops on the spline. Placement is relative to the closest position of your pointer.

3 Drag the pointer  following the shape of the spline to position the curvature control pointer.

4 Click in the graphics area to place the curvature control pointer.

This adds a new spline point that includes a:

- Curvature control handle
- Tangency control handle



> To remove the curvature control pointer, select the handle and press **Delete**.

## Modifying Curvature Control

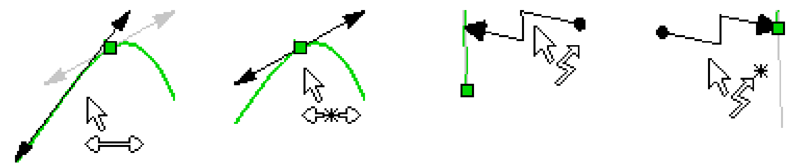
### *To manipulate curvature control:*

To adjust the radius of curvature select the ball of the curvature control pointer and drag it in either direction, along the vector of the handle.

> Spline handles and curvature control pointers tend to snap to the nearest spline point, but you can drag them anywhere along the spline.

Using the pointers to place tangency control and curvature control: handles

Add Tangency Control    Add Curvature Control



Tangency outside spline point    Tangency at spline point    Curvature outside spline point    Curvature at spline point

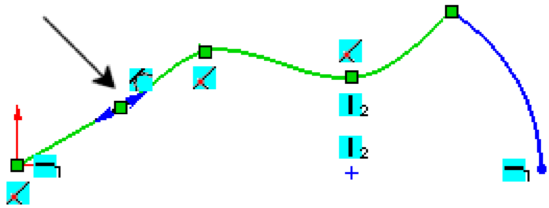
## Show Spline Handles

Show or hide all spline handles.

## Controlling Splines

***To control a spline:***

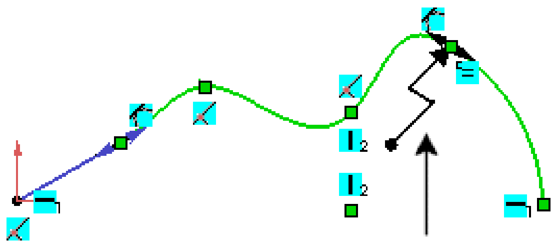
1 Open **sketching\spline_control_new.sldprt** from the **File Explorer** .
2 Right-click **Sketch1** and select **Edit Sketch**.
3 Hold down **Ctrl** and select the line that starts at the origin, and the spline.
4 In the PropertyManager, under **Add Relations**, click **Tangent** .



The tangent relation adds a tangent control handle, and the tangency relations is displayed.

The tangent relation allows you to modify the tangency magnitude, but not the tangency radial direction.

5 In the PropertyManager, right-click in **Selected Entities**, select **Clear Selections**, and then in the graphics area hold down **Ctrl** and select the spline and the arc.
6 Under **Add Relations**, click **Equal Curvature** .

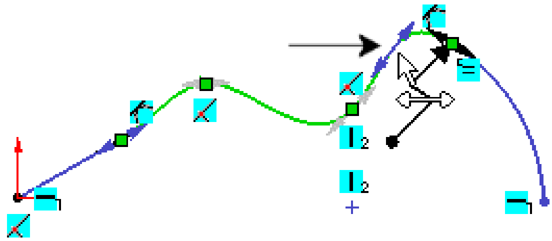


The equal curvature relation adds a curvature control handle, and the equal and the tangency relations is displayed.

The equal relation prevents modification to the curvature.

7 Click **OK** .
8 Select the spline, right-click, and select **Add Tangency Control** .

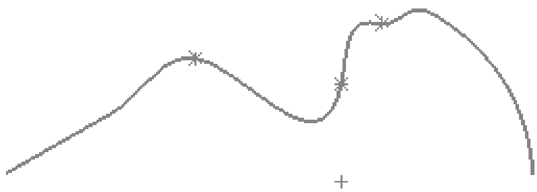
9 Drag the tangency control handle, and click to position it between the third and fourth spline points, as shown.



10 Select the handle, and in the PropertyManager under **Parameters**, set **Tangent Radial Direction** to **0**.

11 Click **OK** and close the sketch.

The final spline should resemble the one below:



## Add Relations to Splines

You can add relations between spline points, between spline handles, and between spline handles and external sketch entities.

For example, you can add a curvature constraint between a spline and an arc so the curvature between the two entities is equal.

In the **Spline** PropertyManager, the **Add Relations** group includes new selections that appear when the relevant geometry exists in the sketch.

<table>
<tr><th colspan="2">Relations with Splines</th></tr>
<tr><th>Constraints to internal spline points</th><th>Constraints to end spline points</th></tr>
<tr><td>Coincident<br>Concentric<br>Perpendicular<br>Tangent<br>Mid-point<br>Fix</td><td>Horizontal<br>Vertical<br>Tangent<br>Coincident<br>Concentric<br>Mid-point<br>Fix<br>Merge</td></tr>
<tr><th>Constraints to internal handles</th><th>Constraints to end handles</th></tr>
<tr><td>Tangent</td><td>Equal Curvature<br>Tangent</td></tr>
<tr><th colspan="2">Constraints between handles</th></tr>
<tr><td colspan="2">Parallel<br>Horizontal<br>Vertical<br>Curvature</td></tr>
</table>

## Restrictions with Splines

The following restrictions exist when controlling splines:

- With 3D splines, curvature control is not available.
- Surface splines cannot cross multiple faces (see **Splines on Surfaces** on page 2-16).

## New Tools on the Spline Tools Toolbar

The **Fit Spline** tool is moved from the Sketch toolbar to the Spline Tools toolbar.

The **Show Curvature Combs** tool was added to the Spline Tools toobar.

## Splines on Surfaces

You can sketch splines on surfaces. Splines sketched on surfaces include standard spline attributes, as well as the following capabilities:

- Add and drag points along the surface.
- Generate preview that is automatically smoothed through the points.

All spline points are bounded to the surface on which they are sketched. Splines cannot cross multiple surfaces.

Benefits of sketching splines on surfaces include:

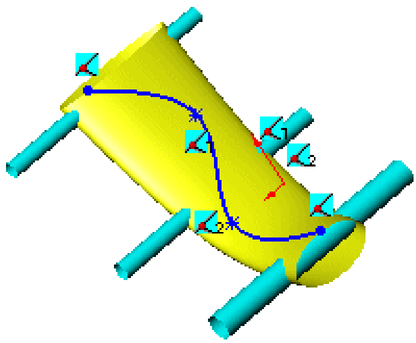
- In part and mold design, surface splines enable you to create more visually accurate parting lines or transition lines.
- With complex sweeps, surface splines facilitate creating guide curves that require curves bounded to surface geometry.

***To sketch a spline on a surface:***

1 Open **sketching\spline-on-surface.sldprt** from the **File Explorer** .

2 Click **Spline on Surface** on the Sketch toolbar, or click **Tools**, **Sketch Entities**, **Spline on Surface**.

3 Sketch a spline on surface that extends from one profile to the other profile.



4 Click **OK** .

You can apply any of the spline controls (see **Splines PropertyManager** on page 2-9) to splines on surfaces.

## Offset Splines

You can offset splines. With offset splines, you can apply these options under **Parameters**:

- **Cap ends**. Adds a cap to the geometry when **Bi-directional** is selected.
- **Make base construction**. Converts the original geometry to construction geometry.

---

You cannot offset a spline from an existing offset. You cannot offset fit splines, ellipses, or parabolas. You cannot create an offset spline if the results of the offset self-intersect, or use offset with infinite lines (see **Infinite Lines** on page 2-18).

---

***To offset a spline:***

1 Open **sketching\offset-spline.sldprt**.

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

3 Click **Offset Entities** on the Sketch toolbar, or click **Tools**, **Sketch Tools**, **Offset Entities**.

4 Select the sketch in the graphics area.

5 In the PropertyManager, under **Parameters**:

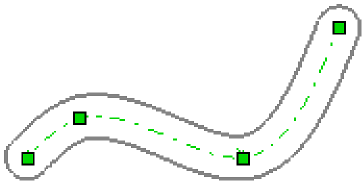
a) Set **Distance** to **5**.

b) Select **Add dimension**.

c) Select **Bi-directional**.

d) Select **Make base construction**.

e) Select **Cap ends**, **Arcs**.



6 Click **OK** and close the sketch.

## Infinite Lines

When sketching lines with the **Line** tool, you can create lines of infinite length. This option is available in the **Insert Line** PropertyManager, regardless of line orientation. Capabilities include:

- Trim lines of infinite length when they intersect other sketch entities using the **Trim Entities** tool (see **Trim** on page 2-20).
- Add relations such as **Horizontal**.

***To create lines of infinite length:***

1 Create a sketch using the **Line** tool.

2 In the PropertyManager, under **Options**, select **Infinite length**.

3 Click **OK** and close the sketch.

## Insert Line

The **Insert Line** PropertyManager enables you to sketch multiple lines. For each line you sketch, you can select different combinations of settings under **Orientation**, **Options**, or **Parameters**.

### Orientation

You sketch lines using any of four orientations including:

- **As sketched**. With the click-drag method, sketch a line in any direction until you release the pointer. With the click-click method, sketch a line in any direction, and continue sketching other lines in any direction, until you double-click.
- **Horizontal**. Sketch a horizontal line until you release the pointer. You can also enter a value for **Length** under **Parameters** to create a horizontal line with that value. To add and display the dimension, select **Add dimensions**.
- **Vertical**. Sketch a vertical line until you release the pointer. You can also enter a value for **Length** under **Parameters** to create a vertical line with that value. To add and display the dimension, select **Add dimensions**.
- **Angle**. Sketch a line at an angle until you release the pointer. The angle is created relative to the horizontal. You can also enter values for **Length** and **Angle** under **Parameters** to create a line with those values. To add and display the dimension, select **Add dimensions**.

## Options

All four line orientations under **Orientation** allow you to select either of the following under **Options**:

- **For construction**. Creates a construction line.
- **Infinite length**. Creates a line of infinite length which you can later trim (**Trim** on page 2-20).

## Parameters

The **Parameters** group appears when you select all orientations except **As sketched**. With **Angle** selected, you can specify angle and length.

---

Some selections under **Orientation** and the **Infinite Length** selection under **Options** override settings for **Quick Snaps**.

---

Once you sketch a line, the **Line Properties** PropertyManager appears. It includes new groups, such as **Add Relations** or **Additional Parameters**. You can:

- Edit the line using selections from the groups in the **Line Properties** PropertyManager.
- Continue sketching using the same line orientation.
- Select a different line orientation by clicking **OK** or by double-clicking. This returns you to the **Insert Line** PropertyManager which includes the **Orientation** group.

## Insert Line with 3D Sketches

With 3D sketches, only **As sketched** is available as a line orientation. Both **For construction** and **Infinite length** are available under **Options**.

# Trim

**Trim Entities** has moved from a dialog box to the PropertyManager. The **Trim Entities** tool includes control over what entities to trim, as well the capability to extend sketch entities.

## Power Trim

Select **Power trim** in the PropertyManager to:

- Extend sketch entities.
- Trim single sketch entities to the nearest intersecting entity as you drag the pointer.
- Trim one or more sketch entities to the nearest intersecting entity as you drag the pointer, and cross the entity.

***To trim using power trim:***

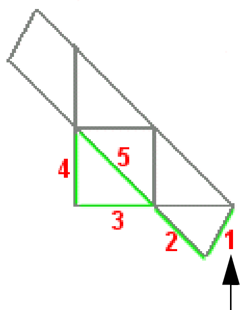
1 Open **sketching\trim_power_trim.sldprt** from the **File Explorer**.

2 Right-click **Sketch1** and select **Edit Sketch**.

3 Click **Trim Entities** on the Sketch toolbar, or click **Tools**, **Sketch Tools**, **Trim**.

4 In the PropertyManager, under **Options**, click **Power trim**.

5 Click in the graphics area, just below the sketch entity (1), as shown, and drag the pointer across the sketch entity.

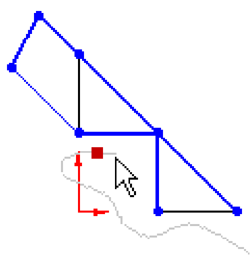



Click under (1), and drag through the sketch entity

A marker appears as it reaches the sketch entity, and disappears as the sketch entity is trimmed.

6 Continue to drag the pointer over the remaining sketch entities (2, 3, 4, and 5), as shown.

A motion trail is created as you drag the pointer and trim the entities.



7 Click **OK** and close the sketch.

## Corner

Select **Corner** to modify two selected entities until they intersect at a virtual corner. A virtual corner is created if you extend one or both entities along their natural path. Factors governing the **Corner** trim option include:

- The sketch entities can be different. For example, you can select a line and an arc, a parabola and a line, and so on.
- Depending on the sketch entities, the trim operation can extend one sketch entity and shorten the other, or extend both sketch entities.
- Behavior is affected by which end of the sketch entities you select. The trim operation can take place at either end of the sketch entities you select.
- Behavior is not affected by the order in which you select the sketch entities.
- If no natural intersection is geometrically possible between the two entities selected, the trim operation is invalid.

***To trim using corner:***

1 Open **sketching\trim_corner.sldprt** from the **File Explorer** .

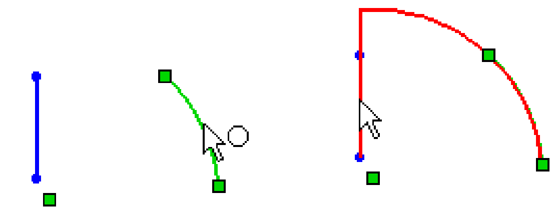
2 Right-click **Sketch1** and select **Edit Sketch**.

3 Click **Trim Entities** on the Sketch toolbar, or click **Tools**, **Sketch Tools**, **Trim**.

4 In the PropertyManager, under **Options**, click **Corner** .

5 Select the arc and the vertical line, as shown below.

The arc and the vertical line extend and meet at the common vertex.



Select the arc.

Select the vertical line to extend both entities

6 Click **OK** and close the sketch.

## Trim away inside

Select **Trim away inside** in the PropertyManager to trim open entities that exist within two selected boundaries.

A closed sketch entity, such as an ellipse, creates a bounding region in the same manner as selecting two open entities as the boundaries.

Factors governing the **Trim away inside** option include:

- The sketch entities you select as the two bounding entities can be different.
- The sketch entities you select to trim must either intersect each bounding entity once, or not intersect the two bounding entities at all.
- The trim action removes any valid sketch entities inside the selected boundaries.
- Valid sketch entities to trim include open sketch segment (but not closed sketch entities such as circles).

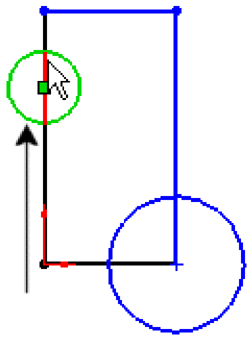
***To trim using trim away inside:***

1 Open **sketching\trim_inside.sldprt** from the **File Explorer** .

2 Right-click **Sketch1** and select **Edit Sketch**.

3 Click **Trim Entities** on the Sketch toolbar, or click **Tools**, **Sketch Tools**, **Trim**.

4 In the PropertyManager, under **Options**, click **Trim away inside** .

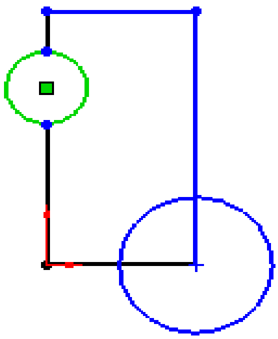


5 Select the small circle, as shown, as the bounding entities.

6 Select the vertical line that intersects the circle, as shown, as the segment to trim.

Because the line intersects the circle, whether you select the line segment inside the circle, or the line segments outside the circle, the inner segment is removed.

7 Click **OK** and close the sketch.



## Trim away outside

Select **Trim away outside** in the PropertyManager to trim open entities that exist outside two selected boundaries. Factors governing the **Trim away outside** option include:

- The sketch entities you select as the two bounding entities can be different.
- Boundaries are not limited by the endpoints of the sketch entities you select. The boundaries are defined as an infinite continuation of the sketch entities.
- The trim action results removes any valid sketch entities that lie outside the selected boundaries.
- Valid sketch entities to trim include open sketch segments (but not closed sketch entities such as circles).

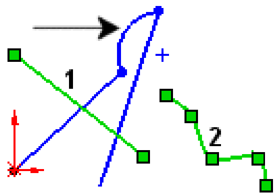
***To trim using trim away outside:***

1 Open **sketching\trim_outside.sldprt** from the **File Explorer** .

2 Right-click **Sketch1** and select **Edit Sketch**.

3 Click **Trim Entities** on the Sketch toolbar, or click **Tools**, **Sketch Tools**, **Trim**.

4 In the PropertyManager, under **Options**, click **Trim away outside** .

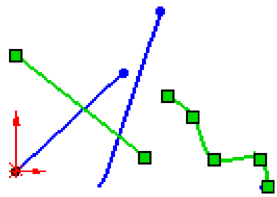



5 Select the angled line (1), and the spline (2) as the bounding entities, as shown.

The boundaries selected are not limited by the endpoints.

6 Select the arc, between the bounding entities, as shown.

The segment is removed.



7 Click **OK** and close the sketch.

## Trim to Closest

Select **Trim to closest** in the PropertyManager to trim or extend the selected sketch entities. Factors governing the **Trim to closest** trim option include:

- Remove the selected sketch entity up to the closest intersection with another sketch entity.
- Extend the selected entity. The direction in which the entity extends, depends on the direction you drag the pointer.

***To trim using trim to closest:***

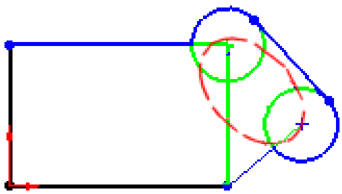
1 Open **sketching\sketch_trim_closest.sldprt** from the **File Explorer** .

2 Right-click **Sketch1** and select **Edit Sketch**.

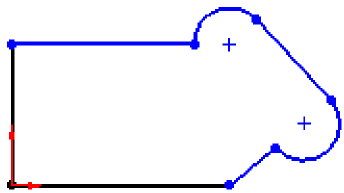
3 Click **Trim Entities** on the Sketch toolbar, or click **Tools**, **Sketch Tools**, **Trim**.

4 In the PropertyManager, under **Options**, click **Trim to closest** .

5 Select each of the sketch entities, as shown, so that the final sketch only includes a single, exterior contour.



Select the sketch segments that are highlighted within the ellipse.



The resulting sketch is a single, exterior contour.

6 Click **OK** and close the sketch.

## Mirror

**Mirror Entities** has moved from a dialog box to the PropertyManager. The **Mirror Entities** tool now includes the following capabilities:

- Mirror to only include the new entity, or both the original and the mirrored entity.
- Mirror some or all of the sketch entities.
- Mirror about any type of line, not just a construction line.
- Mirror about edges in a drawing, part, or assembly.

The Sketch toolbar also includes the **Dynamic Mirror Entities** tool that allows you to preselect a sketch entity to mirror about, then sketch additional entities. The **Dynamic Mirror Entities** tool requires that you use a centerline to mirror about.

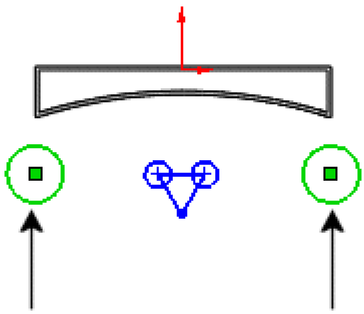
***To mirror selected entities:***

1 Open **sketching\mirror.sldprt** from the **File Explorer**.

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

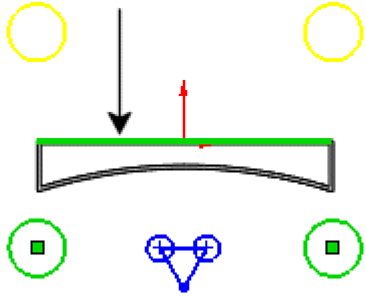
3 Click **Mirror Entities** on the Sketch toolbar, or click **Tools**, **Sketch Tools**, **Mirror**.

4 In the PropertyManager, under **Options**, select the two outer circles, as shown, for **Entities to mirror**.

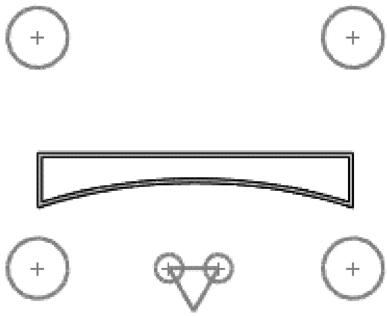


5 Select **Copy**.

6 Click in **Mirror about**, and then select the horizontal edge, as shown.



7 Click **OK** and close the sketch.

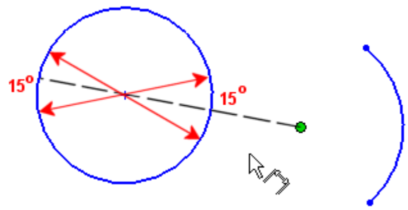


## Minimum/Maximum Dimensions with Arcs

When you add dimensions between an arc and any other sketch entity using the **Smart Dimensions** tool, you can create linear and ordinate dimensions to minimum, center, and maximum arc extents.

The minimum and maximum arc extents are confined to 15° zones. The minimum zone is located closest to the entity you measure from. When you move beyond the 15° zones, the dimension reverts to a linear dimension between the center of the arc and the other sketch entity.

The dimension between two arcs provides two 15° zones where you can dimension to the perimeter of the arc as opposed to the center of the arc. The 15° zones are based on an imaginary line between the centers of the arcs. When you dimension between another sketch entity and an arc, the imaginary line is from the initial position on the sketch entity to the center of the arc.

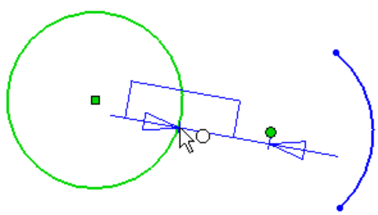



***To dimension to the arc:***

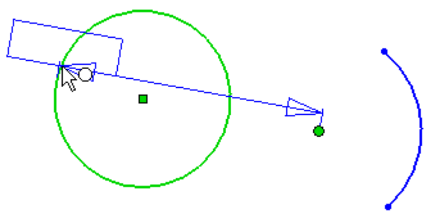
1 Sketch a circle and an arc.

2 Click **Smart Dimensions** on the Dimensions and Relations toolbar, or click **Tools**, **Dimensions**, **Smart**.

3 Select the center of the arc, and move the pointer on a straight line towards the center of the circle until you reach the closest perimeter of the circle (minimum).



Minimum arc dimension



Maximum arc dimension

4 Move the pointer up and down along the perimeter, staying within the 15° zone. Move the pointer across to the other side of the circle (maximum). Note that an equivalent 15° zone exists.

5 Move outside the 15° zone on either side of the perimeter of the circle, and the dimension changes to linear between the two arc centers.

## Sketch Relations Display

You can now simultaneously display multiple relations between multiple sketch entities. Display enhancements are available when you select:

- **Edit Sketch**. Display symbols representing relations for all sketch entities in the graphics area.
- **Display/Delete Relations** . Display and highlight symbols in the graphics area by selecting the item under **Relations** in the PropertyManager.

***To automatically display sketch relations:***

Click **View**, **Sketch Relations**.

When appropriate, the symbols in the graphics area include numerical subscripts to indicate relations between two corresponding entities. The subscript also identifies the incidence (starting with **0**) of that relation, as listed in the **Display/Delete** PropertyManager under **Relations**.

Edit Sketch mode

The sketch on the right includes various symbols that represent relations.

Symmetric to corresponding symbol with same subscript

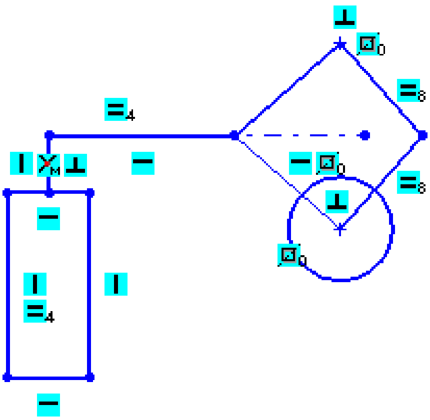
Horizontal entities

Vertical entities

Midpoint of line entities

Equal to corresponding symbol with same subscript

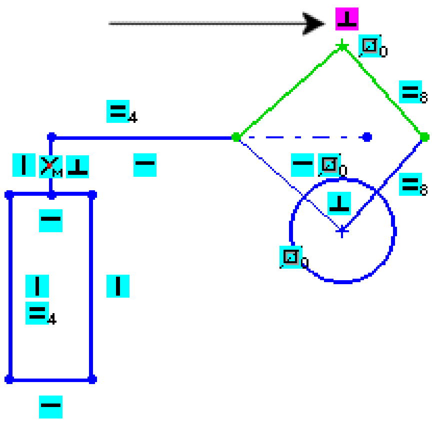
Perpendicular entities



When you display relations symbols with the **Display/Delete** PropertyManager, you can select any item under **Relations** to highlight the relevant sketch entities and relation symbols.

Display/Delete Relations

The sketch on the right highlights the perpendicular relation between the two lines.



3

# Features

This chapter describes enhancements to features in the following areas:

- General
- Deforms
- Extrusions
- Fillets
- Flexes
- Indents
- Lofts
- Mirrors
- Move/Copy Bodies
- Split Line curves
- Sweeps

# General

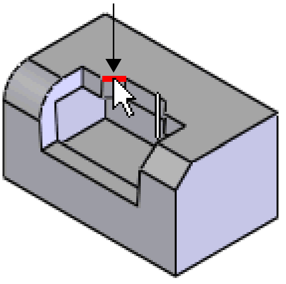
## Smart Selection

Smart Selection is a new shortcut menu item that enhances edge or guide curve selection when you create lofts or sweeps. Smart selection can propagate edge selection along these edge types:

- Chains
- Tangents
- Open loops

***To use smart selection to create a lofted surface:***

1. Open **SmartSelection.sldprt** from the **File Explorer**.
2. Click **Lofted Surface** on the Surfaces toolbar, or click **Insert**, **Surface**, **Loft**.



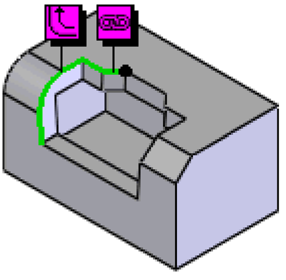
3. In the PropertyManager, under **Profiles**, right-click the edge shown in the graphics area for **Profile** and select **Start Smart Selection**.

   You must select exactly on the edge or the item does not appear in the menu.

   A **Chain** callout appears, indicating smart selection has started for a chain. In the PropertyManager, **Smart Selection<1>** appears for **Profile**.

4. Move the pointer over a non-contiguous edge and note that the edge does not highlight because it is not an acceptable selection to chain. Only valid smart selection edges highlight when you move the pointer over them.
5. Select the next two edges to the left of the selected edge to add them to the chain.

   A **Tangent** callout appears, indicating the next smart selection edges are tangent.

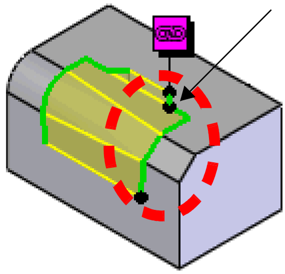


6. Click the **Tangent** callout.

   The chain extends to select all tangent edges. The **Tangent** callout changes color to indicate it is selected.

7. Right-click anywhere in the graphics area and select **End Smart Selection**.

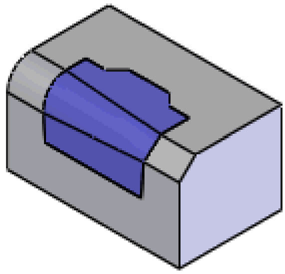
8 Use smart selection to select the five opposing edges for **Profile**, starting from the top edge.



**Smart Selection <2>** is listed for the new profile name. A preview of the surface loft appears.

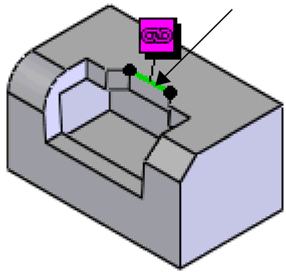
> If you have trouble creating the surface loft or activating smart selection, zoom to the area to make sure you pick the correct edges.

9 Click **OK**, then close the file without saving changes.



***To use smart selection to trim surface lofts:***

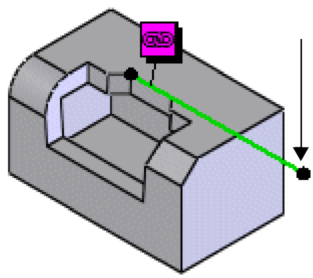
1 Open **SmartSelection.sldprt** from the **File Explorer**.



2 Click **Lofted Surface** on the Surfaces toolbar, or click **Insert**, **Surface**, **Loft**.

3 In the PropertyManager, under **Profiles**, right-click the top edge shown in the graphics area for **Profile** and select **Start Smart Selection**.

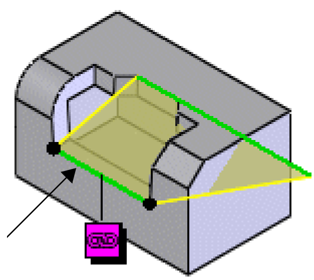
4 Drag the black circular handle on the right end of the edge to extend it approximately as shown.



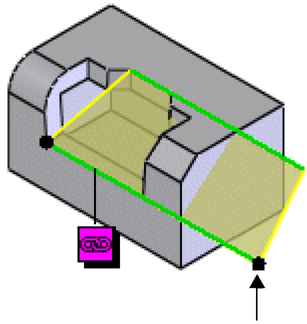
> You can extend or shorten the edge length to adjust its size.

5 Right-click anywhere in the graphics area and select **End Smart Selection**.

6 Right-click the bottom edge in the graphics area and select **Start Smart Selection**.



A preview of the surface loft appears.



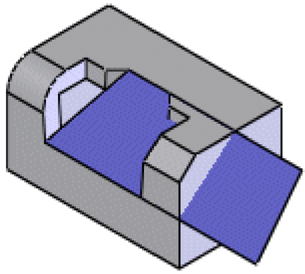
7 Drag the black circular handle on the right end to extend it approximately as shown.

The loft preview adjusts as you trim the profile line.

8 Right-click anywhere in the graphics area and select **End Smart Selection**.

9 Click **OK**.

The surface loft is trimmed to the adjusted smart selection profile.



## Deforms

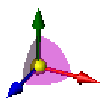
Enhancements to the deform feature include:

- A new **Surface Push** deform type
- New **Curve to curve** match tangency options

### Surface Push Deform Type

The new **Surface Push** deform type deforms a target body by pushing a tool body through it. You select a customizable pre-built tool body, such as a polygon or sphere, or use your own tool body. Use the triad callout in the graphics area to size the tool body.

Wherever the triad is in the graphics area, it controls the tool body movement. Drag the triad to push the tool body into the target body to deform the target body.



If the triad is positioned outside the model body, when you drag starting from the triad, only the tool body moves; the triad does not move.

To restrict the tool body axis of movement to a specific axis, drag a triad handle: red X axis, green Y axis, or blue Z axis. To rotate the tool body around a specific axis, right-click a triad handle and drag. The pointer guides your action: left-click pans the tool body, while right-click rotates it. To move the tool body on a specific plane, drag the triad in the area between the axes handles.

To set a specific tool body location, use the **Delta X**, **Delta Y**, or **Delta Z** settings under **Tool Body Position** in the PropertyManager. You can align the triad to the target body or return it to the original position with shortcut menu options.

***To create a surface push deform:***

1 Open **Deform_Push.sldprt** from the **File Explorer**.

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

3 In the PropertyManager, under **Deform Type**, select **Surface push**.

4 Under **Push Direction**:

a) Select the top of the body in the graphics area for **Deform direction**.

A handle points upwards, indicating that the tool body pushes normal to the selected face.

b) Select **Show preview**.

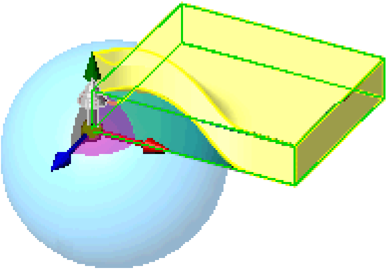
5 Under **Deform Region**:

a) Select anywhere on the body in the graphics area for **Bodies to be deformed**.

b) Select **Sphere** in **Tool Body**.

c) Set **Deform deviation** to **5** to shape the transition area between the tool body and the target body.

A preview appears of the tool body deforming the target body.

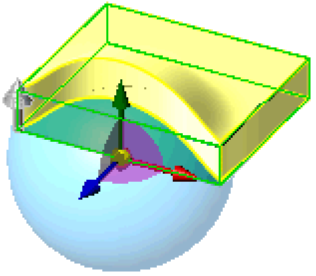


6 In the graphics area, drag the white area of the callout so the entire triad is visible, then set the callout radius to **75** to set the size of the sphere.

Now use the triad to shape the deform feature.

1 Select the triad's red handle and drag the sphere along the X axis.

The sphere moves along the X axis, deforming the target body.



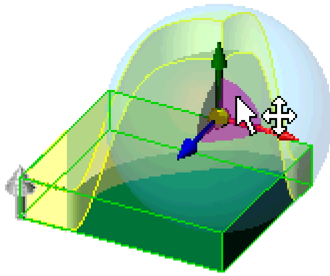
If only the top face seems to deform in the preview, under **Deform Region**, make sure you have not selected a face under **Additional faces to be deformed**.

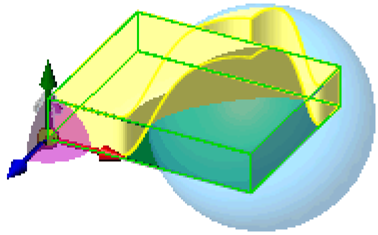
2 Drag the triad from between the green and red arrows to deform the body along the X-Y plane.

3 In the PropertyManager, under **Tool Body Position**, set **Delta Y** to **60**.

The tool body repositions itself to 60mm above the Y axis starting position.

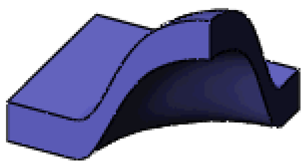
4 Right-click the triad center and select **Align to component**.





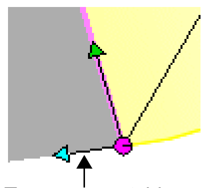
The triad returns to the start position, which is the origin of the body being deformed.

5 Click **OK**.



## Curve to Curve Match Tangency Options

When you create a deform feature with the **Curve to curve** option, you can match tangency between surfaces. The new blue arrows indicate the direction of tangency matching. You can use a **Curve direction** or **Surface tangent** match.




For a **Curve direction** match, the body is deformed by rotating the tangent direction vector of the initial curve to match the tangent direction vector of the target curve. The body is deformed normal to the curve, so the body thickness is maintained.

For a **Surface tangent** match, the body is deformed by rotating and matching the surface tangent of the initial curve to the surface tangent of the target curve.

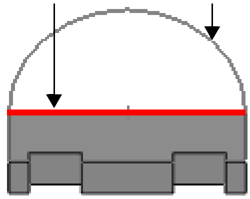
***To use the Curve direction option:***

1 Open **Deform_CurveDirection.sldprt** from the **File Explorer**.

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

3 In the PropertyManager, under **Deform Type**, select **Curve to curve**.

4 Under **Deform Curves**, select the lines shown for **Initial curves** and **Target curves**.



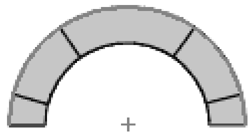

5 Under **Deform Region**:

- Select **Fixed edges** and **Uniform**.
- Select the body for **Bodies to be deformed**.

6 Under **Shape Options**, select **Curve direction** for **Match**.

7 Click **OK**.

The body is deformed normal to the curve while the body thickness is maintained.



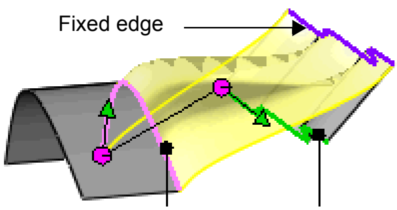
Deformation using Curve direction option

*To use the Surface tangent option:*

1 Open **Deform_SurfaceTangent.sldprt** from the **File Explorer**.

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

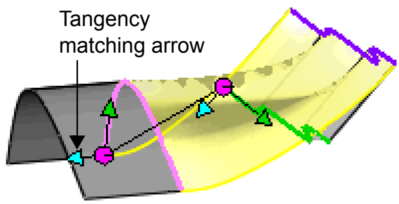
3 In the PropertyManager, under **Deform Type**, select **Curve to curve**.




4 Under **Deform Curves**, select the lines in the graphics area as shown for **Initial curves** and **Target curves**.

5 Under **Deform Region**:

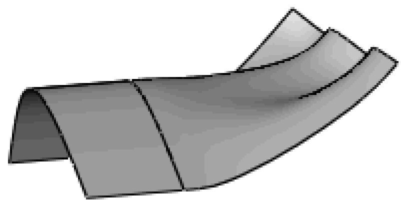
- Select **Fixed edges** and **Uniform**.
- Click in **Fixed curves/edges/faces** and select the fixed edge as shown.




6 Under **Shape Options**, select **Surface tangent** for **Match**.

The preview shows the tangent surface approaching the edge from the opposite direction from what is desired.

7 Select **Reverse tangent** so both blue tangency matching arrows point towards the left as shown.



8 Click **OK**.

## Extrusions

You can specify the start plane of an extrusion to be a surface, face, plane, vertex, or an offset plane parallel to the sketch plane. Previously, extrusions always started from the sketch plane.

***To create an extrusion that starts from an offset plane:***

1 Open a new part document and click **Extruded Boss/Base** on the Features toolbar, or click **Insert**, **Boss/Base**, **Extrude**.

2 Sketch a rectangle on the **Front** plane, then exit the sketch.

3 In the PropertyManager, under the new **From** box:

a) Select **Offset** in **Start Condition**.

b) Set **Enter Offset Value** to **20**.

The preview shows the extrusion starting on a plane that is offset from the **Front** plane by 20mm.

4 Click **OK**.

## Fillets

You can create face fillets with a constant width by selecting the new **Constant width** option in the **Fillet** PropertyManager.

***To create a face fillet with a constant width:***

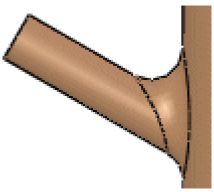
1 Open **ConstantWidthFillet.sldprt** from the **File Explorer**.

2 Click **Fillet** on the Features toolbar, or click **Insert**, **Features**, **Fillet/Round**.

3 In the PropertyManager, under **Fillet Type**, select **Face fillet**.

4 Under **Items To Fillet**:

- Set **Radius** to **3**.
- Select one cylinder for **Face Set 1**, and the other for **Face Set 2**.

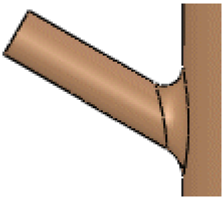


5 Click **OK**.

The fillet has a variable width.

6 Right-click **Fillet1** in the FeatureManager design tree and select **Edit Feature**.

7 In the PropertyManager, under **Fillet Options**, select **Constant width**.



8 Click **OK**.

The fillet has a constant width.

## Flexes

The new flex feature bends, twists, tapers, or stretches a solid or surface body to a set value around an area specified by two user-defined planes. Areas outside the specified area retain their geometry. Use the triad to position the flex feature's central point. The triad start position is the model's center of gravity. With the shortcut menu, you can align the triad to one of the bounding planes or reset the flex feature.

Only one type of flex is permitted per flex feature. To apply more than one type of flex, create separate flex features.

***To bend and twist a solid body using flex features:***

1 Open **Flex.sldprt** from the **File Explorer**.

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

3 In the PropertyManager, under **Flex Input**:

a) Select the body in the graphics area for **Bodies for Flex**.

> You can add flex features to multiple bodies.

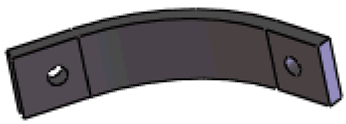
b) Select **Bending** for the flex type.

c) Select **Hard edges** to create the feature using analytical geometry.

4 Under **Trim Plane 1** and **Trim Plane 2**, set the **Plane 1** and **Plane 2 trimming distance** to **20** so the bend area starts 20mm from each edge.

5 Under **Flex Input**, set **Angle** to **60** to bend the body 60 degrees.

A preview of the bend appears.



6 Click **OK**.

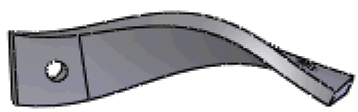
Add another flex feature to twist the body.

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

2 Select the body for **Bodies for Flex**.

3 Select **Twisting** for the flex type.

4 Leave the **Trim Plane 1** and **Trim Plane 2** trimming distance at **0** so the entire body twists.



5 Under **Flex Input**, set **Angle** to **100** to twist the body 100 degrees.

6 Drag the triad in the graphics area back and forth along the body to see how this affects the twist.

> To restrict movement to a specific axis, drag one of the colored triad handles. See **Surface Push Deform Type** on page 3-4 for more information about the triad.

7 Click **OK**.

## Indents

The new indent feature creates a pocket inside a target body that closely matches the contour of a selected tool body. You specify the clearance between the target body and the tool body, and the thickness of the area deformed by the indent feature. The indent feature can deform or cut material from the target body.

If the tool body is a surface, and you are cutting material, a manipulator appears to control the cut direction.

The indent feature uses the form of the tool body to create a pocket in the target body, so more faces, edges, and vertices appear in the final body than in the original body. This differs from the deform feature, where the number of faces, edges, and vertices remains unchanged in the final body.

The indent feature shape updates if you change the shape of the original tool body used to create the pocket.

Note these requirements:

- To deform material, the target body must be in contact with the tool body. To cut material, the target and tool bodies do not have to be in contact with each other.
- The target body or the tool body must be a solid body.

***To create a indent feature:***

1 Open **Indent.sldprt** from the **File Explorer** .

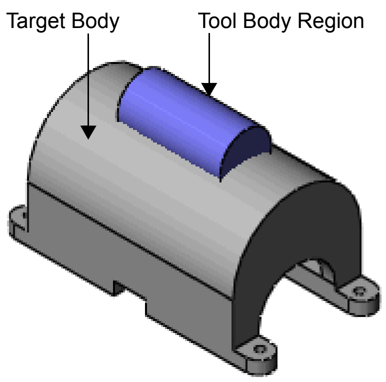
2 Click **Indent** on the Features toolbar, or click **Insert**, **Features**, **Indent**.

3 In the PropertyManager, under **Selections**:

a) Select the body shown in the graphics area for **Target Body** .

b) Select **Keep Selections** to retain the selected target body material.

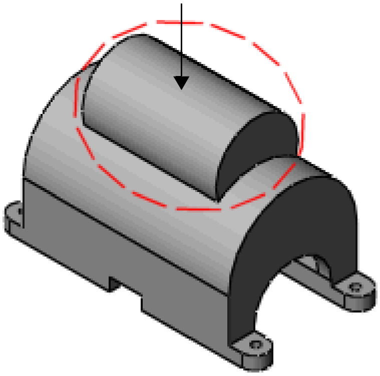
c) Select anywhere on the extruded feature in the middle of the target body, as shown, for **Tool Body Region** .

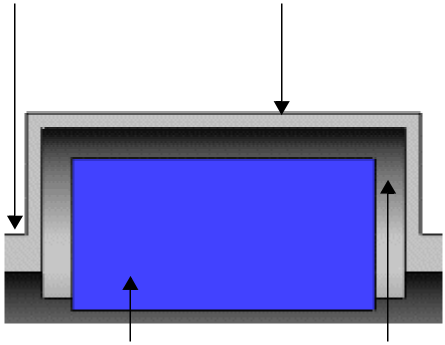


4 Under **Parameters**:

- Set **Thickness** to **2** to set the indent feature thickness.
- Set **Clearance** to **4** to create a 4mm clearance between the tool body and the target body.

5 Click **OK** .




Front section view of indent feature

# Lofts

Enhancements to the loft feature include:

- General enhancements
- Curvature continuity
- Guide curve alignment
- Guide curve influence
- Individual segment weight control
- Mesh previews
- Synchronization

## General Enhancements

### Centerlines

Centerlines can co-exist with guide curves. Previously, you could not use centerlines with guide curves.

### PropertyManager Design

In addition to changes to the **Loft** PropertyManager indicated in the Lofts section, note these changes:

Under **Start/End Constraints**:

- The **Start/End Tangency** box has been renamed to **Start/End Constraints** to reflect the new **Curvature to Face** option.
- **Start tangency type** and **End tangency type** have been renamed to **Start Constraint** and **End Constraint**.

Under **Options**:

- The **Maintain tangency** option has been renamed to **Merge tangent faces** to better reflect its functionality, which is unchanged.
- **Advanced smoothing** has been removed because this functionality is now automatically applied.

## Curvature Continuity

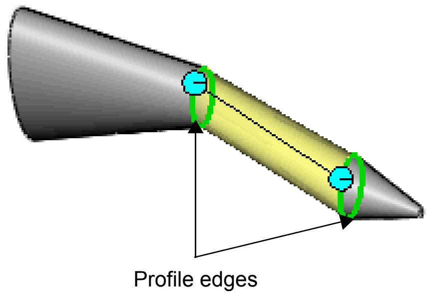
A new **Curvature to Face** constraint is available for the start and end profiles. This curvature creates a smoother, more visually appealing loft than was previously possible.

***To create a loft using Curvature to Face:***

1 Open **Loft_CurvatureContinuous.sldprt** from the **File Explorer** .

2 Click **Lofted Surface** on the Surfaces toolbar, or click **Insert**, **Surface**, **Loft**.

3 In the PropertyManager, under **Profiles**, select the two edges shown in the graphics area for **Profile** .

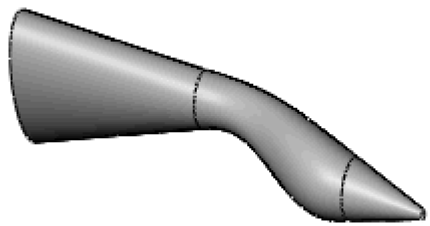



> Select the profiles in locations directly across from one another so the loft geometry is not self-intersecting.

The preview shows abrupt changes at the start and end of the loft as it meets the surfaces

4 In the PropertyManager, under **Start/End Constraints**, select **Curvature To Face** in **Start constraint** and **End constraint**.

5 Click **OK** .



The loft is smoother and more visually appealing because a continuous curvature (C2 continuity) is applied to the loft and surfaces.

## Guide Curve Alignment

You can specify the alignment vector and an angle to apply to the loft at guide curves. This lets you maintain a draft angle along the sides of a loft or ensure that the highest point of a loft remains tangent to a guide curve.

***To specify the alignment vector and apply an angle to the loft at a guide curve:***

1 Open **Loft_GuideCurveAlign.sldprt** from the **File Explorer**.

2 Click **Draft Analysis** on the Mold Tools toolbar, or click **Tools**, **Draft Analysis**.

3 In the PropertyManager, under **Analysis Parameters**:

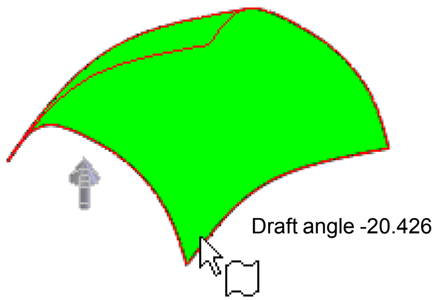
a) Select **Top Plane** in the FeatureManager design tree for **Direction of Pull**.

b) Set **Draft Angle** to **15**.

c) Click **Calculate**.

4 In the graphics area, move the pointer along the edge defined by the guide curve.

The pointer reports the draft angle as approximately 20 degrees.




5 Click **Cancel**.

Now set the draft angle at the guide curve.

1 Right-click **Surface-Loft1** in the FeatureManager design tree and select **Edit Feature**.

2 In the PropertyManager, under **Guide Curves**:

a) Select **GuideCurve1** in **Guide Curves**.

b) Select **Direction Vector** in **GuideCurve1-Tangency** for **Type**.

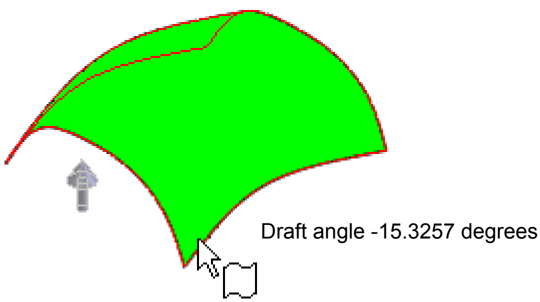
c) Select **Top Plane** in the FeatureManager design tree for **Direction Vector** so the draft is applied normal to that plane.

d) Set **Draft angle** to **15**.

3 Click **OK**.

4 Use the draft analysis tool again with the same **Analysis Parameters** to check the draft along the guide curve edge.

The draft angle is approximately 15 degrees whenever geometrically possible.



## Guide Curve Influence

You have more control of guide curve influence than was previously possible, which allows you to create specific geometry with a minimal number of guide curves.

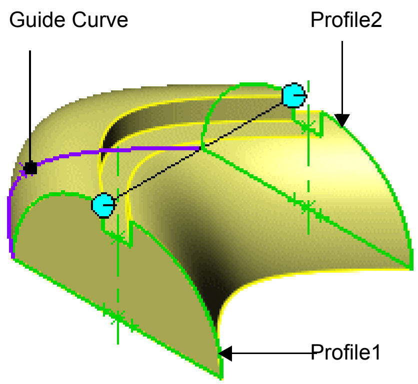
In the Loft PropertyManager, under **Guide Curves**:

- **Guide influence** has been renamed to **Guide curves influence** to reflect enhanced control of guide curves.
- The options **Default** and **Local** have been removed. New options, **To next guide**, **To next sharp**, and **To next edge** provide more control than was previously possible.

***To control guide curve influence:***

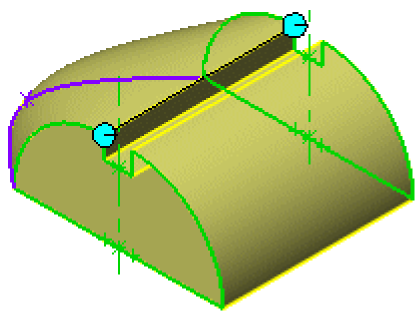
1. Open **Loft_GuideCurveInfluence.sldprt** from the **File Explorer**.
2. Click **Lofted Boss/Base** on the Features toolbar, or click **Insert**, **Boss/Base**, **Loft**.
3. In the PropertyManager, under **Profiles**, select the sketches **Profile1** and **Profile2** in the FeatureManager design tree for **Profile**.
4. Under **Guide Curves**, click in **Guide Curve** and select the sketch **Guide Curve** in the FeatureManager design tree.

   The preview shows that the entire loft is influenced by the guide curve.




5. Under **Guide Curves**, select **To Next Sharp** in **Guide curves Influence**.

   The guide curve region of influence extends to only the next sharp in the profile. Only one side of the loft is influenced by the guide curve.



6. Click **OK**.

## Individual Segment Weight Control

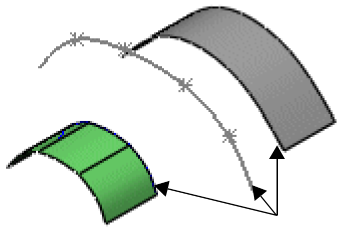
When you apply start or end constraints, you can apply a weight to the constraints for individual segments within the profile, or to the entire profile. The new **Draft angle** option lets you add draft to a loft at the start and end constraint profile.

***To apply individual segment weight control:***

1 Open **Loft_WeightControl.sldprt** from the **File Explorer**.

2 Click **Lofted Surface** on the Surfaces toolbar, or click **Insert**, **Surface**, **Loft**.

3 In the PropertyManager, under **Profiles**, select the three edges in the graphics area in the approximate locations as shown for **Profile**.



4 Under **Start/End Constraints**:

a) Select **Curvature To Face** in **Start constraint**.

b) Clear **Apply to all**.

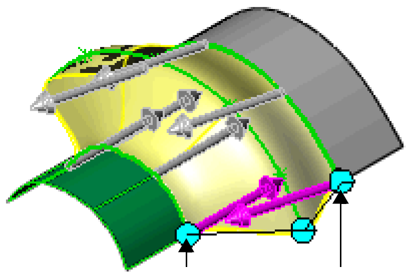
c) Select **Direction Vector** in **End constraint**.

d) Select the two vertices shown for **Direction Vector** .

e) Set **Draft angle** to **2** to draft the loft at the end constraint.

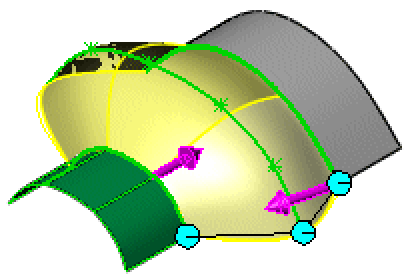
f) Clear **Apply to all**.

If necessary, click **Reverse Tangent Direction** under **Start** or **End Constraint** so the handles point toward each other.




5 Drag individual handles on either the start or end profiles to change the weight for that profile segment.

The **Start** or **End Tangent Length** values update in the PropertyManager.

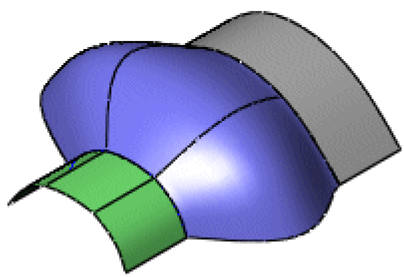


6 Under both **Start constraint** and **End constraint**, select **Apply to all**.

The individual handles for each constraint profile are replaced by one handle that controls each entire constraint profile. The tangent lengths adjust to the last setting in the PropertyManager.

7 Drag the two handles to adjust the constraints.

8 Click **OK** .



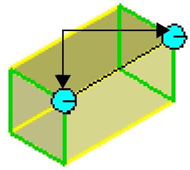
## Mesh Previews

You can apply a preview mesh on the B-spline (non-analytic) surfaces of lofts to better visualize the loft surface. You can apply mesh to selected faces or to all faces.

***To use mesh previews in lofts:***

1 Open **Loft_Mesh.sldprt** from the **File Explorer**.

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



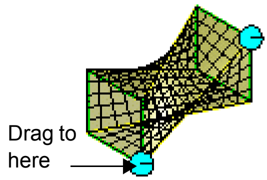
3 In the PropertyManager, under **Profiles**, select the upper-right corner of **Sketch1** and **Sketch2** in the graphics area for **Profile**.

4 Right-click anywhere in the graphics area and select **Mesh Preview**, **Mesh All Faces**.

No preview mesh appears because the loft uses analytic geometry.

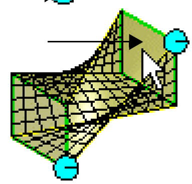
5 Drag the connector handle at the top-right corner of the near face to the lower-right corner.

The mesh appears because the loft uses B-spline geometry.



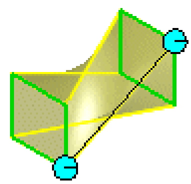

6 Right-click the face of the preview as shown and select **Mesh Preview**, **Clear Meshed Faces**.

The mesh disappears from the selected face only.



7 Right-click in the graphics area and select **Mesh Preview**, **Clear all Meshed Faces** to clear the mesh preview from all faces.

8 Click **OK**.



## Synchronization

Synchronization enhancements using the shortcut menu include:

- **Undo**. You can undo up to six of the last delete, add, or drag connector commands.
- **Delete**. You can delete a connector.
- **Show a Connector**. This new option shows the nearest connector to the selected point. The **Add Connector** command, which previously only showed the nearest connector, now adds a connector where you select on the profile sketch.

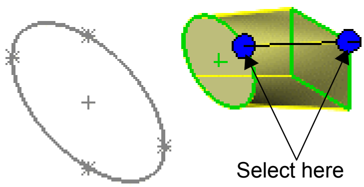
- **Persistent Connectors**. Connector points that you reposition retain the new position when you add profiles to the loft. Previously, repositioned points reverted to their previous position when you added profiles.
- **Handle color in chains**. Handles within the same chain highlight so you can more easily see the connections. You can also set the color of the starting chain. Click **Tools**, **Options**, **System Options**, **Colors**. In **System colors**, **Dynamic highlight** sets the color of the chain when you hover over it, and **Selected Item 4** sets the color for the starting chain.

***To test the loft synchronization enhancements:***

1 Open **Loft_Synchronize.sldprt** from the **File Explorer** .

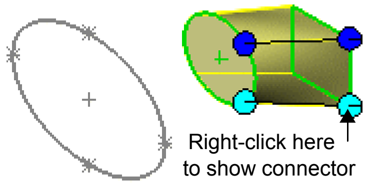
2 Click **Lofted Boss/Base** on the Features toolbar, or click **Insert**, **Boss/Base**, **Loft**.

3 In the PropertyManager, under **Profiles**, select sketches 1 and 2 in the graphics area in the approximate locations as shown for **Profile** .

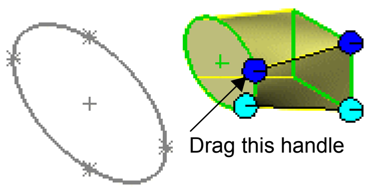



4 Right-click near the lower-right corner of the square sketch and select **Show Connector**.

The nearest connector appears.




5 Drag the upper connector handle on the ellipse as shown to reposition it.



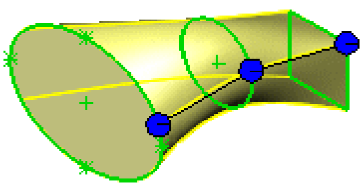

6 Right-click in the graphics area and select **Undo connector edit**.

The handle returns to its original position.

7 Drag the same handle to reposition it again.

8 Under **Profiles**, select the large ellipse in the graphics area in the approximate location as shown for **Profile** .

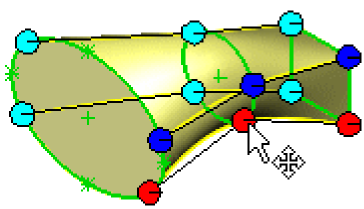
The middle profile handle retains its new position when you add a profile to the loft.



9 Right-click and select **Show All Connectors**.

10 Hover over the various connectors' handles.

All handles within the same connector chain highlight.



## Mirrors

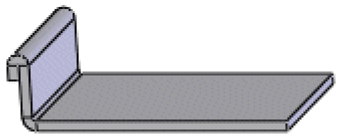
You can mirror the following individual sheet metal features:

- Closed corners
- Edge flanges
- Hems
- Mitered flanges

Previously, you could mirror only entire sheet metal bodies, not individual sheet metal features.

***To mirror individual sheet metal features:***

1 Open **Mirror_SheetMetal.sldprt** from the **File Explorer**.



The sheet metal part has an edge flange and a hem feature.

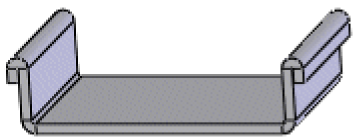
2 Click **Mirror** on the Features toolbar, or click **Insert**, **Pattern/Mirror**, **Mirror**.

3 In the FeatureManager design tree, select:

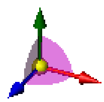
a) **Right Plane** for **Mirror Face/Plane**.

b) **Edge-Flange1** and **Hem1** for **Features to Mirror**.

4 Click **OK**.



## Move/Copy Bodies



The **Move/Copy Bodies** feature has a new triad that lets you drag to move or copy bodies. The triad has the same functionality as the triad in assemblies or the deform-surface push feature. For more information on using the triad, see **Surface Push Deform Type** on page 3-4.

Previously, you had to type exact **Delta X**, **Delta Y**, or **Delta Z** values to move or copy bodies. This functionality remains available.

# Split Line Curves

## Silhouette Split Lines

Two new options in the **Split Line** PropertyManager help you create a silhouette split line:

- **Reverse direction**. Flips the **Direction of pull** in the opposite direction.
- **Angle** . Creates a draft angle for manufacturing considerations. This is commonly used in thermoform packaging.

## Intersection Split Lines

The **Split Line** tool can now split multiple faces with an intersecting solid, surface, face, plane, or surface spline. Split lines can be created at all intersections or selected faces of the splitting tool and the target body. You can create multiple split lines if the splitting tool intersects the target body in more than one place.

> You can use the **Spline on Surface** tool to create a spline for the intersection split line. This is the only sketch entity supported by the intersection split line. See **Splines on Surfaces** on page 2-16.

***To split multiple faces with an intersecting solid:***

1 Open **rocket.sldprt** from the **File Explorer** .

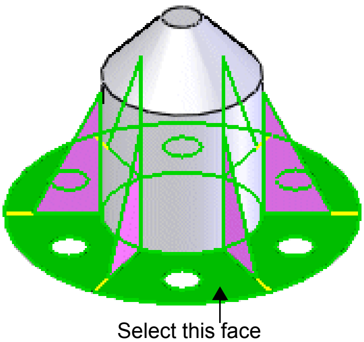
2 Click **Split Line** on the Curves toolbar, or click **Insert**, **Curve**, **Split Line**.

3 In the PropertyManager, under **Type of Split**, select the new **Intersection** option.

4 Under **Selections**, for **Splitting Bodies/Faces/ Planes** , in the FeatureManager design tree:

a) Select **Surface-Plane1**.

b) Expand the **Surface-Bodies** folder and select all six **Body-Move/Copy** surface bodies.

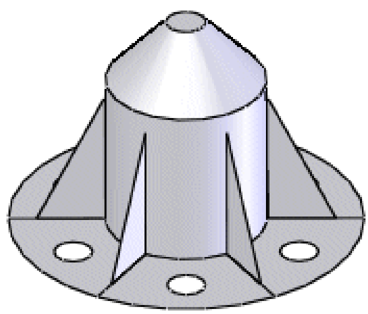



5 Under **Selections**, click in **Faces/Bodies to Split** , then select the face in the graphics area as shown.

6 Under **Surface Split Options**, select **Natural**.

7 Click **OK** .

The face is split into six faces.



## Sweeps

### Direction Vector References

More references for the direction vector are now available, such as pairs of vertices, planes, edges, cylinders, axes, and so on. This applies to sweeps you create using **Follow Path** in **Orientation option/twist type** with **Direction Vector** in **Path alignment type**.

### PropertyManager Enhancements

Under **Options**:

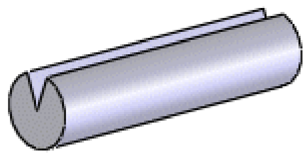
- The **Maintain tangency** option has been renamed to **Merge tangent faces** to better reflect its functionality.
- **Advanced smoothing** has been removed because this functionality is now automatically applied.

### Twist

You can create sweeps that twist along a path. You control the twist by setting the number of degrees, radians, or turns. You can also create sweeps that twist along a path while keeping the normal constant so the start and end profiles remain parallel to each other. You cannot twist sweeps that use guide curves.

***To create a sweep that twists along a path:***

1 Open **SweepTwist.sldprt** from the **File Explorer**, which shows a sweep along a path.



2 In the FeatureManager design tree, right-click **Sweep1** and select **Edit Feature**.

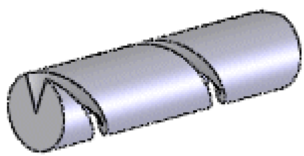
3 In the PropertyManager, under **Options**:

a) Select the new **Twist Along Path** option in **Orientation/twist type**.

b) Select **Turns** in **Define angle by degree, radian or number of turns**.

c) Set **Angle defined in number of turns** to **2**.

4 Click **OK**.



# 4

# Parts

This chapter describes enhancements to parts in the following areas:

- Custom properties
- Equations, global variables, and linked values
- Lighting
- Mass properties
- Measure tool
- Multibody parts

# Custom Properties

The **Custom Properties** and **Configuration Specific Properties** dialog boxes have been redesigned. Properties, values, and so on, are now displayed in a grid.

***To add custom properties:***

1. Open **\parts\well.sldprt**.
2. Click **File**, **Properties**.
3. On the **Custom** tab, click in the cell under **Property Name**, and select **PartNo**.

   The property type **Text** appears under **Type**.
4. Under **Value / Text Expression**, type **J0726**, then press **Enter**.

   **J0726** appears for **Evaluated Value**.
5. Click in the second cell under **Property Name**, and type **Diameter**.
6. Under **Type**, select **Text**.
7. For **Value / Text Expression**, click the diameter dimension, **20**, in the graphics area.

   **"D1@Sketch4@well.SLDPRT"** appears for **Value / Text Expression**.
8. Place the pointer at the end of the dimension name, type **mm**, then press **Enter**.

   **20mm** appears for **Evaluated Value**.
9. Click **OK**.

   New custom properties **PartNo** and **Diameter** are created.

For **Value / Text Expression**, you can also select values from the list, which contains:
- SolidWorks parameters (**Mass**, **Density**, and so on).
- Global variables.
- Linked dimension names.

(See **Global Variables** on page 4-5 and **Linked Values** on page 4-7.)

# Equations, Global Variables, and Linked Values

## Dimension Status Indicator

If a dimension is driven by an equation or linked to another dimension, a symbol appears with the dimension in the graphics area.

Σ dimension driven by an equation

∞ dimension linked to another dimension through a shared value

## Modifying Dimensions

When you double-click a dimension that is driven by an equation, the **Modify** dialog box appears, just as it has previously for linked dimensions. You make selections and changes based on the dimension status.

If the dimension is *not* driven by an equation and is *not* linked to another dimension, you can click in the **Modify** dialog box and select:

- **Add Equation**
- **Link value**

If the dimension is driven by an equation, the dimension is unavailable and Σ appears in the **Modify** dialog box. You can click in the dialog box and select:

- **Edit Equation**
- **Delete Equation**

If the dimension is linked to other dimensions, appears in the **Modify** dialog box. You can:

- Change the dimension. All instances of the shared value update.
- Click in the **Modify** dialog box and select **Unlink value** to change only that dimension.

## Equations

The user interface for **Equations** has been enhanced.

In the **Equations** dialog box:

- Select **Degrees** or **Radians** for **Angular Equation Units** for use with trigonometric functions in equations. This setting is saved as a document setting.
- View comments in a separate **Comment** column.
- View global variables. See **Global Variables** on page 4-5.
- View linked dimension variables. See **Linked Values** on page 4-7.
- View the name of the model being edited.

The user interface for adding and editing equations has been enhanced. The **Add Equation** and **Edit Equation** dialog boxes include:

- Buttons for all previously supported mathematical functions (sine, cosine, and so on).
- New mathematical functions (secant, cosecant, and so on).
- A **Comment** button to add a comment to the equation.
- The ability to switch the keypad visibility on and off.

Also, you can access the **Equations** dialog box from the **Modify** dialog box by double-clicking a dimension driven by an equation.

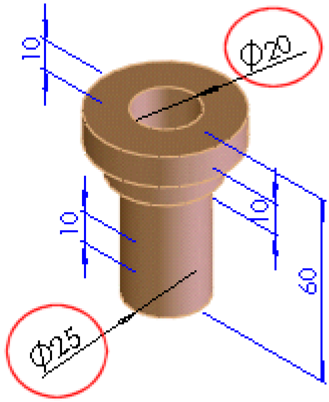
***To add an equation:***

1 Open **\parts\well.sldprt**.

2 Click **Equations** [Σ] on the Tools toolbar, or click **Tools**, **Equations**.

3 In the dialog box, click **Add**.

4 Populate the equation box:




a) Select **25**, the outside diameter of the tube (**D1@Sketch1**).

b) Click [=].

c) Select **20**, the inside diameter of the tube (**D1@Sketch4**).

d) Click [+], then [8].

e) Click **Comment** and type **tube OD**.

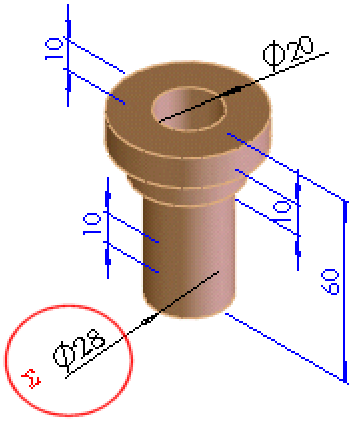
The following appears in the equation box:

**"D1@Sketch1" = "D1@Sketch4" + 8'tube OD**

5 Click **OK** twice.

The value for the outside diameter updates in the graphics area. Σ appears with the dimension to indicate that it is driven by an equation.

6 Keep **well.sldprt** open for the procedures that follow.



### *To edit an equation:*

1 Double-click the dimension, **28**.

In the **Modify** dialog box, note that the dimension is unavailable.

2 Click ▾ and select **Edit Equation**.

3 In the **Edit Equation** dialog box, change **8** to **10**, and click **OK** twice.

The value for the outside diameter updates to **30**.

### *To delete an equation:*

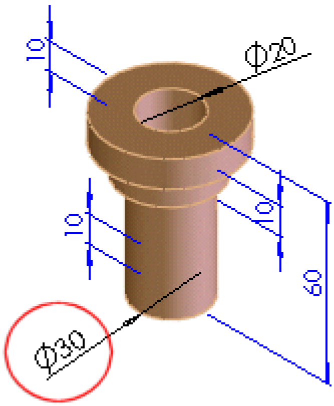
1 Double-click the dimension, **30**.

2 In the dialog box, click ▾ and select **Delete Equation**.

The **Equations** dialog box appears, with the equation that drives **D1@Sketch1** highlighted.

3 Click **Delete**, then click **OK**.

The equation is deleted, and Σ no longer appears with the dimension in the graphics area.




## Global Variables

You can create global variables to use in equations. You define global variables in the **Equations** dialog box (example: **Well_Volume = 20000**). You can:

- Define global variables using other global variables, dimensions, and linked dimension names (example: **Height = "Well_Volume"/(pi*("D1@Sketch4"/2)^2)**).
- Drive dimensions using global variables (example: **D1@Extrude1 = "Height"**).
- Use a part's global variables within an assembly.

Global variables and their current values appear:

- In the **Equations** folder in the FeatureManager design tree.
- Under **Value / Text Expression** on the **Custom** tab in the **Summary Information** dialog box (accessed by clicking **File**, **Properties**).
- Under **Value / Text Expression** in the **Weldment** dialog box.

***To create a global variable:***

1 Open **\parts\well.sldprt**, if it is not already open.

2 Click **Equations** on the Tools toolbar, or click **Tools**, **Equations**.

The **Equations** dialog box appears, and the **Equations** folder in the FeatureManager design tree expands to show a previously created global variable and its value, **“Well_Volume”=20000**.

3 In the dialog box, click **Add**.

4 Populate the equation box:

a) Type **Height**.

b) Click =, then (.

c) In the **Equations** folder in the FeatureManager design tree, click **“Well_Volume”=20000**.

d) Click /, (, pi, *, (.

e) Click **20**, the inside diameter of the tube (**D1@Sketch4**).

f) Click /, 2, ), ^, 2, ), ), +.

g) Click **10**, the thickness at the bottom of the tube (**D1@Cut-Extrude1**).

The following appears in the equation box:

**Height = ( “Well_Volume” / (pi * ( “D1@Sketch4” / 2 ) ^ 2 ) ) + “D1@Cut-Extrude1”**

5 Click **OK**.

In the **Equations** dialog box, the new equation and its evaluate value, **73.662mm**, appear.

6 Click **OK** again.

**“Height”=73.662mm** appears in the **Equations** folder in the FeatureManager design tree, where you can select it for use in other equations.

7 Keep **well.sldprt** open for the procedure that follows.

***To drive a dimension using a global variable:***

1 Click **Equations** on the Tools toolbar, or click **Tools**, **Equations**.

2 In the dialog box, click **Add**.

3 Populate the equation box:

a) Click **60**, the length of the tube (**D1@Extrude1**).

b) Click =.

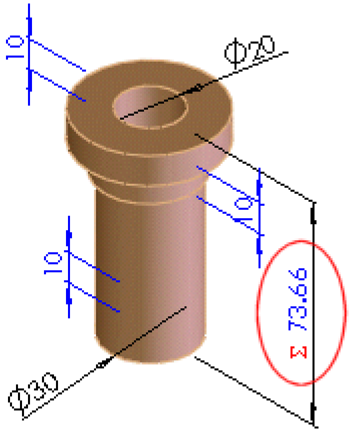
c) In the **Equations** folder in the FeatureManager design tree, click **“Height”=73.662mm**.

The following appears in the equation box:

**“D1@Extrude1” = “Height”**

4 Click **OK** twice.

The value for the length updates in the graphics area. Σ appears with the dimension to indicate that it is driven by an equation.




## Linked Values

The user interface for linked dimension values has been enhanced. You can now:

- Link dimension values from the **Modify** dialog box.
- Use the names of linked dimensions in equations.

Linked dimension names and their current values appear:

- In the **Equations** folder in the FeatureManager design tree.
- In the **Equations** dialog box.
- Under **Value / Text Expression** on the **Custom** tab in the **Summary Information** dialog box (accessed by clicking **File**, **Properties**).
- Under **Value / Text Expression** in the **Weldment** dialog box.

### *To link a dimension:*

1 Open **\parts\well.sldprt**, if it is not already open.

2 Show the dimension names:

a) Click **Options** on the Standard toolbar, or click **Tools**, **Options**.

b) On the **System Options** tab, click **General**.

c) Select **Show dimension names**.

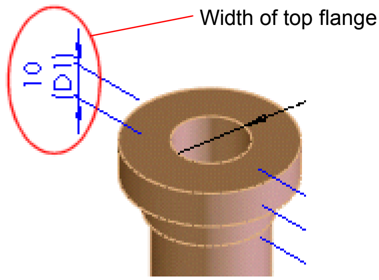
d) Click **OK**.

3 In the graphics area, double-click the dimension for the width of the top flange, **10**.

4 In the **Modify** dialog box, click and select **Link Value**.

5 In the **Shared Values** dialog box, type **width** in **Name**, then click **OK**.

In the graphics area, ∞ appears with the dimension, and the dimension name changes to **width**. The linked dimension name **width** appears in the **Equations** folder in the FeatureManager design tree.

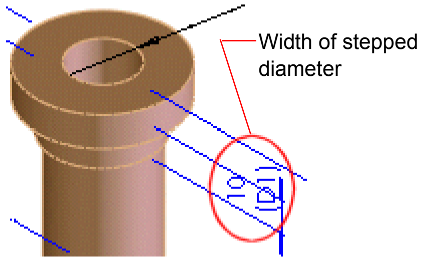


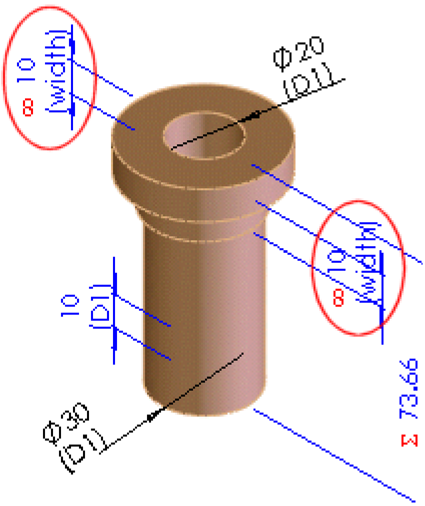

6 Double-click the dimension for the width of the stepped diameter, **10**.

7 In the **Modify** dialog box, click and select **Link Value**.

8 In the **Shared Value** dialog box, select **width** in **Name**, then click **OK**.

In the graphics area, ∞ appears with the dimension.




9 Keep **well.sldprt** open for the procedure that follows.

### *To unlink a value:*

1 In the graphics area, double-click one of the linked dimensions.

2 In the dialog box, click and select **Unlink Value**.

The dimension is no longer linked, and ∞ no longer appears with the dimension.

3 Close the part.

# Lighting

## General Enhancements

With **RealView Graphics**, when a model is viewed in **Shaded With Edges** mode, the edges do not change to white when the model is rotated. Previously, the edges sometimes changed to white when adjacent faces were white, resulting in a loss of visibility.

## Dynamic Lighting

You can position lights (point, spot, and directional) in a model by dragging manipulators, which eliminates the need to type their coordinates. Additionally, the **Ambient Properties**, **Directional Properties**, **Point Properties**, and **Spot Properties** dialog boxes have moved to the **Ambient**, **Directional**, **Point**, and **Spot** PropertyManagers. All previous dialog box functionality is retained in the PropertyManagers.

***To position a Directional light:***

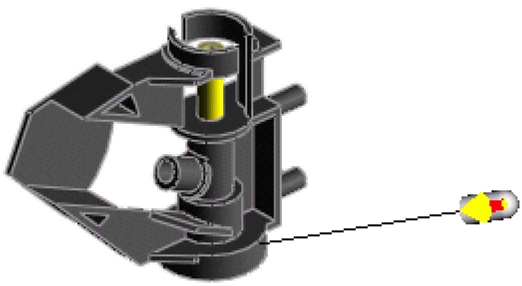
1. Open **\parts\Sprinkler_Body.sldprt**.
2. Expand the **Lighting** folder in the FeatureManager design tree.
3. Double-click **Directional1**.

   The **Directional** PropertyManager appears, and a manipulator appears in the graphics area.

   > The appearance of the manipulator varies, depending on the zoom scale of the graphics area and the position of the light.

4. Move the pointer over the yellow dot on the manipulator.

   The pointer changes to .
5. Drag the manipulator, and note the changes in lighting.
6. Drop the manipulator in the approximate location shown.



7. Click **OK** .

***To add and position a point light:***

1 Right-click the **Lighting** folder in the FeatureManager design tree, and select **Add Point Light**.

2 Double-click **Point1**.

The **Point** PropertyManager appears, and a manipulator appears in the graphics area.

3 In the PropertyManager, under **Basic**, click **Edit Color**, select blue, then click **OK**.

4 Move the pointer over the red dot on the manipulator.

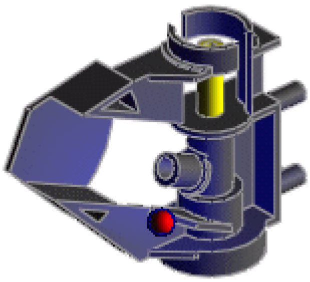
The pointer changes to .

5 Drag the manipulator around the graphics area, and note the changes in lighting.

6 In the PropertyManager, under **Light Position**, set:

- **X coordinate** to **0**.
- **Y coordinate** to **0**.
- **Z coordinate** to **45**.

7 Click **OK** .



***To add and position a spot light:***

1 Right-click the **Lighting** folder in the FeatureManager design tree, and select **Add Spot Light**.

2 Double-click **Spot1**.

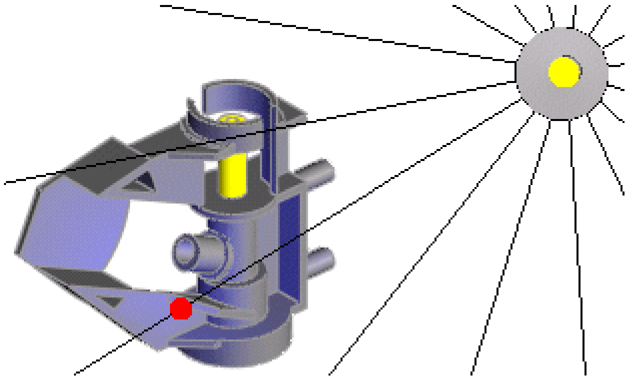
The **Spot** PropertyManager appears, and a manipulator appears in the graphics area.

The appearance of the manipulator varies, depending on the zoom scale of the graphics area and the position of the light.

3 Move the pointer over the center of the manipulator.

The pointer changes to .

4 Drag the manipulator to the approximate location shown.

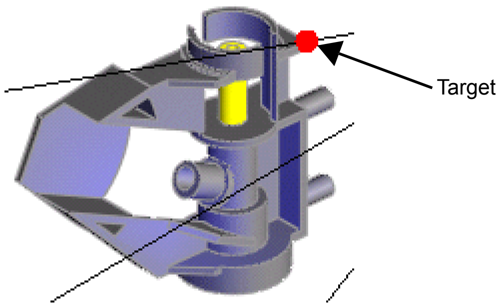


The target appears on the model. The target is the point on the model at which the spot light is aimed.

5 Move the pointer over the target.

The pointer changes to .

6 Drag the target to the upper right corner of the part, as shown.



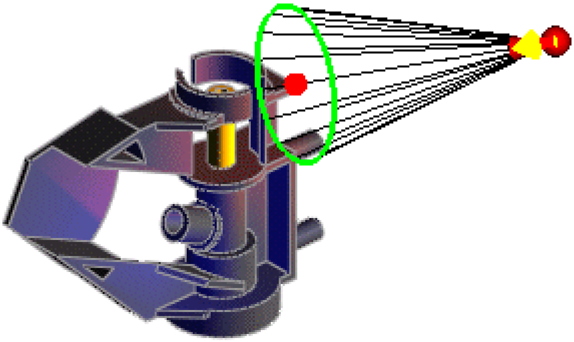

The target snaps to the vertex.

7 In the PropertyManager:

- Under **Basic**, click **Edit Color**, select red , then click **OK**.
- Under **Light Position**, set **Z coordinate** to **75**.

8 In the graphics area, drag the green circle to change the cone angle to the approximate size shown.

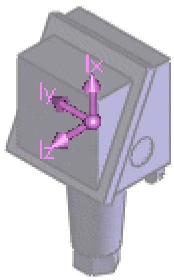
9 Click **OK**.



## Mass Properties

You can now assign mass properties to a part or assembly. This is useful when you create a simplified representation of a component (for example, a purchased component such as a gearbox or valve) and want to assign the correct mass and center of gravity to the model. The mass properties can also be configured in the **Mass Properties** dialog box.

***To assign mass properties to a part:***




1 Open **\parts\pressure_switch.sldprt**.

2 Click **Mass Properties** on the Tools toolbar, or click **Tools**, **Mass Properties**.

A 3D triad appears at the centroid of the part. The calculated values for the mass properties appear in the dialog box.

3 Select **Assigned Mass Properties**.

The **Mass Properties in the Component's Coordinate System** appear.

4 Set **Mass** to **1500**.

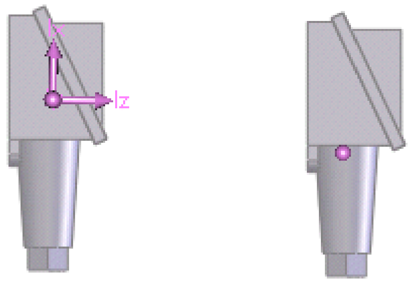
5 For **Center of gravity**, set:

- **X** to **0**.
- **Y** to **-5**.
- **Z** to **23**.

> You can also select any vertex, sketch entity, or reference point in the graphics area for **Center of gravity**.

6 Under **Apply To**, select **This configuration**.

7 Click **OK**.



Calculated Center of Gravity    Assigned Center of Gravity

In a design table, you can assign mass with the keyword **$SW-MASS**. You can assign center of gravity using the keyword **$SW-COG** (with a value of **x, y, z**). If these fields are left blank for a configuration, then the calculated mass properties are used. If you assign values in the design table, then later clear **Assigned Mass Properties** in the **Mass Properties** dialog box, the fields clear in the design table.

## Measure Tool

The **Measure** tool includes enhanced functionality and user interface:

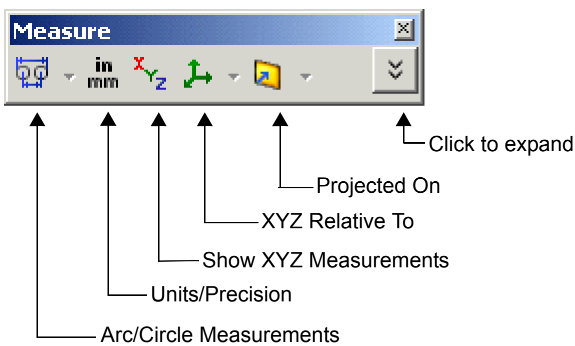
- Displays measurement results in the graphics area, on or near the entities measured.
- Displays coordinates in the graphics area.

---

When the **Measure** tool is not active, commonly-used measurements for selected entities appear in the status bar; in many cases you do not need the **Measure** tool.

---

The **Measure** dialog box has been reduced to the size of a small toolbar, which you can expand as needed.

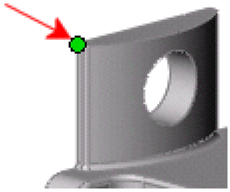


Additionally, the **Measure Units/Precision** dialog box replaces the **Measurement Options** dialog box. Options from the old dialog box which do not belong to the **Measure** tool do not appear in the new dialog box. You can select **Use document settings**, or select **Use custom settings** to set various options such as **Length unit**, **Decimal places**, and so on.

***To display coordinates:***

1 Open **\parts\bracket_b.sldprt**.

2 Select the vertex on the top left corner of the part, as shown.



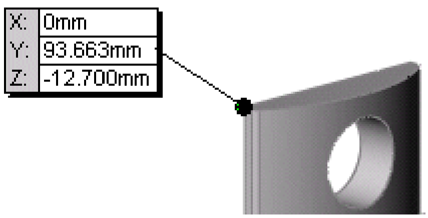
The **X**, **Y**, and **Z** coordinates of the point appear in the status bar.

3 Click **Measure** on the Tools toolbar, or click **Tools**, **Measure**.

4 In the dialog box:

- Click **Show XYZ Measurements** if it is not already selected.
- Click to expand the dialog box, if it is not already expanded.

The **X**, **Y**, and **Z** coordinates of the point appear in the dialog box and in the graphics area.



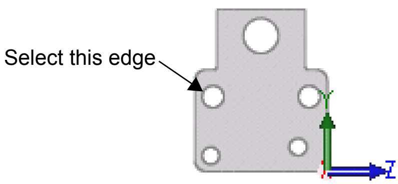

5 Keep **bracket_b.sldprt** open for the procedure that follows.

***To measure between entities:***

1 Click a blank area of the graphics area to clear your selection from the previous procedure.

2 Click **Left** on the Standard Views toolbar.

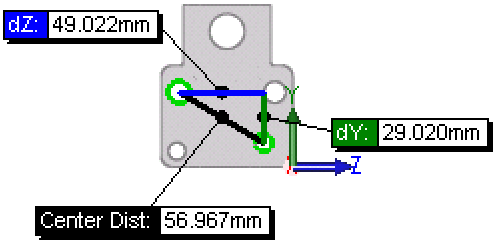
3 Select the edge of the hole shown.



A callout displays the diameter of the circle and the coordinates of the center of the circle.

4 Select the edge of the hole in the lower right corner.

A callout displays the distance between the centers of the two holes. Callouts also show the difference in coordinates between the two holes.




When **Show XYZ Measurements** is selected in the **Measure** dialog box, the **dY**, **dZ**, and center distance callouts appear.

When **Show XYZ Measurements** is cleared, only the center distance callout appears.

5 Click **Arc/Circle Measurements** and select **Minimum Distance**.

The callouts update to show information based on the minimum distance between the two circles.

## Multibody Parts

The organization of solid bodies for multibody parts was improved in the FeatureManager design tree. You can:

- Group bodies into folders in the **Solid Bodies** folder.
- Select commands to apply to all bodies within a folder.
- List the features that belong to each body.

***To group bodies into folders:***

1 Expand the **Solid Bodies** folder in the FeatureManager design tree.

2 Right-click a solid body name, select **Add to New Folder**, and name the folder.

The selected solid body is listed in the new folder. You can drag other bodies into the same folder, and create other new folders and subfolders.

***To apply commands to all bodies within a folder:***

1 Right-click the folder.

2 Select a command such as **Hide Solid Body**, **Delete Body**, **Appearance**, and so on.

The command is applied to all bodies in the folder.

***To list the features that belong to each solid body:***

1 Right-click the **Solid Bodies** folder in the FeatureManager design tree.

2 Select **Show Feature History**.

3 Expand the solid body to see the features that belong to that body.

4 To hide the feature history, right-click the **Solid Bodies** folder and clear **Show Feature History**.

5

# Mold Tools

This chapter describes enhancements to mold tools in the following areas:

- Cores
- Heal edges
- MoldflowXpress
- Mold folders
- Move face
- Parting lines
- Parting surfaces
- Planar surface from co-planar loops
- Tooling splits
- Shut-off surfaces
- Undercut detection

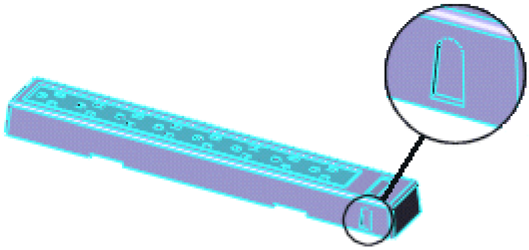
## Cores

With the new **Core** tool, you can extract geometry from the tooling solid to create a core feature. You can also create lifters and trimmed ejector pins.

***To add a core:***

1 Open **\mold_tools\power_strip.sldprt**.

The main core and cavity for this plastic part were previously created, but are hidden for clarity. The recessed feature on the front of the part is a trapped molding area that requires a core. A core sketch was previously created, but is hidden for clarity.

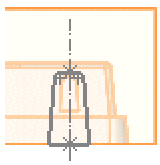


2 In the FeatureManager design tree, expand the **Solid Bodies** folder, then:

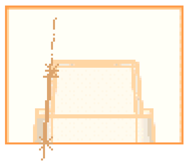
- Right-click **Cavity Body**, and select **Show Solid Body**.
- Right-click **Plastic Part Body**, and select **Hide Solid Body**.

The cavity appears and the plastic part disappears.

3 In the FeatureManager design tree, right-click **Side Core Sketch** and select **Show**.



Front view



Right view

The plane on which you create a core sketch does not need to be parallel to the direction that the side core travels. **Side Core Sketch** was created on the inside face of the cavity body, which is drafted 5° from the direction the core travels.

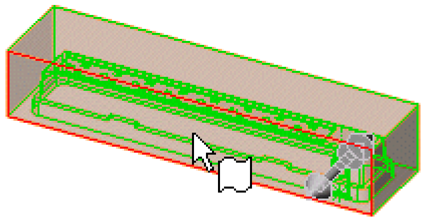
4 Click **Core** on the Mold Tools toolbar, or click **Insert**, **Molds**, **Core**.

5 Select **Side Core Sketch** in the flyout FeatureManager design tree.

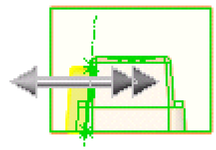
The **Core** PropertyManager appears, with the following selected:

- **Side Core Sketch** for **bounding sketch for core**.
- **Face <1>** (the sketch plane) for **extraction direction**.
- **Solid Body <1>** (the cavity body) for **Core/Cavity body**.

6 In the PropertyManager, under **Selections**, click in **extraction direction**, then select the front face of the cavity body as shown.



7 Click **Reverse direction** if necessary, so the single-headed arrow (extraction direction) appears as shown.



Right view

8 Under **Parameters**, set:

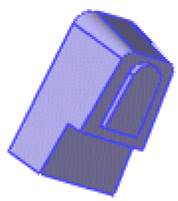
- **End Condition** to **Blind**.
- **Depth along extraction direction** to **50**.
- **End Condition** to **Blind**.
- **Depth away from extraction direction** to **25**.

9 Click **OK**.

- A new body is created for the core, and is subtracted from the cavity body.
- In the FeatureManager design tree, in the **Solid Bodies** folder, a new folder named **Core bodies** appears. Additional core bodies you create are stored in this folder.

10 In the graphics area, right-click the cavity body, and select **Hide**.

The core is complete.

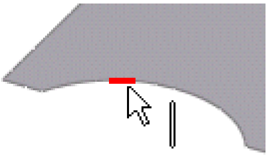


## Heal Edges

You can merge multiple edges into a single edge with **Heal Edges**. This is useful on imported parts, where edges sometimes import as multiple short edges.

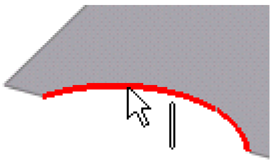
***To heal edges:***

1. Open **\mold_tools\heal_edges.sldprt**.



2. Move the pointer over the edges of the part. Note that the edges of the part are actually made up of many small edges.
3. Click **Heal Edges** on the Features toolbar, or click **Insert**, **Face**, **Heal Edges**.
4. In the PropertyManager, under **Faces**:
   - Select the large face in the graphics area.
   - Set **Angular Tolerance** to **2**.
   - Set **Edge Length Tolerance** to **5**.
5. Click **Heal Edges**.

   Edges less than 5mm long, with angles less than 2° at the vertex between the edges, merge into one edge. Under **Edge Information**, **Before** and **After** display the number of edges in the part before and after the **Heal Edges** operation.
6. Click **OK**.
7. Move the pointer over the edge of the part as shown. Note that long edges replace the multiple short edges.



## MoldflowXpress

With MoldflowXpress, you can analyze plastic parts and their molds based on geometry, material, temperature, and injection gate location. The analysis produces an animated plot of the time it takes to fill the mold. This enables you to:

- Verify that the mold will fill in the allowed time.
- Assess the quality of the resulting part.
- Optimize the location of the injection gate.

To access MoldflowXpress, click **MoldflowXpress Analysis Wizard** on the Mold Tools toolbar, or click **Tools**, **MoldflowXpress**.

For more information, click **Help**, **MoldflowXpress Help**.

## Mold Folders

As in previous releases, the **Cavity Surface Bodies** folder, **Core Surface Bodies** folder, and **Parting Surface Bodies** folder are created automatically when you use various mold tools, and the appropriate surfaces are added automatically to the folders. Now, if you want to define a mold using surfaces that were not created with mold tools, you can create the mold folders manually, then add the surfaces to the folders. To create the mold folders manually, click **Insert Mold Folders** on the Mold Tools toolbar, or click **Insert**, **Molds**, **Insert Mold Folders**.

## Move Face

With the new **Move Face** tool, you can offset, translate, and rotate faces and features directly on solid or surface models.

***To offset a face:***

1 Open **\mold_tools\boss.sldprt**.

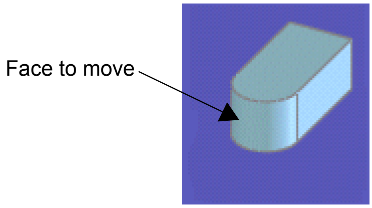
2 Click **Move Face** on the Mold Tools toolbar, or click **Insert**, **Face, Move**.

3 In the PropertyManager, under **Move Face**:

a) Select **Offset**.

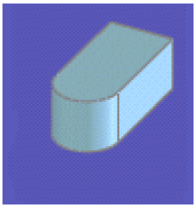
b) Select the curved face of one of the bosses for **Face(s) to move** .

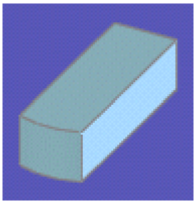
c) Set **Distance** to **5**.




4 Click **OK** .

The selected face is offset the specified distance. The radius of the curved face increases.


Original boss


Offset face

***To translate a face:***

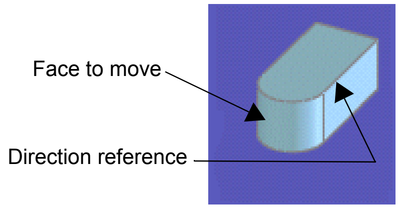
1 Click **Move Face** on the Mold Tools toolbar, or click **Insert**, **Face, Move**.

2 In the PropertyManager, under **Move Face**:

a) Select **Translate**.

b) Select the curved face of another boss for **Face(s) to move** .

c) Set **Distance** to **5**.

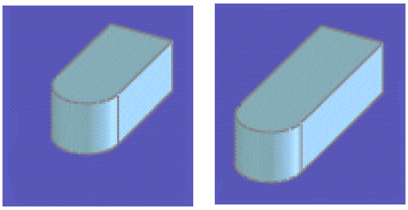



3 Under **Parameters**, for **Direction reference** , select the long edges of the boss, as shown.

4 Under **Move Face**, select **Flip direction**.

5 Click **OK** .

The selected face is translated the specified distance. The radius of the curve of the face remains the same.


Original boss Translated face

***To rotate a face:***

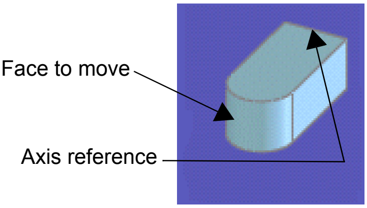
1 Click **Move Face** on the Mold Tools toolbar, or click **Insert**, **Face, Move**.

2 In the PropertyManager, under **Move Face**:

a) Select **Rotate**.

b) Select the curved face of another boss for **Face(s) to move** .

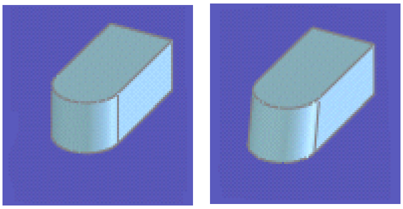
c) Set **Draft Angle** to **20**.




3 Under **Parameters**, for **Axis reference** , select the back edge of the boss.

4 Click **OK** .

The selected face is rotated by the specified angle.



Original boss     Rotated face

You can also select features (such as **Extrudes** and **Cut-Extrudes**) to offset, translate, and rotate with **Move Face** .

## Parting Lines

Parting line functionality has been enhanced, so you can create:

- Multiple parting line features in a single part.
- Partial parting line features.
- Parting line features that contain errors, reported as warnings.

To make parting lines easier to define, you can automatically split straddle faces found during draft analysis, either along the +/- boundary or at a specified draft angle. You can also split a face by selecting a:

- Sketch segment.
- Pair of vertices.
- Spline on a surface (see **Splines on Surfaces** on page 2-16).

Additionally, parting line features are always visible, even when they are not selected.

***To automatically split straddle faces:***

1 Open **\mold_tools\parting_line.sldprt**.

2 Click **Parting Lines**  on the Mold Tools toolbar, or click **Insert**, **Molds**, **Parting Line**.

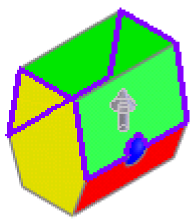
3 In the PropertyManager, under **Mold Parameters**:

a) Select **Top Plane** in the flyout FeatureManager design tree for **Direction of Pull** .

b) Set **Draft Angle**  to **1**.

c) Click **Draft Analysis**.

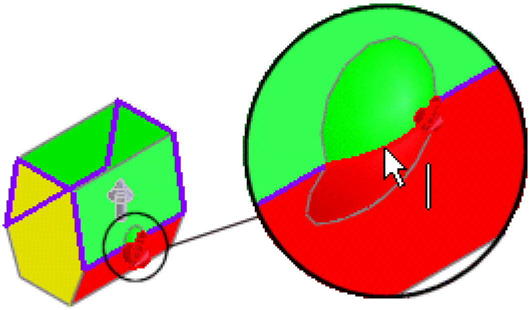
Note that the spherical surface is identified as a straddle face (it has both positive and negative draft).



4 Under **Mold Parameters**, select **Split faces**.

The spherical face is split.

5 Select the new edge in the graphics area.



In the PropertyManager, the edge appears under **Parting Lines**.

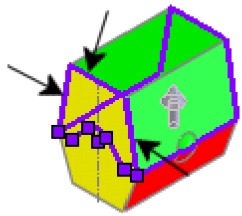
6 Click **OK** .

***To split a face by selecting a sketch segment:***

1 Click **View**, **Sketches** to make sketches visible.

2 In the FeatureManager design tree, right-click **Parting Line1** and select **Edit Feature**.

3 In the PropertyManager, click in **Entities To Split**, and select the sketch segments on the front face of the part.



The new edges appear under **Parting Lines**.

4 Under **Parting Lines**, select each of the three redundant edges from the list, and press **Delete** for each edge.

5 Click **OK** .

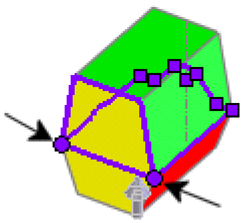
***To split a face by selecting a pair of vertices:***

1 In the FeatureManager design tree, right-click **Parting Line1** and select **Edit Feature**.

2 In the PropertyManager, click in **Entities To Split**.

3 Rotate the part so you can see the back face.

4 Select the two vertices as shown.



A parting line segment appears.

5 Under **Parting Lines**, select each of the three redundant edges from the list, and press **Delete** for each edge.

6 Click **OK** .

## Parting Surfaces

The parting surface functionality has been enhanced so you can:

- Display a callout to show the minimum radius of curvature.
- Use advanced selection tools, previously available only in the **Parting Line** PropertyManager.

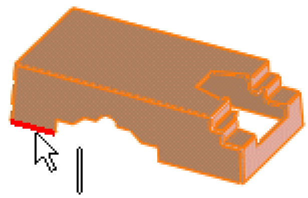
***To create a parting surface manually:***

1 Open **\mold_tools\box02.sldprt**.

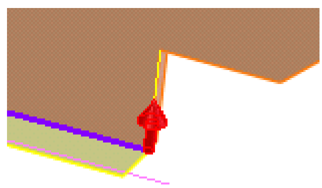
2 Click **Parting Surfaces** on the Mold Tools toolbar, or click **Insert**, **Molds**, **Parting Surface**.

3 In the PropertyManager, under **Mold Parameters**, select **Top Plane** in the flyout FeatureManager design tree for **Direction of Pull** .

4 Under **Parting Line**, select the edge shown for **Edges** .



In the PropertyManager, the selection tools appear under **Parting Line**. In the graphics area, the edge highlights and a red handle appears.



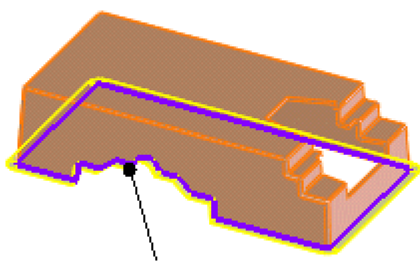
5 In the PropertyManager, under **Parting Line**, click **Select next edge** if necessary, to change the direction of the red handle, as shown.

6 Click **Add selected edge** to add the next edge.

7 Continue the parting line. Click **Select next edge** as needed, to choose edges along the outer perimeter of the part; click **Add selected edge** to add the edges.

> If you select **Zoom to the selected edge** , the image crops to the relevant area each time you click **Add selected edge** .

The callout indicates the minimum radius on the parting surface.



8 Click **OK** .

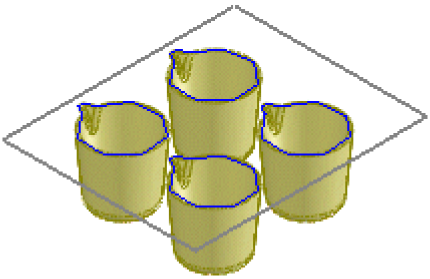
## Planar Surface from Co-planar Loops

You can construct a planar surface from two or more co-planar loops. Select parting line features or edges of other features to define the loops.

***To construct a planar surface from co-planar loops:***

1 Open **\mold_tools\cups.sldprt**.

The rectangle (**Sketch6**) and the top faces of the four cups are in the same plane.



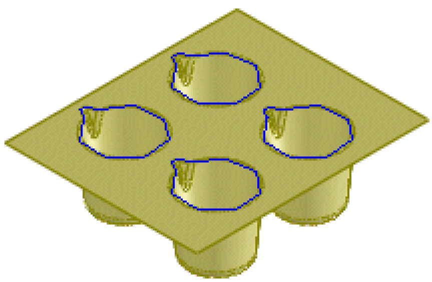
2 Click **Planar Surface** on the Surfaces toolbar, or click **Insert**, **Surface**, **Planar**.

3 In the PropertyManager, for **Bounding Entities**, select the following in the flyout FeatureManager design tree:

- **Parting Line1** through **Parting Line4**.
- **Sketch6** (the rectangle).

4 Click **OK**.

A planar surface is created, bounded by the rectangle and the edges of the four cups.



## Tooling Splits

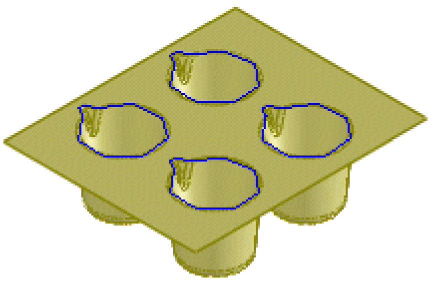
You can create a tooling split for multiple bodies.

***To create a tooling split for a multibody part:***

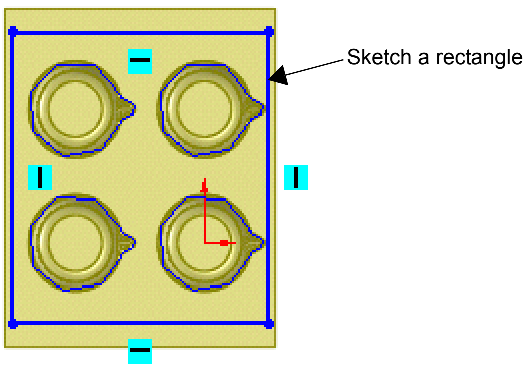
1 Open **\mold_tools\cups2.sldprt**.

2 Click **Tooling Split** on the Mold Tools toolbar, or click **Insert**, **Molds**, **Tooling Split**.

In the graphics area, for the sketch plane, select the flat surface.



3 Sketch a rectangle approximately as shown.

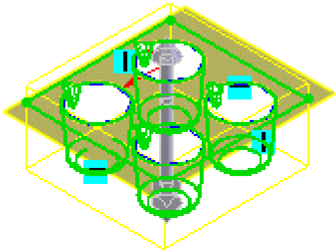



4 Close the sketch.

5 Click **Isometric** on the Standard Views toolbar.

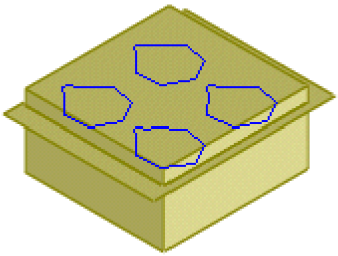
6 In the PropertyManager, under **Block Size**, set **Depth in Direction 2** to **40**.

7 Under **Parting Surface**, click in **Parting Surface Bodies**, then select the flat surface shown.



8 Click **OK**.

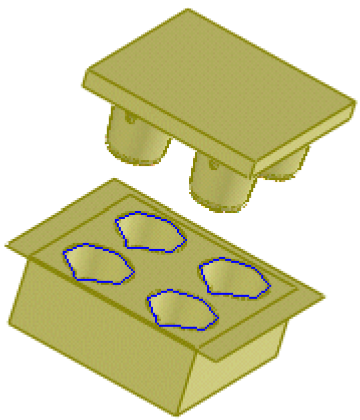
The tooling split is complete. **ToolingSplit1[1]** and **Tooling Split1[2]** appear in the **Solid Bodies** folder in the FeatureManager design tree.



---

You can use **Move/Copy Bodies** on the Features toolbar to separate the tooling split bodies to facilitate viewing them.

---



## Shut-Off Surfaces

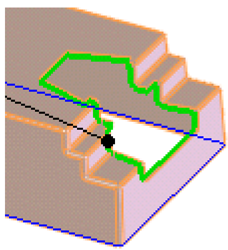
Shut-off surfaces have been enhanced so you can now fill telescopic shut-off areas. You can also:

- Specify a parting line feature as a selection.
- Use advanced selection tools, previously available only in the **Parting Line** PropertyManager.
- Select **Show Preview**.

***To create a shut-off surface:***

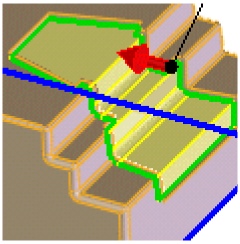
1. Open **\mold_tools\box01.sldprt**.
2. Click **Shut-off Surfaces** on the Mold Tools toolbar, or click **Insert**, **Molds**, **Shut-Off Surfaces**.

   The edges of the area requiring shut-off are highlighted, and a callout for the shut-off surface type appears. In the PropertyManager, the selected edges appear in **Edges** .



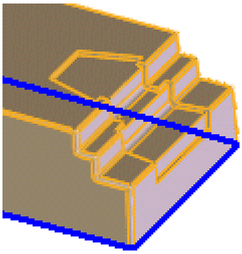
3. In the PropertyManager, under **Edges** , select **Show Preview**.

4 Click **Contact** in the callout to change it to **Tangent**, and if necessary, click the red arrow to change the preview as shown.



5 Click **OK** .

The shut-off surface appears.



## Undercut Detection

For parts with non-planar parting lines, you can specify to use the parting line during **Undercut Detection**, which produces more accurate results.

# 6

# Weldments

This chapter describes enhancements to weldments in the following areas:

- Cut lists
- Structural members along arcs
- Weldment trimming
- Weldment properties

# Cut Lists

## Cut List Folder

In the FeatureManager design tree, the **Solid Bodies** folder is renamed to **Cut list** when the first weldment feature is inserted.

## Automatic Cut List

You can generate a cut list automatically.

***To generate a cut list automatically:***

1 Open **\weldments\welded_frame.sldprt**.

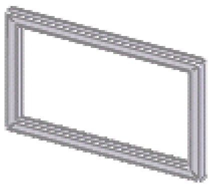
2 Click **Structural Member** on the Weldments toolbar, or click **Insert**, **Weldments**, **Structural Member**.

3 In the PropertyManager, under **Selections**, select:

- **Iso** in **Standard**.
- **Square tube** in **Type**.
- **30 x 30 x 2.6** in **Size**.
- Each line in the sketch for **Path segments**.

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

5 Click **OK** .



The **Cut list** folder appears in the FeatureManager design tree. The **(4)** indicates the number of items in the cut list. The icon indicates that the cut list needs to be updated.

6 Expand the **Cut list** folder .

The four segments of **Structural Member1** are listed.

7 Right-click the **Cut list** folder , and select **Update**.

The segments sort into two sub-folders. The two identical side segments are grouped in one **Cut-List-Item** folder, and the identical top and bottom segments are grouped in the other **Cut-List-Item** folder. The icon indicates that the cut list is up-to-date.

Although you generate the cut list automatically, you manually specify when to update it. This enables you to make many changes, then update the cut list once. However, the cut list updates automatically when you open a drawing that references the cut list.

**Automatic** is on by default for cut lists in new weldment parts. To turn it off, right-click the **Cut list** folder and clear **Automatic**.

## Structural Members Along Arcs

You can create structural member features along arc segments. Previously, only straight lines were supported. You can create the arc segment body as a separate body, or you can merge it with adjacent segment bodies to create one continuous segment.

***To create structural members along arc segments:***

1 Open **\weldments\welded_arc.sldprt**.

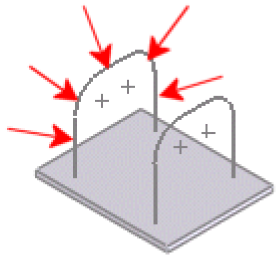
2 Click **Structural Member** on the Weldments toolbar, or click **Insert**, **Weldments**, **Structural Member**.

3 In the PropertyManager, click **Keep Visible**.

4 Under **Selections**, select:

- **Iso** in **Standard**.
- **Square tube** in **Type**.
- **30 x 30 x 2.6** in **Size**.
- Each sketch segment shown for **Path segments**.
- **Merge arc segment bodies**.

Select these sketch segments for the first structural member:

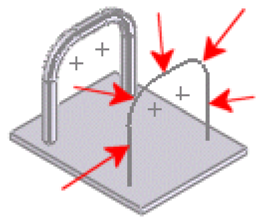


5 Click **OK**.

6 For the next structural member, select each sketch segment shown for **Path segments**.

7 Clear **Merge arc segment bodies**.

Select these sketch segments for the second structural member:



8 Click **OK**.

9 Click **Cancel** to close the PropertyManager.

10 In the FeatureManager design tree, expand the **Cut list** folder.

For **Structural Member1,** because **Merge arc segment bodies** was selected, one segment appears. For **Structural Member2**, because **Merge arc segment bodies** was cleared, five individual segments appear.

## Weldment Trimming

The **Trim/Extend** feature is improved so you can:

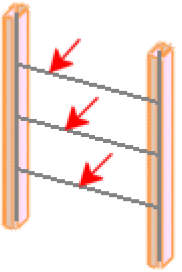
- Trim both ends of a structural member at once by selecting more than one trimming boundary.
- Create more accurate trims with round pipes.
- Preview trimmed features before accepting them in the PropertyManager.

***To trim with more than one planar face:***

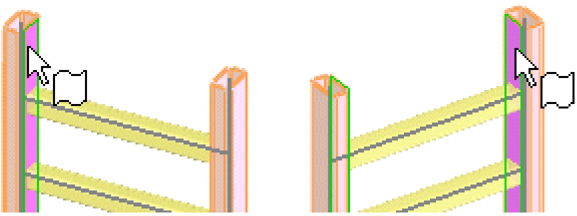
1. Open **\weldments\weld_trim_01.sldprt**.
2. Click **Structural Member** on the Weldments toolbar, or click **Insert**, **Weldments**, **Structural Member**.
3. In the PropertyManager, under **Selections**, select:
   - **Iso** in **Standard**.
   - **Square tube** in **Type**.
   - **40 x 40 x 4** in **Size**.
   - Each line shown for **Path segments**.
4. Click **OK** .

   Three cross members appear.

Select these lines:



5. Click **Trim/Extend** on the Weldments toolbar, or click **Insert**, **Weldments**, **Trim/Extend**.
6. In the PropertyManager, under **Corner Type**, select **End Trim** .
7. Under **Bodies to be Trimmed**, select the three cross members you just created.
8. Under **Trimming Boundary**, select:
   - **Planar face**.
   - The two faces shown for **Faces/Bodies**.
   - **Preview**.



Face <1> Face <2>

Trimming Boundaries

9. Click **OK** .

   The three cross members are trimmed at both ends.

***To trim with round pipe segments:***

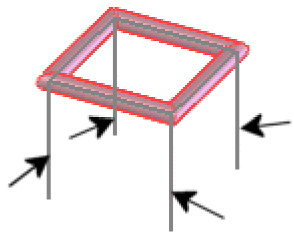
1 Open **\weldments\weld_trim_02.sldprt**.

2 Click **Structural Member** on the Weldments toolbar, or click **Insert**, **Weldments**, **Structural Member**.

3 In the PropertyManager, under **Selections**, select:

- **Iso** in **Standard**.
- **Pipe** in **Type**.
- **33.7 x 4.0** in **Size**.
- Each sketch segment shown for **Path segments**.

Select these sketch segments:

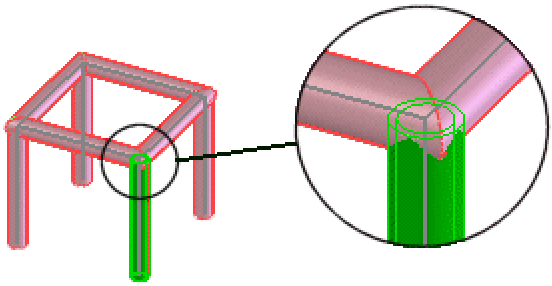


4 Click **OK**.

5 Click **Trim/Extend** on the Weldments toolbar, or click **Insert**, **Weldments**, **Trim/Extend**.

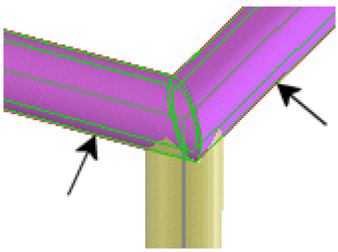
6 In the PropertyManager, under **Bodies to be Trimmed**, select the front pipe segment in the structural member that you just created.

Note that the segment intersects two segments of **Structural Member1**.



7 Under **Trimming Boundary**, select:

- **Bodies**.
- The two bodies shown for **Faces/Bodies**.
- **Preview**.

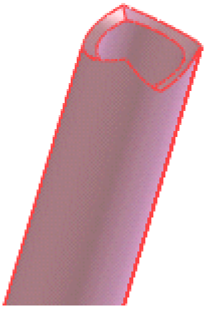


Trimming Boundaries

8 Click **OK** .

The end of the selected pipe segment is trimmed to fit up to the two segments of **Structural Member1**.

9 Hide **Structural Member1**, then rotate the part so you can see the trimmed shape.



## Weldment Properties

The user interface for weldment properties (the **Weldment** dialog box) was updated in the same manner as the **Custom Properties** user interface. See **Custom Properties** on page 4-2. Additionally, global variables and linked dimension names now appear under **Value / Text Expression** in the **Weldment** properties dialog box. See **Global Variables** on page 4-5 and **Linked Values** on page 4-7. Access the **Weldment** dialog box in the same way as in SolidWorks 2004 (right-click the **Weldment** feature in the FeatureManager design tree).

# 7

# Assemblies

This chapter describes enhancements to assemblies in the following areas:

- ❑ Assembly performance
- ❑ Exploded views
- ❑ External references
- ❑ General enhancements
- ❑ Flexible sub-assemblies
- ❑ Interference detection

# Assembly Performance

## Level of Detail in Dynamic Views

In **Performance** options and **Large Assembly Mode** options, the **Remove detail during zoom/pan/rotate** option has been replaced by the **Level of detail** slider.

***To change the level of detail shown in views during zoom, pan, and rotate:***

1 Click **Options** on the Standard toolbar, or click **Tools**, **Options**.

2 On the **System Options** tab, select one of the following:

- **Performance**. Sets options for all assemblies.
- **Large Assembly Mode**. Sets options for only large assemblies.

3 For **Level of detail**:

- Move the slider along the scale from **More (slower)** to **Less (faster)**.

  \- or -

- Move the slider to **Off** (which is the equivalent of selecting the old **Remove detail during zoom/pan/rotate** option).

4 Click **OK**.

## Suspend Automatic Rebuilds in Assemblies

You can defer the update of assemblies until you are ready to rebuild the assembly. By deferring the update, you can make many changes, then rebuild the assembly once. The assembly still rebuilds automatically, if necessary, for internal updates and to protect the integrity of the model.

***To defer the update of assemblies:***

Right-click the assembly name at the top of the FeatureManager design tree, and select **Suspend Automatic Rebuild**.

**(Rebuild Suspended)** appears in the status bar.

To manually update when in **Suspend Automatic Rebuild** mode, click **Rebuild** on the Standard toolbar.

To turn off the option, right-click the assembly name at the top of the FeatureManager design tree, and clear **Suspend Automatic Rebuild**.

You must reset the option every time you load the assembly document.

## Exploded Views

The **Explode** dialog has been moved to the PropertyManager.

You now create exploded views by selecting and dragging parts in the graphics area instead of defining the explode steps in a dialog box.

You can evenly space exploded stacks of components (hardware, washers, and so on) with the new **Auto-space** option.

You can add a new component to existing explode steps. This is useful if you add a new part to an assembly which already has an exploded view.

If a sub-assembly already has an exploded view, you can reuse that view in the top level assembly.

***To create an exploded view:***

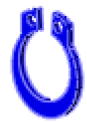
1 Open **\assemblies\Oil Pump\Oil Pump.sldasm**.

2 Click **Exploded View** on the Assembly toolbar, or click **Insert**, **Exploded View**.

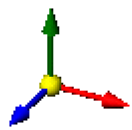
3 In the graphics area, select the retaining ring.

A manipulator appears in the graphics area. **Retaining Ring<1>** appears under **Settings** in the PropertyManager.

> You can drag the yellow sphere in the center of the manipulator to move the manipulator to a different location. If you drop the manipulator on a feature, an axis of the manipulator aligns with the feature.



Retaining ring

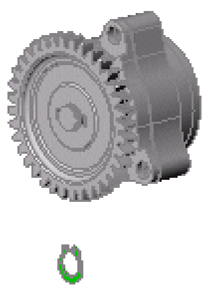


Manipulator

4 Move the pointer over the head of the vertical green arrow of the manipulator.

The pointer changes to .

5 Drag the green arrow of the manipulator down to explode the retaining ring in the vertical direction, and position as shown.



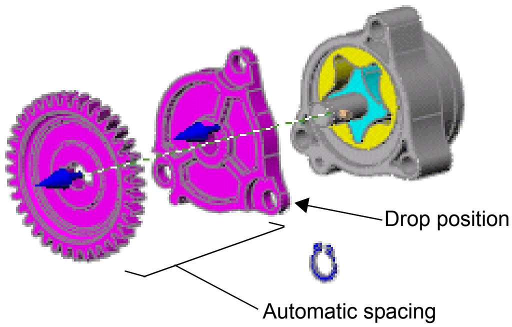
***To auto-space components:***

1 In the flyout FeatureManager design tree, select **Cover<1>** and **Gear<1>**.

2 In the PropertyManager, under **Options**, select **Auto-space components after drag**.

3 Move the pointer over the head of the blue arrow of the manipulator, and drag to position the components as shown.

When you drop the components in the position shown, **Cover** remains where you drop it, and the software automatically spaces **Gear** a distance further along the same axis.




---

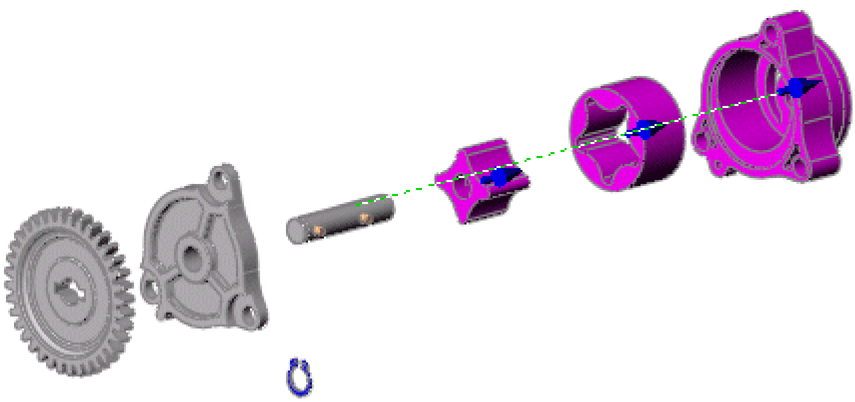
You can change the autospace distance. In the PropertyManager, under **Options**, move the **Adjust the spacing between chain components** slider.

---

4 In the flyout FeatureManager design tree, select **Housing<1>**, **Outer Rotor<1>**, and **Inner Rotor<1>**.

5 Move the pointer over the head of the blue arrow of the manipulator, and drag back to position the components.

The components are spaced automatically.



You can attach a component to existing explode steps. For example, you can add the retaining ring to the explode step with the gear and cover.

***To attach a component to an existing explode step:***

1 In the PropertyManager, under **Explode Steps**, right-click **Chain1** and select **Edit Step**.

Under **Settings**, the components or **Chain1** are listed in **Component(s) of the explode step** .

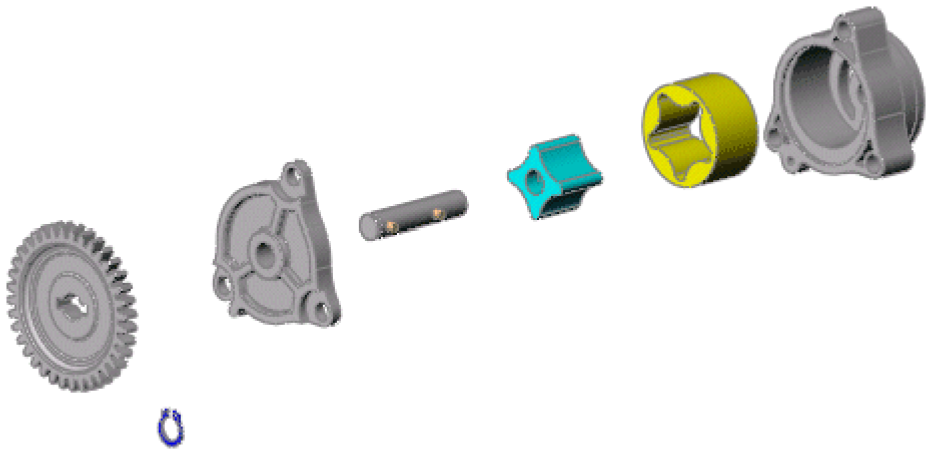
2 Select the retaining ring.

**Retaining Ring<1>** is added to **Component(s) of the explode step** .

3 Click **Apply** to preview the change, then click **Done**.

The retaining ring maintains its downward explode direction, but now also follows the forward explode direction of **Chain1**.

4 Click **OK** .



In SolidWorks 2005, collapse and explode the view in the same way as in previous releases. Right-click in the graphics area and select **Collapse**, or see the *SolidWorks Online User's Guide* for information on **Explode**, **Animate Explode** and **Animate Collapse**.

## External References

You now have the ability to *not* create external references when designing in the context of an assembly. If you select **Do not create external references**, no in-place mate is created when you create a new component. Also, external references are not created when you reference the geometry of other components, such as when you use **Convert Entities** or **Offset Entities**, or extrude **Up to Vertex** of another component.

***To specify that external references not be created:***

Click **Do not create external references** on the Assembly toolbar.

- or -

Click **Tools**, **Options**, **System Options**, **External References**, and select **Do not create references external to the model**.

---

You can switch **Do not create external references** off and on as you work in an assembly, to create external references for some components, but not for others.

---

## General Enhancements

### Assignable Mass Properties

You can now assign mass properties to a part or assembly. This is useful when you create a simplified representation of a component, and want to assign the correct mass to the component. See **Mass Properties** on page 4-12.

### Managing Files in a Shared Environment

The **Reload** tool on the Standard toolbar brings up a new dialog box for managing files in a shared environment. You can reload specific components of an assembly, and you can determine if read-only files in your session have been modified on disk or are available for write. See **Reload** on page 1-15.

### Creation and Management of Assembly Component Configurations

You can now create and manage configurations for components (sub-assemblies and parts) from the FeatureManager design tree of the top level assembly. See **Creation and Management of Assembly Component Configurations** on page 8-4.

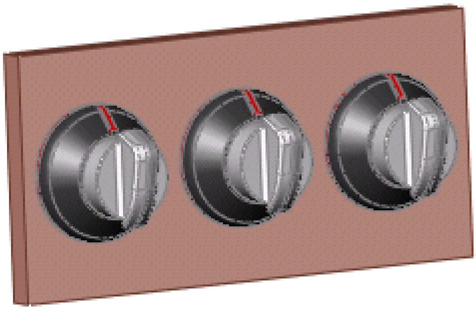
## Flexible Sub-Assemblies

You can create flexible sub-assemblies without having to create separate configurations in the sub-assembly file. Previously, for each instance of the sub-assembly where you wanted a different position of components, you had to create a separate configuration. Now, you can change a sub-assembly from **Rigid** to **Flexible** from the parent assembly without creating configurations in the sub-assembly. You can insert more than one instance of the sub-assembly in the same configuration of the parent assembly. You can make some instances rigid and others flexible, and the different instances can have different positions of the sub-assembly components.

***To create flexible sub-assemblies:***

1 Open **\assemblies\Control Panel\control_panel.sldasm**.

The assembly contains three instances of sub-assembly **knob_and_base.sldasm**. Note that you cannot rotate the knobs on the sub-assemblies.

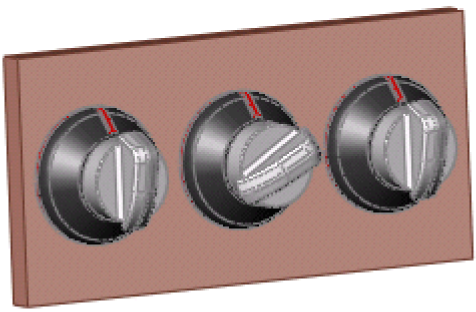


2 In the FeatureManager design tree, right-click the second instance of the sub-assembly (**knob_and_base<2>**), and select **Properties**.

3 In the dialog box, under **Solve as**, select **Flexible**, then click **OK**.

The sub-assembly is now flexible. In the FeatureManager design tree, the icon for the assembly changes to .

4 In the graphics area, rotate the knob of **knob_and_base<2>**.



## Interference Detection

Interference detection has been enhanced so you can:

- Display the true volume of interference as a shaded volume.
- Change the display settings of the interfering and non-interfering components to see the interference better.
- Select to ignore interferences that you want to exclude, such as press fits, interferences of threaded fasteners, on so on.
- Choose to include interferences between bodies within a multibody part.
- Choose to treat a sub-assembly as a single component, so that interferences between the sub-assembly's components are not reported.
- Distinguish between coincidence interferences and standard interferences.

Previously, multiple interferences between a pair of parts were sometimes reported as one interference. In SolidWorks 2005, each interference volume is reported separately. This can result in more interferences being reported.

***To check for interferences in an assembly:***

1 Open **lens_frame.sldasm**.

2 Click **Interference Detection** on the Assembly toolbar, or click **Tools**, **Interference Detection**.

3 In the PropertyManager:

- Under **Selected components**, leave **lens_frame.sldasm** in **Components to Check**.
- Under **Options**, select **Make interfering parts transparent**.
- Under **Non-interfering Components**, select **Use current.**

4 Under **Selected Components**, click **Calculate**.

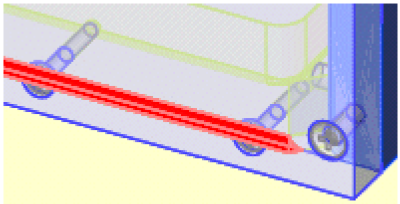
Four interferences are detected, and are listed under **Results**. The volume of each interference appears to the right of each listing.

When you select an interference under **Results**, it highlights in red in the graphics area.

***To examine the results:***

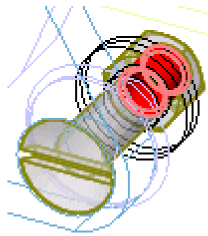
1 Under **Results**, select the long horizontal interference, then click ⊞ to expand.

The interference is between the bottom corner of **frontplate** and a fillet on **baseplate**. This is a valid interference, which can be fixed later by adding a chamfer to the edge of **frontplate**.



2 Under **Results**:

- Select another interference, and click ⊞ to expand.
- Right-click the interference and select **Zoom to selection**.



3 Under **Non-interfering Components**, select **Wireframe**.

The interference is between a countersunk bolt and a hex nut. Examination of the interference reveals that the interference is between the threads of the two components. The bolt and nut are specified correctly, so you want to ignore this interference.

4 Under **Results**, with the interference still selected, click **Ignore**.

The interference disappears from the **Results** list.

5 Repeat steps 2 and 4 for the remaining interferences between a bolt and a nut.

---

If you want to show ignored interferences in the **Results** list, select **Show ignored interferences** under **Options**.

---

6 Click **OK** to close the PropertyManager.

# 8

# Configurations

This chapter describes enhancements to configurations in the following areas:

- Configurable tolerances
- Creation and management of assembly component configurations
- Configurable materials
- Flexible sub-assemblies

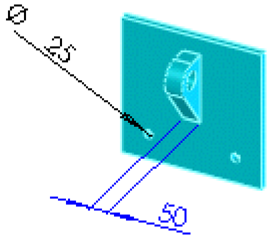
## Configurable Tolerances

Tolerances assigned to dimensions are now configurable in the **Dimension** PropertyManager and design tables.

***To configure a tolerance in the Dimension PropertyManager:***

1 Open **\configurations\bracket_plate.sldprt**.

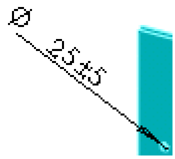
The part has two configurations, **rough** and **machined**, that have identical dimensions. **Rough** is the active configuration.




2 In the graphics area, select the diameter dimension, **25**.

3 In the PropertyManager, under **Tolerance/Precision**:

- Select **Symmetric** in **Tolerance Type** .
- Enter **5** for **Maximum Variation** .
- Click **Configurations**, select **This configuration**, then click **OK**.




4 Click **OK** .

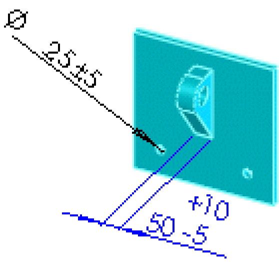
5 In the graphics area, select the width dimension **50**.

6 In the PropertyManager, under **Tolerance/Precision**:

- Select **Bilateral** in **Tolerance Type** .
- Enter **10** for **Maximum Variation** .
- Enter **5** for **Minimum Variation** .
- Click **Configurations**, select **This configuration**, then click **OK**.

7 Click **OK** .

The tolerances appear in the **rough** configuration, but not in the **machined** configuration.



*To configure tolerances in a design table:*

1 Click **Design Table** on the Tables toolbar, or click **Insert**, **Design Table**.

2 Under **Source**, select **Auto-create**.

3 Click **OK** .

A **Design Table** is created that includes the following tolerances for the **rough** configuration:

- For the width dimension: **$TOLERANCE@D1@Extrude2**, with a value of **BILATERAL;10.000000;-5.000000**.
- For the diameter dimension: **$TOLERANCE@D1@Sketch4**, with a value of **SYMMETRIC;5.000000**.

The values for the **machined** configuration are **NONE**.

4 Type values for **machined**:

- For **$TOLERANCE@D1@Extrude2**, type **Symmetric;5**.
- For **$TOLERANCE@D1@Sketch4**, type **Symmetric;2**.

5 In the design table, click in cell E2.

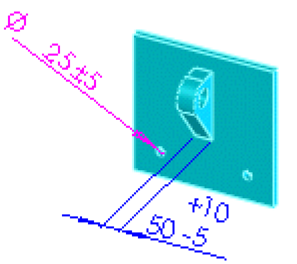
6 In the graphics area, double-click the diameter dimension, **25**.

A column is added to the design table for parameter **D1@Sketch4**, with a value of **25** for **rough**.

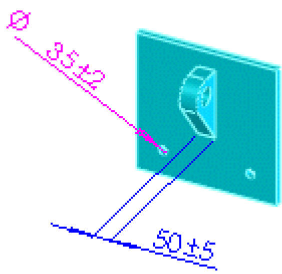
7 For **machined**, type **35**.

8 Click in the graphics area to close the design table.

The dimension and tolerance values for the two configurations update in the graphics area.




**rough** configuration




**machined** configuration

## Creation and Management of Assembly Component Configurations

You can create configurations of assembly components from the FeatureManager design tree, without opening the assembly component in its own window. The configurations are still stored in the file of the component for which the configuration was created.

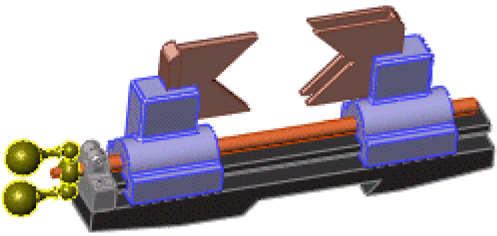
You can also switch configurations of a component from the FeatureManager design tree of the assembly. You can switch configurations for more than one component at a time if they have configurations with the same name.

To create or switch configurations, you must be editing the top-level assembly. The component can be at any level in the FeatureManager design tree.

***To switch a component to another configuration:***

1 Open **\configurations\vise\vise.sldasm**.

2 If the **Open External Referenced documents** dialog box appears, select **Open any documents which have changed**, then click **OK**.



3 In the FeatureManager design tree, hold down **Ctrl** and select **jaw1** and **jaw2**.

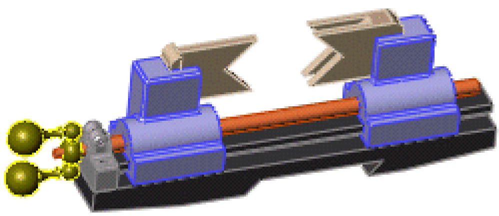
4 Right-click and select **Properties**.

5 In the dialog box, under **Referenced configuration**, select **Use named configuration**.

6 Select **Small**.

7 Click **OK**.

The configurations of **jaw1** and **jaw2** change to **Small** in the graphics area and in the FeatureManager design tree.



***To add configurations to components:***

1 In the FeatureManager design tree:

   a) Expand **rest handle**.

   b) Right-click **connector**, and select **Add Configuration**.

2 In the PropertyManager:

   a) Under **Configuration Properties**, type **Long** in **Configuration name**.

   b) Under **Advanced Options**, select **Use configuration specific color**.

   c) Click **Color**.

3 In the dialog box, select red, then click **OK**.

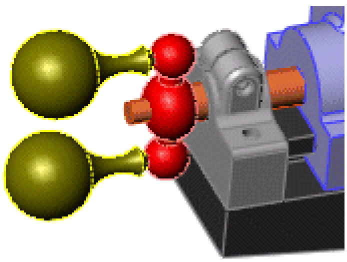
4 Expand **Parent/Child Options**.

   A simplified tree displays **connector** and all related components from the same branch of the FeatureManager design tree.

5 Select **vise**, **rest handle**, and **connector**.

6 Click **OK**.

   New configurations named **Long** are created in **connector.sldprt**, **rest handle.sldasm**, and **vise.sldasm**, and become the active configurations in the FeatureManager design tree. The connector changes to red in the graphics area. You can edit **connector.sldprt** later to define different dimensions and properties in the new configuration.



## Configurable Materials

You can specify a different material for each configuration of a part. When you specify a material for a part, you can select whether the material applies to **All configurations**, **This configuration**, or **Specified configurations**.

## Flexible Sub-Assemblies

In assemblies, you can create flexible sub-assemblies without creating separate configurations in the sub-assembly file. See **Flexible Sub-Assemblies** on page 7-7.

9

# Drawings

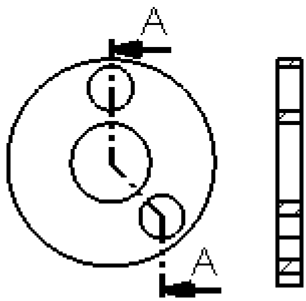
This chapter describes enhancements to drawings in the following areas:

- Aligned section views
- Comparing drawings
- Crop views
- Detached drawings
- Detail views
- Lightweight drawings
- OLE objects
- Options
- Projected views
- Section views
- User interface

## Aligned Section Views

You can use more than two lines to create an aligned section view. The lines must be connected at an angle and cannot form multiple contours.

To create an aligned section view with more than two lines, you must sketch the lines first, then click **Aligned Section View** .




## Comparing Drawings

Use **DrawCompare** to compare all entities between two drawing documents. The differences between the drawings are displayed in color codes. Click **Tools**, **DrawCompare** to access the tool.

## Crop Views

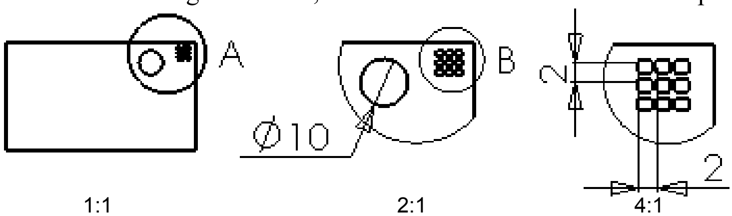
The **Crop View** menu item has been removed from the **Tools** menu, but it is still available at **Insert**, **Drawing View**, **Crop**.

## Detached Drawings

When you load a model in a detached drawing view, all unsuppressed models in the drawing document are loaded (including models on different sheets), regardless of whether the models are related. This ensures that all models are updated when you make a change.

## Detail Views

You can create a detail view of a detail view or of a crop view. The second detail view uses the scale relative to the original model, not relative to the first detail or crop view.



## Lightweight Drawings

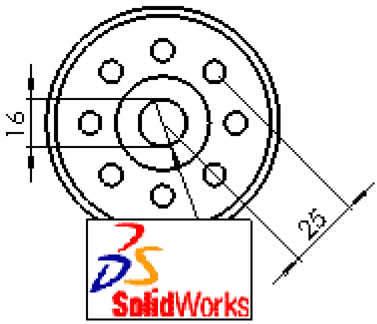
When you select a lightweight component in a drawing, it is no longer set to resolved. In previous versions of the SolidWorks software, the component became fully resolved.

## OLE Objects

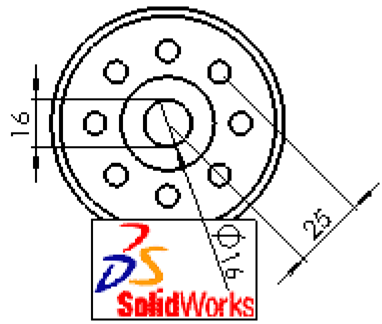
When you insert an OLE object into a drawing, the object is placed:

- On top of geometry
- Beneath annotations, tables, and sketches

In previous versions, OLE objects were placed on top of geometry, annotations, and so on.




SolidWorks 2004: Image covers the model and a dimension




SolidWorks 2005: Image covers the model, not the dimension

## Options

In **Tools**, **Options**, **System Options**, **Drawings**, new and deleted options include:

| Option | Status | Notes |
|---|---|---|
| **Keyboard movement increment** | New | Specifies the unit value of movement when you use the arrow keys to move (nudge) drawing views, annotations, or dimensions. |
| **Display drawing view borders** | Deleted | See **Drawing Views** on page 9-6. |
| **Dynamic drawing view activation** | Deleted | The view closest to the pointer is activated automatically when you create entities such as sketches, annotations, or dimensions. |

# Projected Views

You can create an isometric projected view by moving the pointer to a corner of the parent view before placing the view.

# Section Views

## Exclude Fasteners from Section Views

When you create a section view of an assembly drawing, you can exclude fasteners from being sectioned. Fasteners include any item inserted from SolidWorks Toolbox (nuts, bolts, washers, and so on). You can also designate any component as a fastener so it will not be sectioned.

***To exclude fasteners from a section view:***

1 Open **drawings****testAssem1.slddrw** from the **File Explorer**.

2 In **Drawing View2**, select the sketch line.

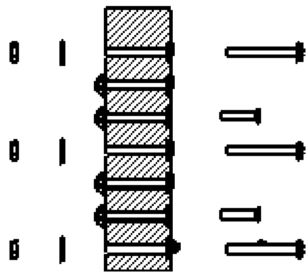
3 Click **Section View** on the Drawing toolbar, or click **Insert**, **Drawing View**, **Section**.

4 In the dialog box, select **Exclude fasteners**.

Note that you can also select **Flip direction** to change the orientation of the view.

5 Click **OK**.

6 Click in the drawing sheet to place the section view.



The view shows a section view of the part but not of the fasteners.

To designate any component as a fastener, open the component and click **File**, **Properties**. In the dialog box on the **Custom** tab, select **IsFastener** in **Property Name**, and type **1** for **Value / Text Expression**.

## Section Views and Aligned Section Views

You can pre-select sketch entities that belong to the drawing sheet to create section views and aligned section views. The sketch entities do not have to belong to an existing drawing view.

Additionally, you can create section views or aligned section views from:

- Crop views
- Detail views
- Exploded views (the exploded view must be an orthographic view)

# User Interface

## Drawing Sheets

### Adding Drawing Sheets

When you add a new drawing sheet to an existing drawing, the **Sheet Properties** dialog box no longer appears. Instead, the new sheet properties are based on the active sheet.

### Delete

To delete a drawing sheet, right-click a sheet tab or sheet icon in the FeatureManager design tree and select **Delete**. You can also right-click anywhere on the sheet (except in a drawing view) and select **Delete**.

To prevent accidentally deleting a drawing sheet, you can no longer delete a sheet by clicking an empty area of the sheet and pressing **Delete**.

### Lock Sheet Focus

Double-click anywhere in a drawing sheet (except in a drawing view) to toggle between **Lock Sheet Focus** and **Unlock Sheet Focus**.

### Re-order

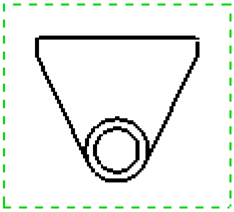
You can re-order drawing sheets by dragging:

- Sheet names in the FeatureManager design tree.
- Sheet tabs at the bottom left corner of the drawing document. Drag a tab (the pointer changes to ) to another tab. To re-order multiple sheets, hold down **Ctrl** to select the tabs, then release **Ctrl** before you drag.

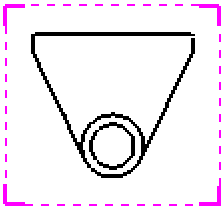
# Drawing Views

## Borders

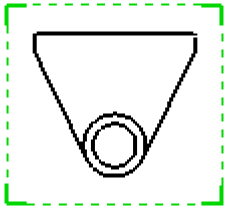
**Border colors**. New border colors indicate the different states of a drawing view. Detached drawings no longer have blue borders.



Green dotted line indicates the view is selected.



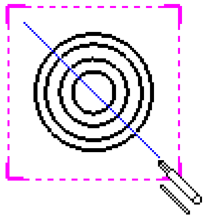
Pink dotted line with solid corners indicates the view is locked.



Green dotted line with solid corners indicates the view is selected and locked.

Empty views and predefined views have a dotted black line border when no model geometry is present. After a model is inserted into the view, the borders disappear.

**Dynamic border activation**. When you create a sketch, dimension, or annotation that belongs to a drawing view, the border highlights in a pink dotted line with solid corners.



**Parent and child view borders**. When you click any of the following, the parent view borders are highlighted:

- Child views
- Section lines
- Detail circles

**Resizing borders**. You can no longer resize a drawing view border. The borders are tightly fitted around the view by default. If you add sketch entities to a drawing view, the border automatically resizes to include these items. The border does not resize to include dimensions or annotations.

Drawing view borders from previous releases of the SolidWorks software do not automatically resize.

## Component and View Commands

When you right-click model geometry in a drawing view, the shortcut menu shows component and view information. Previously, only component information was shown.

## Creating Child Views

If you create a child view in a drawing with only one drawing view, the SolidWorks software automatically creates the child view from the existing view, regardless of whether it was selected.

Additionally, when you create any type of child view (section view, detail view, and so on), you do not have to pre-select an existing view before you click the drawing view tool.

## Deleting Views

To delete a drawing view, you can:

- Select model geometry in a drawing view and press **Delete**.
- Right-click model geometry in a drawing view and select **Delete**.

With either method, you are asked to confirm the deletion.

## Lock View Focus

Double-click a drawing view to toggle between **Lock View Focus** and **Unlock View Focus**.

## Lock View Position

After you position a drawing view, right-click anywhere in the drawing view and select **Lock View Position** to fix the drawing view in place. To release the drawing view, right-click the drawing view and select **Unlock View Position**.

## Moving Drawing Views

You can click and drag many entities (including edges, vertices, cosmetic threads, and so on) in a drawing view to move the view. The pointer includes the pan icon, , to indicate that you can use the selected entity to move the view. You can also select a drawing view, then move (nudge) it with the arrow keys. See **Options** on page 9-3.

## Names

You can rename drawing views. In the FeatureManager design tree, click-pause-click the drawing view name and then type the new text.

## Overlapping Drawing Views

If you select a drawing view that overlaps another drawing view, the view whose center is closest to the pointer is selected.

### PropertyManagers

Click anywhere in a drawing view (including model geometry) to display the drawing view PropertyManager. Previously, you had to click within the drawing view, but outside the model geometry to activate the drawing view PropertyManager.

In drawing view PropertyManagers, you can select a predefined scale and assign it to the view.

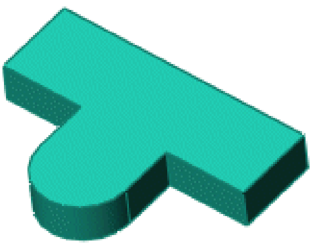
### Selection

When you box-select one or more drawing views that do not have dimensions or annotations, the drawing views are selected. Otherwise, the dimensions and annotations are selected, not the drawing views.

## Edges

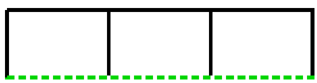
When you select an edge in a drawing, the entire line is highlighted if all of the line segments are collinear. This behavior does not apply to components in an assembly drawing that are mated to form a collinear edge.

To override this behavior, hold down **Shift**, then click an edge.



Original part

SolidWorks 2004: Only the selected edge is highlighted



SolidWorks 2005: Entire collinear edge is highlighted

## FeatureManager Design Tree

New icons in the FeatureManager design tree identify the drawing item:

| | |
|---|---|
| Aligned Section View | Empty View |
| Alternate Position View | Horizontal Break |
| Auxiliary View | Model View |
| Blocks | Predefined View |
| Broken-out Section View | Projected View |
| Crop View | Section View |
| Detail View | Vertical Break |

## Sketches

### Selection

You can select a sketch anywhere in the drawing - the sketch does not need to belong to a drawing view. The drawing view does not need to be active or have locked focus.

### Snapping and Inferencing

When you drag a sketch point in a drawing, you can snap or infer to all visible sketch points on the sheet.

## Toolbars

Two new buttons have been added to the Drawing toolbar:

- **Horizontal Break**
- **Vertical Break**

Previously, these tools were available from the **Insert** menu only.

10

# Detailing

This chapter describes enhancements to detailing in the following areas:

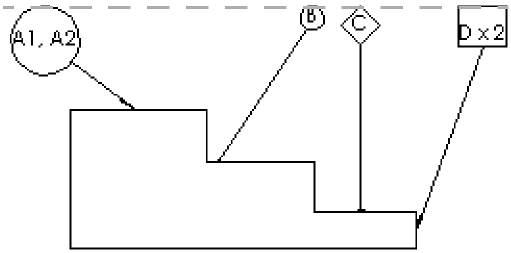
- ❑ Annotations
- ❑ Autodimension
- ❑ Bills of materials
- ❑ Design Library
- ❑ Insert model items
- ❑ Options
- ❑ Tables
- ❑ Toolbars

# Annotations

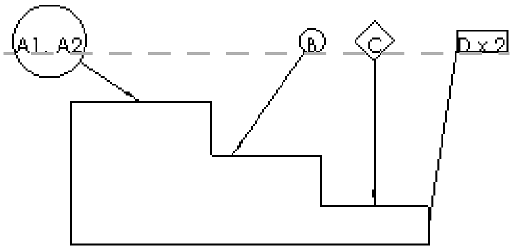
## Alignment

Annotations are aligned based on the baseline of the text. Previously, leaders and text boxes were used to align annotations.

When you select **Space Evenly Across** and **Space Evenly Down**, annotations are spread evenly across based on the first and last selected annotation in the group. Previously, annotations were spread based on the distance between the first and second entities in the group.




SolidWorks 2004: Alignment based on the top of bounding boxes




SolidWorks 2005: Alignment based on the baseline of the text

## Area Hatch/Fill

The **Area Hatch/Fill** dialog box has moved to the PropertyManager. All previous functionality is retained. Additionally:

- You do not have to select an area to fill before using the **Area Hatch/Fill** tool.
- You can click in a closed region bounded by model edges or sketch entities to create an area hatch or fill that covers the entire region. Previously, you had to select individual boundaries and you could not apply an area hatch or fill to a region bounded by a combination of model edges and sketch entities.
- If you change the region boundaries, (for example, if you add a sketched rectangle that you want to exclude from the area hatch or fill), you can right-click and select **Recreate Area Hatch** to update the area.
- When you select an area hatch or fill pattern, the **Area Hatch/Fill** PropertyManager appears. A new option, **Apply changes immediately**, updates the drawing view immediately with your changes to the pattern. Otherwise, click **Apply** to update the drawing view when you want. This improves performance time.

**Apply** is available only when **Apply changes immediately** is cleared.

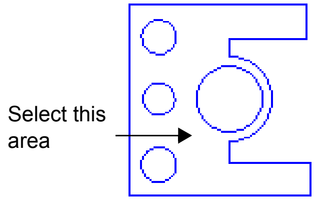
***To add an area hatch pattern to a drawing:***

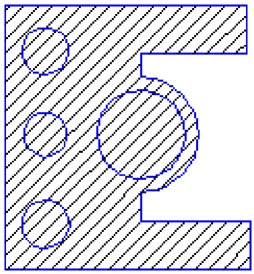
1 Open **detailing****crosshatch.slddrw** from the **File Explorer** .

2 Click **Area Hatch/Fill** on the Annotations toolbar, or click **Insert**, **Annotations**, **Area Hatch/Fill**.

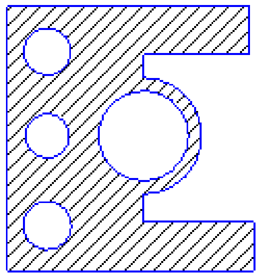
3 In the drawing, select the area as shown.

4 Click **OK** .



SolidWorks 2004



SolidWorks 2005

## Blocks

You can define leader attachment points for all instances of a block. Additionally, the **Definition** section of the **Block Instance** PropertyManager contains the following functionality:

- **Edit**. Accesses the **Block Definition** PropertyManager.
- **Select a block definition**. Allows you to change the selected block to a different block definition. You do not have to delete a block, then insert a new block to change the definition.

***To define a leader attachment point in a block:***

1 Open **detailing****block_with_leader.slddrw** from the **File Explorer** .

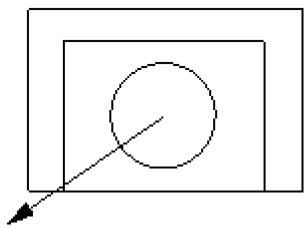
2 Select the block in the graphics area.

3 In the PropertyManager, under **Definition**, click **Edit**.

The **Block Definition** PropertyManager appears.

4 In the PropertyManager, under **Leader Point**, set X coordinate to **62** and set Y coordinate to **34**. The leader point is indicated by .

5 Click **OK** .



## Cosmetic Threads

Cosmetic threads are indicated by the icon in the FeatureManager design tree. If the cosmetic thread is hidden, it appears as .

You can use the option, **Display precise threads**, to check all cosmetic threads to determine if they should be visible or hidden. See **Cosmetic Threads** on page 10-18.

## Datum Target Symbols

When you select **No target symbol** in the **Datum Target** PropertyManager, you place the symbol in the graphics area with one click. Previously, two clicks were required to place the symbol.

## Dimensions

### Alignment

You can now use the tools on the Align toolbar to align dimensions.

### Configurable Tolerances

When you apply a tolerance to a dimension, you can apply the tolerance to specific configurations. See **Configurable Tolerances** on page 8-2.

### Display

When you insert or edit a dimension in a drawing, you can select display options (**Parenthesis**, **Dual dimension**, or **Inspection dimension**) directly from the PropertyManager.

### Modifying Dimensions

You can use the **Modify** dialog box to access the **Add Equation** or **Shared Values** dialog boxes. See **Modifying Dimensions** on page 4-3.

### Overrides

You can add tolerances to a dimension that you have overridden. In previous releases, you could not override a dimension value and add a tolerance.

***To add a tolerance to an overridden dimension:***

1 Open **detailing****hole_callouts.slddrw** from the **File Explorer** .

2 Select the dimension, **96**.

3 In the PropertyManager, under **Primary Value**:

   a) Select **Override value**.

   b) Type **95** for the dimension value.

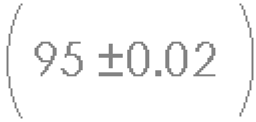
4 Under **Tolerance/Precision**:

a) Select **Symmetric** in **Tolerance Type**.

b) Type **0.02** for **Maximum Variation**.

5 Click **OK**.

The dimension is overridden and includes a tolerance value. If you clear **Override value**, the dimension returns to its original value but retains the tolerance.




## Geometric Tolerance Symbols

The dialog box for geometric tolerance symbols offers selections based on the symbol you choose. Only the attributes that are appropriate for the selected symbol are available. For example, if you select the concentric symbol, only the diameter attribute is available. Additionally, symbol selection is easier because you can choose from symbol images rather than their names.

### *To insert a geometric tolerance symbol:*

1 Open a new drawing.

2 Click **Geometric Tolerance** on the Annotation toolbar, or click **Insert**, **Annotations**, **Geometric Tolerance**.

3 In the dialog box:

a) Select **Position** ⌖ in **Symbol**.

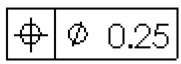
b) Type **0.25** for **Tolerance 1**.

c) Click **Diameter** ∅.

4 In the PropertyManager, select favorites, leaders, formatting, or layer properties as necessary.

5 Click in the graphics area to place the symbol.

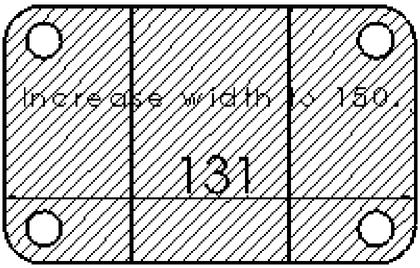
6 Click **OK** in either the dialog box or the PropertyManager.



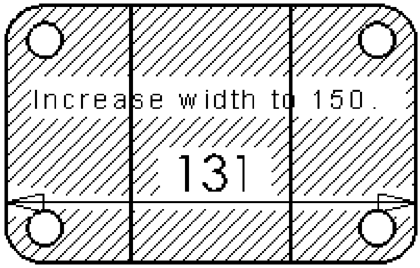
## Halos

When you insert an annotation that is on top of a crosshatch or area hatch pattern, a halo of space appears around the annotation so it is easier to read. The halo also surrounds leaders, dimensions, and dimension lines.

Halos appear only if the annotations belong to the drawing view.




SolidWorks 2004




SolidWorks 2005

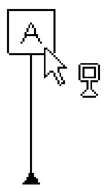
## Hide/Show Annotations

You can use the **Hide/Show Annotations** tool to hide or show the following annotation types:

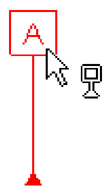
- Auto Balloons
- Balloons
- Bill of Materials
- Blocks
- Center Marks
- Centerlines
- Cosmetic Threads
- Datum Features
- Datum Targets
- Dowel Pin Symbols
- Geometric Tolerances
- Hole Tables
- Multi-jog Leaders
- Notes
- Revision Symbols
- Revision Tables
- Stacked Balloons
- Surface Finish Symbols
- Weld Symbols
- Weldment Cut Lists

## Highlighting

When you drag the pointer over an annotation, it dynamically highlights. Previously, annotations did not highlight.




SolidWorks 2004




SolidWorks 2005

## Hole Callouts

You can modify hole callouts to include tolerances.

*To add tolerances to hole callouts:*

1 Open **detailing****hole_callouts.slddrw** from the **File Explorer** .

2 Select the hole callout.

3 In the PropertyManager, under **Tolerance/Precision**:

a) Select **Diameter (1.78)** in **Callout value**.

b) Type **0.02** for **Maximum Variation** .

c) Select **Depth (25.4)** in **Callout value**.

d) Select **Symmetric** in **Tolerance Type** .

e) Type **0.03** for **Maximum Variation** .

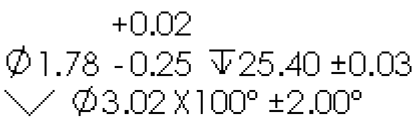
f) Select **CounterSink Angle (100°)** in **Callout value**.

g) Select **Symmetric** in **Tolerance Type** .

h) Type **2** for **Maximum Variation** .

4 Click **OK** .

The annotation is updated with the new tolerances.




## Library Features

If you insert an annotation into a library feature, the annotation is saved with that feature. For example, a surface finish symbol attached to a face remains attached to the same face and at the same relative location when you insert the feature.

When you insert the annotation into a library feature, either the annotation itself or its leader must touch the feature to be saved with the feature.

## Moving Annotations and Drawing Views

You can use the arrow keys to move (nudge) annotations or drawing views. See **Options** on page 9-3.

# Notes

## PropertyManager

The following functionality has been added to the **Note** PropertyManager:

- Annotations. You can insert the following annotations in a note: **Geometric Tolerance**, **Surface Finish**, and **Datum Feature**. See **Annotations in Notes** on page 10-10.
- Leader display. New display options are available when you attach a leader to a multi-line note:

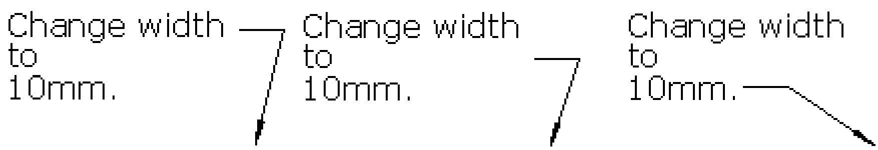
  **Attach Leader Top**

  **Attach Leader Center**

  **Attach Leader Bottom**

  **Attach Leader Nearest**

If you move the note to either side of the leader, the leader remains in the same justification, except if you select **Attach Leader Nearest**. This option attaches the leader to the top when the text is to the left of the leader, and attaches the leader to the bottom when the text is to the right of the leader.




Attach Leader Top | Attach Leader Center | Attach Leader Bottom

## Rich Text Formatting

Notes use rich text formatting so you can format each item in the note independently. Notes also support stacked text, bulleted and numbered lists, and so on.

***To apply rich text formatting to a note:***

1. Open **detailing****rich_text.slddrw** from the **File Explorer** .

   The note appears without rich text formatting.

2. Double-click the note.

   The note is active and the **Formatting** pop-up toolbar appears.

3. In the note:

   a) Double-click the word **Notes**, then click **Bold** on the **Formatting** pop-up toolbar.

   b) Highlight **1/32**, then click **Stack** on the **Formatting** pop-up toolbar.

   c) In the dialog box, under **Stack**, type **1** for **Upper** and type **32** for **Lower**.

   d) Click **OK**.

4. To create a numbered list, select the two lines under **Notes**, then click **Number** on the **Formatting** pop-up toolbar.

5. Click an empty area of the drawing sheet to close the note.

**NOTES**:
1. GRIND SHARP CORNERS.
2. ALL CAST FILLETS AND CORNER RADII 0 TO $\frac{1}{32}$.

## Annotations in Notes

When you insert an annotation into a note, you can either select an existing annotation in the drawing or create a new one in the **Note** PropertyManager.

***To insert an annotation from the Note PropertyManager:***

1 Open **detailing****note_symbols.slddrw** from the **File Explorer** .
2 Double-click the first note in the drawing.
3 Select **ZZZ**.
4 In the PropertyManager, under **Text Format**, click **Surface Finish** √.
5 In the **Surface Finish** PropertyManager, type **125** for **Minimum Roughness**.
6 Click **OK** .
7 Click **OK**  again to close the **Note** PropertyManager.
8 Keep **note_symbols.slddrw** open for the next procedure.

All machined surfaces to be 125√ unless otherwise specified.

To edit the annotation, double-click the annotation and edit it in the **Surface Finish** PropertyManager.

***To insert an annotation from a drawing:***

1 Open **detailing****note_symbols.slddrw** from the **File Explorer**  if you do not have it open from the previous procedure.
2 Double-click the second note in the drawing.
3 Select **ZZZ**.
4 Select the note | // | 0.05 | A | in the drawing sheet.
5 Click **OK** .

Tolerance zone: | // | 0.05 | A |

To edit the annotation, you must edit the existing annotation in the drawing sheet. You cannot edit annotations in notes if they were inserted from an existing annotation. When you edit the existing annotation, all instances of the annotation are updated in the drawing and the drawing sheets.

## Variables in Notes

When you edit a note that contains a variable, you can either show the variable name or show the contents of the variable. Click **View**, **Annotation Link Variables** to see the variable name.

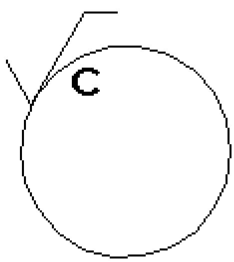
## Selection

All annotation tools are available whether you pre- or post-select an entity in the drawing.

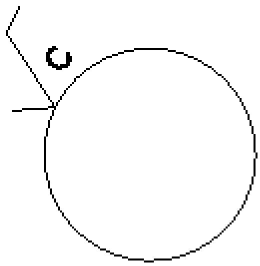
## Surface Finish Symbols

When you insert a surface finish symbol, the PropertyManager lists all symbols regardless of the dimensioning standard. Additionally, improvements to surface finish symbols make it easier to place the symbols exactly:

- You can rotate the symbols to any degree.
- Surface finish symbols are placed normal to an edge.



SolidWorks 2004: Surface finish symbol upright and rotated 90°



SolidWorks 2005: Surface finish symbol normal to edge

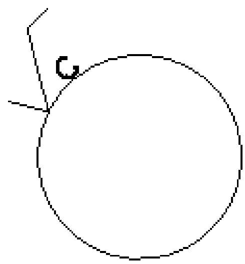
***To rotate a surface finish symbol:***

1 Open **detailing****surface_finish_symbol.slddrw** from the **File Explorer**.

2 Select the surface finish symbol in the drawing sheet.

3 In the PropertyManager, under **Angle**, set **Angle** to **45°**.

4 Click **OK**.



## Weld Beads

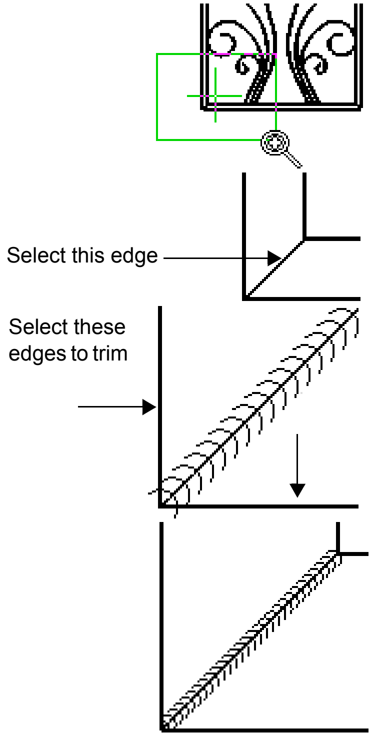
Two types of weld beads are now available: **Caterpillar** and **End Treatment**.

---

The button for **Weld Symbol** has changed to . The **End Treatment**  tool uses the old **Weld Symbol** button.

---

***To add weld beads to a weldment:***




1 Open **detailing****tree_gate.slddrw** from the **File Explorer** .

2 Zoom in to the lower left corner of the view as shown.

3 Click **Caterpillar**  on the Annotation toolbar, or click **Insert**, **Annotations**, **Caterpillar**.

4 In the drawing, select the edge as shown.

5 In the PropertyManager, under **Caterpillar**:

   a) Select **Full length** in **Bead type**.

   b) Set **Bead size**  to **5**.

6 In the graphics area, select the two edges as shown for **Trimming edges**. These edges define the boundaries for the weld bead.

7 Click **OK**  to accept the trimming edges.

8 Click **OK**  again to close the PropertyManager.

The caterpillar edges are trimmed at the corner and the weld bead is applied.

***To add an end treatment to the weldment:***

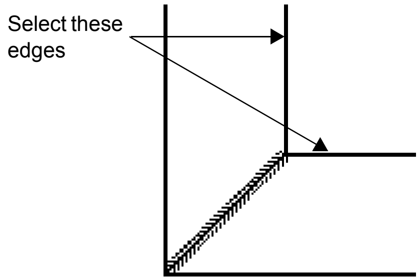
1 Click **End Treatment** on the Annotations toolbar, or click **Insert**, **Annotations**, **End Treatment**.

2 In the PropertyManager:

a) Select the two edges as shown for **Edges** .

b) Click **ISO type** .

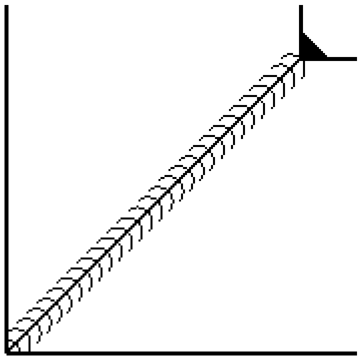
c) Click **Bead throat thickness** and set the value to **5**.




3 Click **OK** to accept the edges.

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

The end treatment is applied to the drawing.



5 Keep **tree_gate.slddrw** open for the next procedure.

---

If you create end treatments or intermittent or partial caterpillar symbols, you can display and change the dimensions of the welded and non-welded areas. If you change the dimensions, the symbol updates.

---

## Weld Symbols

When you insert a weld symbol, you can create multiple symbols and change them while the dialog box is open. You can also move the weld symbol within the model and place the leader and symbol as necessary.

***To insert a weld symbol:***

1 Open **detailing****tree_gate.slddrw** from the **File Explorer**  if you do not have it open from the previous procedure.

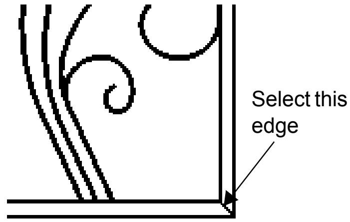
2 Click **Weld Symbol**  on the Annotations toolbar, or click **Insert**, **Annotations**, **Weld Symbol**.

3 In the dialog box:

a) Click the **Weld symbol** underneath the leader line. In the **Symbols** dialog box, select **Fillet** in the symbols list, then click **OK**.

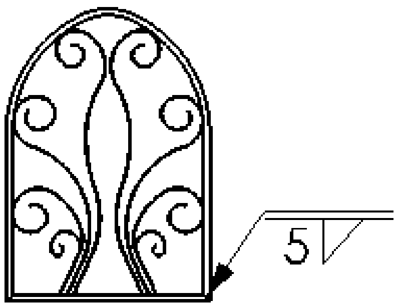
b) Type **5** in the box to the left of the **Weld symbol** button to define the size.

4 In the graphics area, select the lower right edge as shown to attach the leader. Notice that the weld symbol is attached to the pointer.




5 Click again to place the weld symbol.

6 Click **OK**.



## Autodimension

You can use the **Autodimension** tool to insert reference dimensions into drawing views as baseline, chain, and ordinate dimensions.

***To autodimension a drawing:***

1 Open **detailing****bushing.slddrw** from the **File Explorer**.

2 Click **Autodimension** on the Dimensions/Relations toolbar.

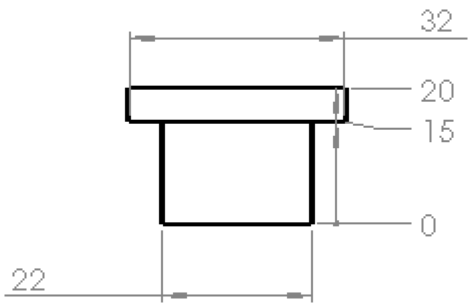
3 In the PropertyManager:

a) Under **Entities to Dimension**, select **All entities in view**.

b) Under **Horizontal Dimensions**, select **Baseline** in **Scheme** and set **Dimension placement** to **Above view**.

c) Under **Vertical Dimensions**, select **Ordinate** in **Scheme** and set **Dimension placement** to **Right of view**.

4 Click **OK**.



## Bills of Materials

### Custom Properties

The **Column** PropertyManager now uses the term **Custom Property** instead of **User Defined** to insert a custom property into a BOM.

### Equations and Formulas

Table-based bills of materials now support equations and formulas that were previously available in Microsoft Excel-based bills of materials only.

***To add an equation to a table-based bill of materials:***

1 Open **detailing****casting.slddrw** from the **File Explorer**.

The drawing has a bill of materials with a column labeled, **COST**.

2 Select the **COST** column by clicking when the pointer changes to.

3 In the **Column** PropertyManager, under **Column Properties**:

a) Select **Equation**.

b) Click **Equation Editor**.

4 In the dialog box:

a) Under **Columns**, double-click **QTY**.

b) Click * for multiply.

c) Under **Custom properties**, double-click **Price**. The value for **Price** is included in each part's custom properties.

d) Select **Allow extra characters in column and custom property value** to allow non-numeric text in each cell.

e) Click at the beginning of the equation. The pointer should be before the equation, 'QTY.'*`Price`.

f) Type **$** for **Text**, then click **OK** to add **"$"** to the equation. The equation appears as "$"'QTY.'*`Price`.

g) Click **OK**.

Each cell in the column lists the quantity multiplied by the price per part.

5 Select the last cell in the **COST** column.

6 In the **Cell** PropertyManager, select **Sum/Total Cell** and:

a) Type **$** for the box before **Sum**.

b) Click **OK**.

The sum of the **COST** column is displayed as **$80**.

---

If you insert a component below the **Sum** row, the sum does not update. You must insert a component above the **Sum** row for it to be included in the calculation.

---

## Design Library

Use the **Design Library** to see previews of saved annotations. You can also drag-and-drop annotations to and from the **Design Library**. DXF/DWG files are supported in the **Design Library**, but previews are available for DWG files only. See **Design Library** on page 1-4.

## Insert Model Items

The **Insert Model Items** dialog box has been replaced with the **Model Items** PropertyManager, which includes the following new functionality:

- Dimension placement. Select **Use dimension placement in sketch** to place model dimensions from the part in the same locations in the drawing.
- Feature selection. Insert model items for selected features.
- Layers. Insert model items for different layers of the drawing.
- Hide/show. Use the hide/show pointer while the PropertyManager is active. The left mouse button moves items, and the right mouse button hides/shows items. When the **Model Items** PropertyManager is displayed, hidden model items are gray.

Additionally, when the **Model Items** PropertyManager and the hide/show pointer are active, the following options are available:

- Delete. Use the **Delete** key to delete model items.
- Drag. Use the **Shift** key to drag model items to another drawing view.
- Copy. Use the **Ctrl** key to copy model items to another drawing view.

***To insert model items:***

1 Open **detailing\\tool_post.slddrw** from the **File Explorer**.

2 Click **Model Items** on the Annotation toolbar, or click **Insert**, **Model Items**.

3 In the PropertyManager:

a) Under **Dimensions**, select **Eliminate duplicates**.

b) Under **Import from**, click **Selected feature**.

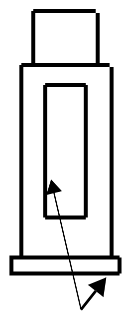
c) Under **Options**, select **Use dimension placement in sketch**.

4 In **Drawing View1**, select the two edges as shown.

5 Hover over any of the dimensions. The pointer changes to .

6 Hold down the **Shift** key and drag the dimension **74.61** to **Drawing View3**, located to the right of **Drawing View1**.

7 Click **OK**.



Select these edges

# Options

## Cosmetic Threads

A new option, **Cosmetic thread display**, is available in **Tools**, **Options**, **Document Properties**, **Detailing**. When you select **High quality**, this option checks all cosmetic threads to determine if they should be visible or hidden. For example, if a hole (not a through hole) is on the back of a model and the model is in a front view, the cosmetic thread is hidden.

System performance is slower with this option selected. It is recommended that you clear this option until you finish placing all annotations.

## Line Style

You can define custom line styles, then apply them with the **Line Style** or **Layer Properties** tools. In previous versions of the SolidWorks software, line styles were a system-wide property. Now they are document-based properties.

---

You can also set a location to save line styles in **Tools**, **Options**, **System Options**, **File Locations**. Select **Line Style Definitions** in **Show folders for** and set a file location.

---

***To create a line style:***

1 Open a new drawing.

2 Click **Options** on the Standard toolbar, or click **Tools**, **Options**.

3 On the **Document Properties** tab, click **Line Style**.

4 In the dialog box:

a) Click **New**.

b) Type **whatsnew** for the line style name, then press **Enter**. The name is not case sensitive.

c) Under **Line length and spacing values**, clear the default text.

d) Type **A,2,-2**, where **A** indicates a normal line, **2** creates a 2mm line width, and **-2** creates a 2mm gap between segments.

---

The line style units are based on the units used in the document.

---

5 Click **Save**.

6 In the dialog box:

a) Select **Save to the default user line style file**. This ensures that the line style is available for all documents, not just the current document.

b) Select the name of the new line style in the list, then click **OK**.

7 Click **OK** to close the **Document Properties - Line Style** dialog box.

8 Click **Line Style** on the Line Format toolbar.

The new line style is available.

# Toolbars

## Align Toolbar

### Align Between Lines

The **Align Between Lines** tool allows you to align annotations evenly between horizontal or vertical lines. Unlike the **Space Evenly Across** and **Space Evenly Down** tools, which require that you select at least three annotations, the **Align Between Lines** tool requires that you select only one annotation and two horizontal or vertical lines.

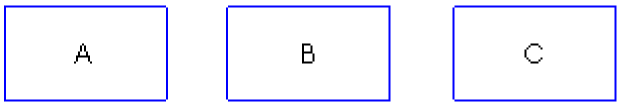
***To align annotations between lines:***

1 Open **detailing\align_between_lines.slddrw** from the **File Explorer**.

2 Select all entities in the graphics area.

3 Click **Align Between Lines** on the Align toolbar, or click **Tools**, **Align**, **Align Between Lines**.

The annotations are aligned within each rectangle.




### Grouping

You can select multiple annotations and group them so they move together when you drag them. Similarly, you can ungroup annotations so they move independently. Limitations of grouping include:

- If you hold down **Shift** and drag a dimension in a group, you are asked if you want to dissolve the group so the annotations move independently.
- If you hold down **Alt** and drag an annotation in a group, the annotation moves independently.
- If you select an annotation within a group, then create a block, only the selected annotation is included in the block. Grouped annotations are automatically ungrouped when you create a block. You must hold down **Ctrl** and select individual annotations (even if they are grouped) to include them in a block.
- When you edit the definition of a block, you cannot create a group of entities within the block.

***To group annotations:***

1 Open **\detailing\axle_support.slddrw** from the **File Explorer**.

2 Hold down **Ctrl** and select the two dimensions and note.

3 Click **Group** on the Align toolbar, or click **Tools**, **Align**, **Group**, **Group**.

4 Click an empty area of the drawing to clear all of the selections.

5 Drag any annotation in the group.

The annotations move together as a single entity.

6 Again, click an empty area of the drawing to clear all of the selections.

7 Hold down **Alt** and drag the dimension **79**.

The dimension moves independently of the group.

8 Release **Alt**, then drag any of the annotations in the group.

The annotations move together as a single entity.

To ungroup the annotations, select an annotation in the group and click **Ungroup** on the Align toolbar, or click **Tools**, **Align**, **Group**, **Ungroup**.

### Moved Tools

The following tools have moved from the Drawing toolbar to the Align toolbar:

- **Align Collinear/Radial** .
- **Align Parallel/Concentric** .

### Name Changes

The following tools on the Align toolbar have changed names:

| Old Tool Name | New Tool Name |
|---|---|
| **Uppermost** | **Align Top** |
| **Leftmost** | **Align Left** |
| **Rightmost** | **Align Right** |
| **Lowermost** | **Align Bottom** |
| **Center Horizontally** | **Align Horizontal** |
| **Center Vertically** | **Align Vertical** |
| **Compact Horizontally** | **Space Tightly Across** |
| **Compact Vertically** | **Space Tightly Down** |

To access these tools on the menu, click **Tools**, **Align**, and select the tool.

## Annotation Toolbar

The **Revision Symbol** tool has been added to the Annotation toolbar.

The table buttons (**Hole Table**, **Bill of Materials**, and so on) have moved from the Annotation toolbar to the Table toolbar.

## Formatting Toolbar

The Font toolbar has been renamed to the Formatting toolbar.

## Table Toolbar

The Table toolbar includes the **Hole Table**, **Bill of Materials**, **Excel based Bill of Materials**, **Revision Table**, **Design Table**, and **Weldment Cut List** tools. These were moved from the Annotations toolbar.

11

# Library Features

This chapter describes enhancements to library features in the following areas:

- General enhancements
- Design Library
- Adding library features
- FeatureManager design tree
- PropertyManager
- Library features and links
- Library features for machine design

## General Enhancements

Enhancements include the following capabilities:

- Select a configuration while inserting the library feature into a part.
- Link to parent part.
- Edit by changing configurations, selecting a different position, and so on.
- Add descriptions to references when saving the library feature.
- Add annotations to a library feature, and have the annotations to the feature inserted into the part with the library feature.
- Store a helix as a library feature.
- Flip the sketch orientation during insertion of a library feature by clicking the direction arrow in the graphics area.
- Transfer visual properties such as textures specified in the library feature to the inserted features.

### Design Library

The **Design Library** has replaced the **Palette Features** window and the menu option (**Insert**, **Features**, **Library Features**). The **Design Library** now controls all library feature functions including:

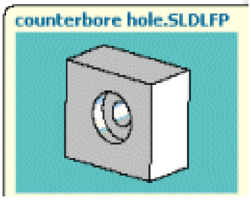
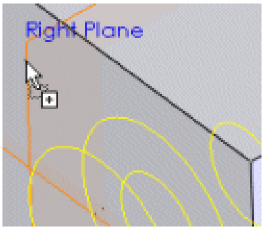
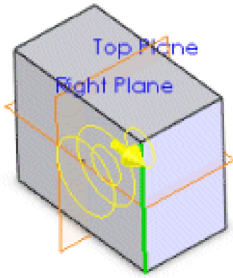
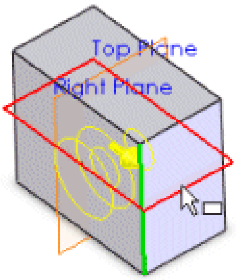
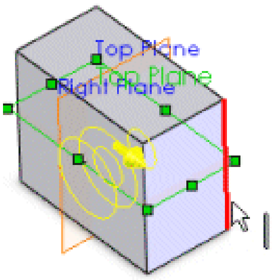
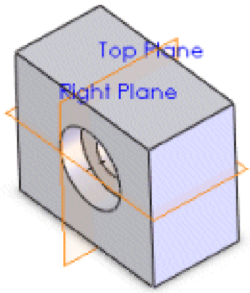
- Display of the library features and subfolders that contain library features.
- Preview of the library feature parts.
- Insertion of the library features on the face of a part or on a plane in the graphics area.

The **Design Library** includes multiple top-level folders, such as **annotations**, **assemblies**, and **forming tools**.

## Library Features on a Plane

You can drag library features to a plane, or anywhere in the graphics area. If you drag a library feature to the graphics area, you are prompted to select a plane. To place a library feature on a plane, select the edges or the label.

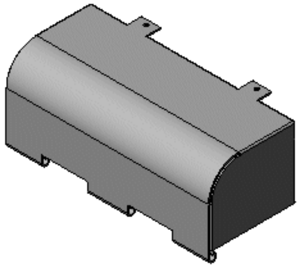
### Drag a Library Feature to a Plane

| Select library feature. | Drag the library feature to a plane and select an edge (or label) to position the sketch. | Select one or more References to position the sketch (see **References** on page 11-7). |
| --- | --- | --- |
|  |  |  |
| Use References to position the sketch. | | Library feature positioned. |
|  |  |  |

The arrow in the graphics area allows you to flip the sketch orientation when inserting the library part.

## Adding Library Features

***To add a library feature:***



1 Open **sheet_metal_cover.sldprt** from the **File Explorer** .

2 Click **Design Library** to display the **Design Library** Task pane.

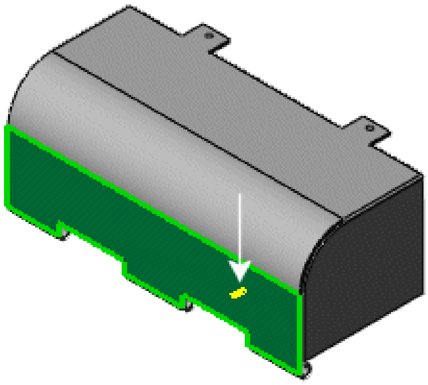
3 Click to pin down the **Design Library**.

4 Click **features**, **metric**, **slots**.

Slots appear in the bottom panel of the Task Pane.

5 Select the **straight slot**, and drag the slot to the bottom area of the flat **Base-Flange1**.

6 Position the slot just over the cut, between the second and third hems, as shown.

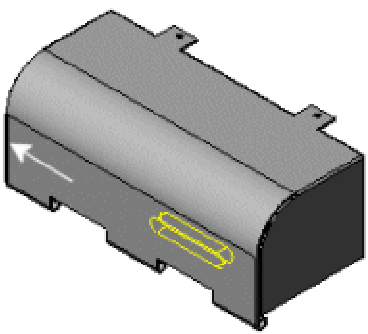


7 In the PropertyManager:

- Select **12.2 X 61** in **Configuration**.

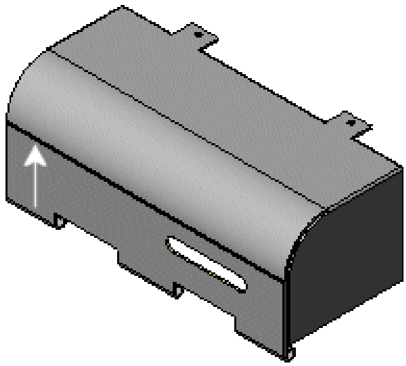
  **Locating edge1** highlights in **References**, and a popup window highlighting the reference to select appears, with the reference edge highlighted.

- Select the left edge of the **Base-Flange1**, as shown**.**



In the popup window, the top edge is highlighted, and **Locating edge2** is highlighted in **References**.

- Select the top edge of the **Base-Flange1**, just below the fillet, as shown.



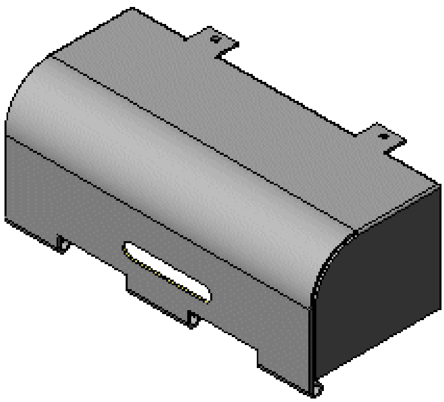
This positions the slot and allows you to modify the **Locating Dimensions**.

8 In the PropertyManager, under **Locating Dimensions**:

a) Select **Value**, and type **135** for **Locating dim1**.

b) Select **Value**, and type **32** for **Locating dim2**.

9 Click **OK**.



## Editing Library Features

Once you insert a library feature, you can change configurations, regardless of whether the inserted feature is linked to the parent part in the library. You can also change the references (see **References** on page 11-7).

# FeatureManager Design Tree

You can create library features by selecting some or all features from the FeatureManager design tree.

- Hold down **Ctrl** and select multiple features from a previously saved part, and drag the features from the FeatureManager design tree into a folder in the **Design Library**.
- Drag a folder with features from a previously saved part into the **Design Library**.

When you create a part that includes multiple extruded features that you want to save as a library feature, make sure that **Merge result** is selected. This ensures that all features are included when you drag the library feature to a model.

## Creating a Library Feature

***To save features as a library feature:***

1. Open **library_feature_save.sldprt** from the **File Explorer** .
2. Click **Design Library** to display the **Design Library** Task pane.
3. Click  to pin down the **Design Library**.
4. Select **parts**, **hardware**.

   The bottom panel displays samples from the **hardware** folder.
5. In the FeatureManager design tree, hold down **Ctrl** and select the following:
   - **Extrude1**
   - **Fillet1**
   - **CutExtrude2**
6. Hold down **Ctrl** and drag the features to the lower pane of the Task Pane.

   This opens the **Save As** dialog box, with the **.sldlfp** extension already selected.
7. Save the library feature as **model1.sldlfp**.
8. Click **Yes** in the dialog box.

   This saves **model1.sldlfp** in the **hardware** folder of the **Design Library**, where you can see a preview of the library feature.

You can save a library feature by selecting the features and clicking **File**, **Save As**. Select **Lib Feat Part** (***.sldlfp**) in **Save as type**.

## Library Feature Subfolders

When you open a part saved as a library feature, the SolidWorks application adds two subfolders in the FeatureManager design tree under the library feature.

### References

Lists the references that need to be specified when inserting a library feature.

To rename any reference, click-pause-click the reference name (see **Library Features and Links** on page 11-9).

### Dimensions

Lists the dimensions that belong to features marked as library features. The **Dimensions** folder includes two subfolders: **Locating Dimensions** and **Internal Dimensions**.

You can specify a dimension as a **Locating Dimension** by dragging the dimension from the **Dimensions** folder to the **Locating Dimensions** subfolder. You can edit the **Locating Dimension** when you insert a library feature.

If you do not want other users to change a dimension value while inserting a library feature, mark the dimension as an **Internal Dimension**, and drag the dimension to the **Internal Dimensions** folder.

To edit a dimension name, click-pause-click the dimension name (for example **D1** in **D1@Extrude2**).

# PropertyManager

When you insert a library feature, the PropertyManager includes the following groups:

## Placement Plane

Prompts you to select a face or a plane.

If you drag a library feature to a reference plane, the drop point defaults to the center of the selected reference plane.

## Configuration

Prompts you to select a configuration for the library feature. **Default** is selected if no configurations exist.

**Link to library part**. This is cleared by default.

If selected, the library feature you insert remains linked to the parent library feature part, and if updated, the library feature in the parent part changes. You cannot edit the inserted library feature (see **Library Features and Links** on page 11-9).

## Location and References

Depending on the library feature, either **Location** or **References** appears after you select a configuration (or automatically if only a default configuration exists).

**Location**. If you do not have references, click **Edit**. This enables you to position the library feature by editing the sketch.

**References**. Lists the references you need to specify to insert the library feature. The **Locating Dimensions** (see **Dimensions** on page 11-7) allow you to specify values to position the library feature.

---

Library features with face references such as fillets do not need reference dimensions.

---

The popup window in the graphics area indicates the references you must select on the model to position the library feature.

## Size Dimensions

Select **Override dimension values** to change the size of the library feature. This option is only available if you have not linked the library feature to the parent part.

---

Since **Link to Library part** (see **Library Features and Links** on page 11-9) is cleared, you are informed when a library feature is linked to the parent and **Override dimension values** is checked.

---

## Library Features and Links

Selecting or clearing **Link to library part** determines the behavior of the library feature and its relation to the parent part.

| **Capability** | **Available with Link to library part *selected*** | **Available with Link to library part *cleared*** |
|---|---|---|
| Select a face or a plane for the **Placement Plane**. | ✔ | ✔ |
| Select **Edit Sketch** from the FeatureManager design tree after the library feature is inserted. | ✘ | ✔ |
| Select **Edit Definition** from the FeatureManager design tree after the library feature is inserted. | ✘ | ✔ |
| Change the values of **Locating Dimensions** in the **References** group. | ✔ | ✔ |
| Select **Override dimension values** in the **Size Dimensions** group to edit the dimensions of the library feature sketch after the library feature is inserted. | ✘ | ✔ |
| Change configuration of the feature. | ✔ | ✔ |
| Select **Edit Sketch** in the PropertyManager when the library feature includes no references. | ✘ | ✔ |

Library features specific to machine design are now included under **Library Features** in the SolidWorks application. These library features are generated using design tables to facilitate design modifications. Examples, with configuration names, are listed on the next page.

| Library feature | Configuration naming convention |
|---|---|
| **Slots** | |
| • **Straight slot** | Width x length. For example, 0.250 x 1.025. |
| • **Curved slot** | Slot width x outer radius x angle (the slot angle, is currently a constant at 90 degrees). For example, 0.125 x 1.25 x 90. |
| **Keyways** | |
| • **Square** | Width x height (for the shaft) and (for the hub).<br>For example, 0.8125 x 0.8175. |
| • **Rectangular** | Width x height (for the shaft) and (for the hub).<br>For example, 2.5 x 0.625. |
| **O-ring grooves** | **bs 4518** - Pneumatic seal 0036-24. |
| **Retaining ring grooves** | Configuration names for **Internal** and **External retaining ring grooves** are available. |
| • **Internal** | **basic internal** - ANSI b27-7. For example, diameter = 10. |
| • **External** | **basic external** - ANSI b27-7. For example, diameter = 24. |
| **Hole patterns** | |
| **Linear hole pattern** | Configuration names use the formula **diameter a** x **b**, where **diameter** size is the hole diameter, and the values **a** and **b** represent the rows and columns, respectively.<br>For example: 0.188 2 x 2.<br>Linear hole patterns use standard hole sizes 1/8 - 3 (inches) in increments of 1/16. Spacing is based on twice the diameter, with patterns ranging from 2x2 to 8x8. |
| **Radial hole pattern** | Configuration names use the formula **D** x **L** x **A**.<br>For example: 0.188 x 0.500 x 30.<br>Radial hole patterns use standard diameter hole sizes, with diameter represented by the value **D**, range from 1/8 - 3 (inches) in increments of 1/16. The diameter of the pattern, represented by the value **L**, is fixed at 4 times the diameter. The angle, represented by the value **A**, varies from 0 to 90 degrees in increments of 15 degrees. |

12

# Other Functionality

This chapter describes enhancements in the following areas:

- Application Programming Interface and macros
- Import/Export
- Installation

# Application Programming Interfaces and Macros

## Application Programming Interfaces

Click **Help**, **SolidWorks API and Add-Ins Help Topics** to access the SolidWorks, eDrawings, FeatureWorks®, PDMWorks, SolidWorks Toolbox Browser, and SolidWorks Utilities Application Programming Interfaces (API) Help systems.

### SolidWorks API

The major enhancements made to the SolidWorks API in SolidWorks 2005 are:

- **Quick Tips and Bubble ToolTips**. You can create Quick Tips and Bubble ToolTips for any add-in.
- **Manipulator and TriadManipulator**. You can create a generic manipulator and use it to create a triad manipulator, which is similar to the triad in the SolidWorks user interface.
- **Custom properties column in a revision table**. You can get the custom properties for a custom properties column in a revision table.
- **Layers**. You can set the color, name, style, and line width of layers.
- **Display dimensions of drawing sheets and views**. You can get the display dimensions from the current drawing sheet or the current drawing view.
- **Pre-notification of new drawing view**. An event can pre-notify your application when a new drawing view is about to be created.
- **Arrays of annotations**. You can return arrays of annotations.
- **Dynamically highlighted object selection**. You can select dynamically highlighted objects.
- **FeatureData**. Seventeen new FeatureData APIs are included in this release.
- **Texture**. You can access textures on bodies, components, faces, features, and documents.
- **User-specified type libraries**. You can add and remove references to user-specified type libraries.
- **Add-ins and swAppFileOpenNotify2**. Because it is now possible to have a NULL active document when an add-in is notified by swAppFileOpenNotify2, use SldWorks::IGetOpenDocumentByName2 instead of SldWorks::IActiveDoc2.
- **Examples**. Hundreds of new Visual Basic code examples are included in this release.

### eDrawings API

This release includes an eDrawings™ API. This API is an OLE programming interface to eDrawings and is implemented as a Microsoft ActiveX® control. You can use the eDrawings API to customize the eDrawings Viewer, create interactive web pages, and translate files.

### PDMWorks API

The PDMWorks API is now enabled for both PDMWorks client (add-in and standalone) and PDMWorks advanced server installations. Other enhancements in this release of the PDMWorks API include:

- **Document check-ins**. You can check in documents.
- **Documents and projects**. You can move a document to a different project. You can also create sub-projects.
- **Document ownership**. You can change ownership of a document.
- **Lifecyle statuses**. You can get the vault's lifecycle statuses and the status of a document.
- **Revision labels**. You can get the next suggested revisions for a document.
- **SolidWorks version**. You can get the SolidWorks version of a document.

### SolidWorks Utilities API

Support for the Power Select tool is included in the SolidWorks Utilities 2005 API.

## Macros

You can automatically edit macros after recording them. See **Macro Editing** on page 1-17 for details.

# Import/Export

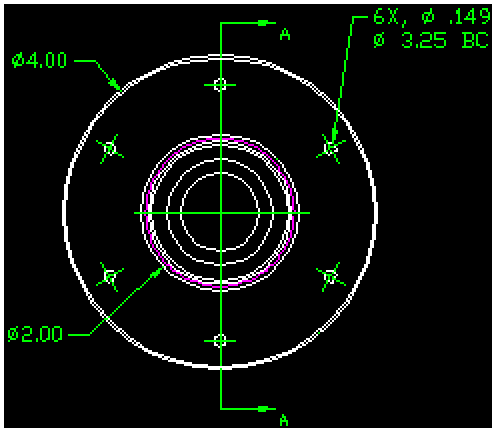
## SolidWorks DWGEditor

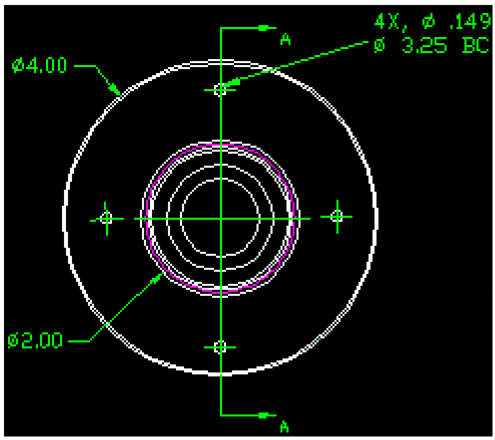
The new SolidWorks DWGEditor™ software allows you to edit DWG and DXF files and save them in their native format. Because the documents are maintained in the DXF/DWG formats, they can be opened again in AutoCAD.

This upgraded environment for supporting DWG and DXF legacy files includes interaction with tools such as PDMWorks and eDrawings in addition to functionality such as cutting and pasting geometry from the editor into SolidWorks.

You can access the SolidWorks DWGEditor in several ways.

- Click **Start**, **SolidWorks 2005**, **SolidWorks DWGEditor**.
- In the SolidWorks **Open** dialog box, select **DXF(*.dxf)** or **DWG (*.dwg)** for **Files of type**. In the DXF/DWG Import Wizard, select **Edit/view in the SolidWorks DWGEditor**, then click **Finish**.
- During the installation of SolidWorks, choose to associate DXF/DWG files with the SolidWorks DWGEditor. The editor appears when you double-click **.dxf** and **.dwg** files in a file explorer or **Design Library** or when you check files out of a PDMWorks Vault.




***To edit a file in SolidWorks DWGEditor:***

1. Click **Open** on the Standard toolbar, or click **File**, **Open**.
2. In the dialog box, select **DWG(*.dwg)** for **Files of type**, browse to **\import\7550-021.dwg**, and click **Open**.
3. In the **DXF/DWG Import** dialog box, select **Edit / view in SolidWorks DWGEditor**, then click **Finish**.
4. In the DWGEditor, edit the drawing; for example, change the pattern of the holes in the view on the left from 6 holes to 4 holes.
5. Click **File**, **Save As** and save the drawing as **7550-021A.dwg**.

## AutoCAD DXF/DWG Files

Import of solids and surfaces in AutoCAD DXF and DWG files has been enhanced. Import and export have been extended to AutoCAD R2004 in DWGDirect format.

Custom line styles are supported for export. On import, custom line styles in AutoCAD are not recognized.

## Mechanical Desktop (MDT) Translator

You can now create associative SolidWorks drawings when importing Mechanical Desktop® (MDT) files. An option, **Create linked drawing document**, has been added to the DXF/DWG Import Wizard dialog box. Mechanical Desktop must be installed on your system.

## eDrawings

You can add passwords when publishing eDrawings files from SolidWorks if you have eDrawings Professional.

You can also export animations in models to eDrawings files.

For more about eDrawings, see **eDrawings** on page 13-2.

## Assemblies

Components are no longer suppressed when importing large assemblies.

## ProENGINEER Translator

The following new ProENGINEER® features are supported for import into SolidWorks:

- Imported features (non-native to ProENGINEER)
- Copy geometry feature
- Thin revolve features

Import of the following features has been enhanced:

- Lofts with multiple sketches
- Reference patterns
- Ambiguities between circular and linear patterns
- Holes (more types recognized)
- Protrusions (values for depth improved)
- Sweep features with open loops
- Cuts with open sketches
- Cut lofts

## Unigraphics Files

The SolidWorks software supports import of Unigraphics® NX files.

Assembly transforms and configurations have been enhanced.

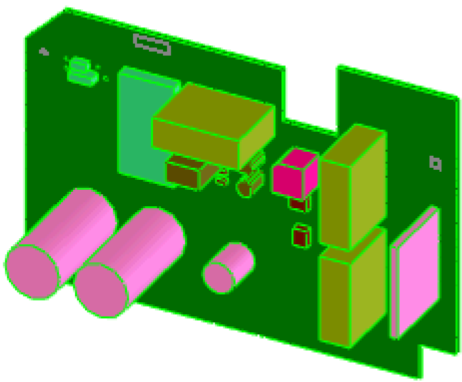
An option, **Import tool bodies**, has been added to the **Import Options** for Unigraphics files. Tool bodies are used to construct the final bodies. Clear the option to import only the final bodies.

## IDF Files

You can now import Intermediate Data Format (IDF) circuit board files (***.emn**, ***.brd**, ***.bdf**, ***.idb**) and create solid models of the circuit board and its components. The model is a single part with each component an extruded feature.

- Each IDF file has two parts: the board file and the library file; for example ***.emn** and ***.emp**. Other possible combinations include ***.brd**/***.lib**, ***.brd**/***.pro**, ***.bdf**/***.ldf**, and ***.idb**/***.idl**. The software looks for a library file with the same name and extension corresponding to the board file. If it does not find the library, it prompts you to browse for the file.
- The name of each extruded feature in the FeatureManager design tree is based on the component name.
- Components of the same type appear in the same color.
- If the board thickness is not defined in the IDF file, you are prompted to specify a value and units.

***To import an IDF file:***



1 In SolidWorks, click **Open** on the Standard toolbar, or click **File**, **Open**.

2 In the dialog box, for **Files of type**, select **IDF(*.emn,*.brd,*.bdf,*.idb)**.

3 Click **Options**, clear the IDF options described below, then click **OK**.

- **Add board drilled holes**. If holes have been drilled in the circuit board, the holes appear in the SolidWorks part.
- **Reverse underside components**. In the rare cases where import data causes the components to be rotated incorrectly, selecting this option compensates for the misplacement and allows proper location of the component features.

4 Browse to **import****circuit_board.emn**, then click **Open**.

A status bar displays the progress and the number of components.

# Installation

## Administrative Images

Use the **SolidWorks Administrative Director** to create an administrative image for use in multi-seat installations of the SolidWorks software. In the **SolidWorks Master Setup** dialog box, click **Administrative Tools** to create an administrative image. Previously, you had to use the command line to create an administrative image.

The administrative image wizard creates an initialization **(.ini)** file that contains information about how to install the SolidWorks software to clients, how to migrate registry settings to clients, various options, and so on. When you complete the wizard, an **.html** file is generated that you can send to each client. A user clicks the link in the **.html** file and the SolidWorks software is installed automatically with all of the options you specified.

## SolidNetWork License

### Borrowing Licenses

If you use SolidNetWork License (SNL) to access a floating license of the SolidWorks software, you can now use that license from a remote location. Previously, you could not access an SNL license if your computer was not connected to the license server.

For instance, you may use a laptop computer that is connected to the license server while you are at work, and then you take the computer home. You can now “borrow” a license so that you can use the SolidWorks software from both home and work, regardless of whether you are connected to the license server. The license is lent for a specified period of time, up to 10 days, (or as defined by your system administrator), and the license is removed from the available pool of licenses.

There are additional options that a system administrator can use to customize the borrowing of SolidWorks licenses. For example, a system administrator can:

- Define users or groups that are allowed to borrow licenses
- Set the number of licenses that cannot be borrowed

See the SolidNetWork License Installation and Administration Guide for more information about borrowing licenses.

### eDrawings Professional

SNL now supports eDrawings Professional.

13

# SolidWorks Office Add-Ins

This chapter describes enhancements to the following SolidWorks Office add-ins:

- eDrawings
- eDrawings Professional
- FeatureWorks
- PDMWorks
- PhotoWorks
- SolidWorks Animator
- SolidWorks Toolbox
- SolidWorks Utilities

# eDrawings

## General

### eDrawings Logo

An eDrawings logo is displayed in the corner of the HTML file if the HTML file does not show the eDrawings application frame and menus when you display it on a web page.

### Email Messages

The instructions in the email sent with eDrawings files include more detailed instructions to the recipient about how to use eDrawings than were previously sent.

### Unshaded Mode Line Color

When you switch the **Shaded** tool off, the model lines are shown in black to make the model more visible. This functionality is supported by all versions of eDrawings files.

## Application Programming Interface

The eDrawings Application Programming Interface (API) is an OLE programming interface to eDrawings and is implemented as a Microsoft ActiveX control. You can use the eDrawings API to customize the eDrawings Viewer, create interactive web pages, and translate files. The API contains functions that you can call from Visual Basic (VB), Visual Basic for Applications (VBA), VB.NET, C++, and C#. These functions provide direct access to the eDrawings environment.

To obtain the eDrawings API Software Development Kit (SDK), which contains sample code and the eDrawings API Help, contact API Support at apisupport@solidworks.com.

## Compression

eDrawings files are now saved using a new compression algorithm that automatically maximizes file compression and maintains high visual quality. The following options have been removed:

- **Compress file** (In the eDrawings Viewer, **Save As** dialog box)
- **Compress this eDrawing** (In the SolidWorks application, **Export Options** dialog box)

## Download Options

When you download the eDrawings application from the solidworks.com web site, you have the option to download the DXF/DWG translator. The DXF/DWG translator lets you open DXF/DWG files in eDrawings. If you do not intend to view DXF/DWG files in eDrawings, do not download the translator, which reduces download time.

Previously the translator was downloaded automatically.

## DXF/DWG XREF Paths

When you import DXF or DWG files that reference XREF, shape, or font files, you can specify the paths used to find these referenced files. You can specify multiple search paths and the search order. Previously you could not specify the path.

The eDrawings application still tries to resolve the paths automatically if none are specified. If eDrawings cannot resolve the path, it prompts you to resolve the path.

***To set the XREF path:***

1 Open **eDrawings\HomeOffice-v3.dwg**.

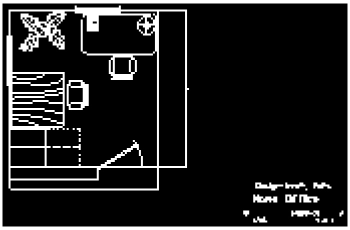
The **Resolve Xref** dialog box appears because eDrawings is unable to find the XREF file named **bsize**.

2 Click **Skip file** to skip resolving the XREF.

The DWG file opens with the XREF unresolved.

3 Select the **Layout1** tab beneath the graphics area.

Note that the sheet format, containing the title block and border, is missing because the XREF is unresolved.

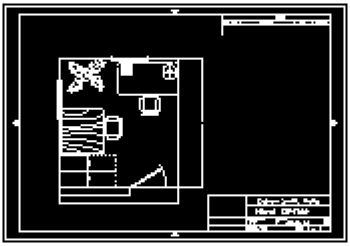


4 Close the DWG file.

5 Click **Tools**, **Options**.

6 On the **Import** tab, under **DXF/DWG Import**, click **Add**.

7 Browse to **eDrawings\New_XREF_Folder\User**, then click **OK** twice to specify the path to the XREF file.



8 Open **HomeOffice-v3.dwg** and select the **Layout1** tab.

The sheet format appears because you set the correct path to the XREF.

## eDrawings Executables

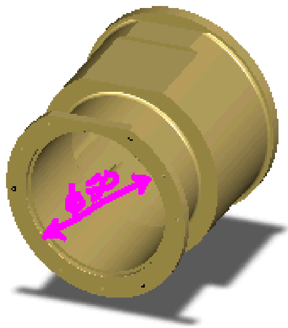
When you close the eDrawings application launched from an eDrawings executable file, a message prompts you to install eDrawings if it is not installed on that computer. The prompt appears every fifth time you exit the eDrawings application launched from the executable file.

## Shadows

The **View**, **Shadows** option has been replaced with the **Tools**, **Options**, **General**, **Display shadows** option. The shadows functionality is unchanged.

## Tablet PCs

The eDrawings application is supported on Tablet PCs. You can use the pen to do electronic markups.



## Toolbar Buttons

Toolbar buttons are now available for these functions:

| Toolbar Button | Menu Item |
|---|---|
| **Save Markup** | **File**, **Save Markup** |
| **Mass Properties** | **Tools**, **Mass Properties** |
| **Help** | **Help**, **eDrawings Help Topics** |

The functionality remains unchanged.

# eDrawings Professional

## Animations

When you publish eDrawings files from SolidWorks files that contain SolidWorks Animator animations (including physical simulations imported into animations), you can view these SolidWorks Animator animations in the eDrawings Viewer. Previously, SolidWorks Animator animations were not saved in eDrawings files.

Animation creation has changed in SolidWorks 2005. See **SolidWorks Animator** on page 13-24.

***To publish animations from the SolidWorks application:***

1 Click **Tools**, **Add-Ins**, select **SolidWorks Animator**, then click **OK**.

  You must have the SolidWorks Animator add-in enabled to publish animations in eDrawings files.

2 Open **eDrawings/speaker.sldasm** from the **File Explorer**.

3 Click **File**, **Save As**.

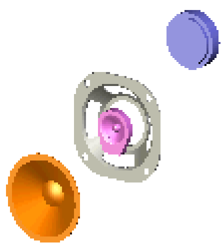
4 Select **eDrawings (*.easm)** in **Save as type**.

5 Click **Options**.

6 Select **Save Animator Animations to eDrawings file** so you can view the animations in the eDrawings Viewer.

7 Click **OK**, then click **Save** to save the file as **speaker.easm** to the same folder.

***To view animations in the eDrawings Viewer:***



1 Open **eDrawings/speaker.easm**.

2 Select the **Animations** tab in the eDrawings Manager.

  **Animation1** is listed as an available animation. The **Time** column shows the animation runs for 5 seconds.

3 Click **Normal** so the animation plays once and stops.

4 Click **Play**.

## Passwords

When you publish an eDrawings file from the SolidWorks application, you can assign a password to protect the eDrawings file.

You must have eDrawings Professional installed to be able to protect eDrawings files with passwords. You can add a password to new or existing eDrawings files. You can open password-protected eDrawings files using the complimentary eDrawings Viewer, provided you have the correct password.

Once protected, the password cannot be removed from the eDrawings file. The password is carried over when you save a protected eDrawings file as another file name.

Passwords do not guarantee security of the eDrawings file. They provide only a moderate level of security.

***To add a password to an eDrawings file:***

1 In the SolidWorks application, open **eDrawings**/**Password.sldprt** from the **File Explorer** .

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

3 Select **eDrawings (*.eprt)** in **Save as type**.

4 Click **Password**.

5 In the dialog box, select **Password Required to Open Document**, then type a password.

6 Type the same password for **Confirm Password**, then click **OK**.

7 Click **Save**.

Now open the eDrawings file to test the password.

1 In the eDrawings Viewer, open **eDrawings**/**Password.eprt**.

2 In the **Password** dialog box, type an incorrect password for **Enter Password**, then click **OK**.

A message prompts you to enter the password again.

3 Click **OK**, enter the correct password, then click **OK** to open the eDrawings file.

## SolidNetWork License

SolidNetWork License (SNL) now supports eDrawings Professional. SNL allows users to install more seats of the eDrawings Professional software than they have licenses for, by floating the use of those seats without exceeding the number of licenses purchased.

For more information about purchasing SNL, contact your reseller.

# FeatureWorks

## Hole Wizard Holes

FeatureWorks can now recognize hole features as Hole Wizard holes using automatic or interactive recognition. You can also use step-by-step recognition to recognize Hole Wizard features in parts that contain imported bodies and Hole Wizard holes.

During the recognition process, you can re-recognize cut extrudes, cut revolves, and simple holes as Hole Wizard holes. FeatureWorks cannot re-recognize legacy holes as Hole Wizard holes.

> FeatureWorks supports recognition of **ANSI Metric** standard **Counterbore** and generic **Hole** type Hole Wizard features. All other types of Hole Wizard holes are recognized as Hole Wizard **Legacy** type holes. Simple holes are recognized as generic **Hole** type Hole Wizard holes.

***To set the FeatureWorks options:***

1 Click **Open** on the Standard toolbar, or click **File**, **Open**.

2 In the dialog box, select **Parasolid (*.x_t,*.x_b,*.xmt_txt,*.xmt_bin)** in **Files of type**, browse to **Fwks_HoleWizard.x_t**, and click **Open**.

3 Click **No** to the feature recognition prompt.

4 Click **FeatureWorks Options** on the FeatureWorks toolbar, or click **FeatureWorks**, **Options**.

5 In the dialog box, under **Advanced**, clear **Recognize holes as wizard holes**.

> When the **Recognize holes as wizard holes** option is cleared, FeatureWorks recognizes all holes as Hole Wizard **Legacy** type holes.

6 Click **OK**.

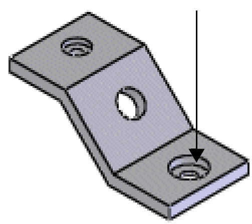
Proceed with feature recognition. All three holes on the imported body are Hole Wizard holes in the original part.

***To recognize Hole Wizard holes using interactive recognition:***

1 Click **Recognize Features** on the FeatureWorks toolbar, or click **FeatureWorks**, **Recognize Features**.

2 In the PropertyManager:

- Under **Recognition Mode**, select **Interactive**.
- Under **Feature Group**, select **Standard features**.



3 Under **Interactive Features**:
   - Select **Cut Revolve** in **Feature type**.
   - Select all three faces of the right-most hole in the graphics area for **Selected entities** .
4 Click **Recognize**.

   The hole disappears from the model.
5 Click **Show** in the PropertyManager.

   The **FeatureWorks - Feature Manager** dialog box appears. In the **Recognized Features Manager**, note that FeatureWorks recognized the hole as a cut revolve feature.
6 Right-click the cut revolve feature in the **Recognized Features Manager** and select **Re-Recognize As**, **Hole**.

   The cut revolve feature is re-recognized as a Hole Wizard **Legacy** type hole because the option to recognize it as a Hole Wizard hole is cleared.

   With the **Recognize holes as wizard holes** enabled, FeatureWorks would re-recognize that hole as a Hole Wizard **Counterbore** type hole.
7 In the **FeatureWorks - Feature Manager**, click **Map Features**.

***To recognize the remaining holes as Hole Wizard holes using automatic recognition:***

1 Click **FeatureWorks Options** on the FeatureWorks toolbar, or click **FeatureWorks**, **Options**.
2 In the dialog box, under **Advanced**, select **Recognize holes as wizard holes**.
3 Click **OK**.
4 Click **Recognize Features** on the FeatureWorks toolbar, or click **FeatureWorks**, **Recognize Features**.
5 Under **Recognition Mode**, select **Automatic**.
6 Under **Automatic Features**, select **Holes** and clear all other options.
7 Click **Recognize**, then in the dialog box, click **Map Features**.

   FeatureWorks recognizes the two remaining holes as **Counterbore** type (**CBORE for M2 Hex Head Machine Screw 1**) and generic **Hole** type (**M6.0(6) Diameter Hole1**) Hole Wizard holes, as shown in the FeatureManager design tree.

## Step-By-Step Recognition

FeatureWorks has enhanced step-by-step recognition so you can recognize more features than was previously possible. If a part contains an imported body and any of the following features, FeatureWorks can now recognize these features:

- Base sweeps (without guide curves)
- Sketched bends
- Hem flanges
- Edge flanges
- Miter flanges
- Face fillets
- Face chamfers
- Revolves without a centerline
- Full round fillets

Except for sheet metal features, FeatureWorks also recognizes all these features if they are in multibody parts.

## Miter Flanges

FeatureWorks can recognize miter flange features using interactive recognition only. FeatureWorks does not recognize miter flanges with offset or those on multiple edges.

***To recognize miter flanges:***

1 Click **Open** on the Standard toolbar, or click **File**, **Open**.

2 In the dialog box, select **Parasolid (*.x_t,*.x_b,*.xmt_txt,*.xmt_bin)** in **Files of type**, browse to **Fwks_MiterFlange.x_t**, and click **Open**.

3 Click **Yes** to the feature recognition prompt.

4 In the PropertyManager:

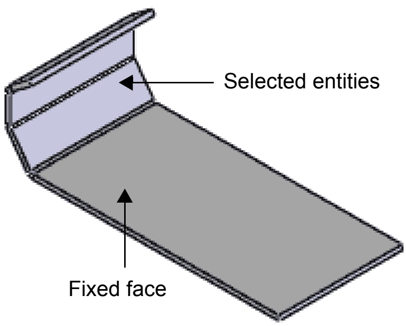
- Under **Recognition Mode**, select **Interactive**.
- Under **Feature Group**, select **Sheet metal features**.

5 Under **Interactive Features**:

a) Select **Miter Flange** in **Feature type**.

b) Select the face shown in the graphics area for **Selected entities**.

c) Click in **Fixed face**, then select the face shown.




6 Click **Recognize**, then click **Map**.

FeatureWorks recognizes the miter flange, as shown in the FeatureManager design tree.

# PDMWorks

## Integration with SolidWorks

### Task Pane

The PDMWorks® local view and vault view are integrated into the SolidWorks Task Pane. You can now view the Vault when no documents are open.

The local view of the Task Pane **File Explorer** includes **My PDMWorks Documents** (displaying documents in the folder specified in **Client Options**). In the **Open in SolidWorks** folder, visual cues include:

- **Solid icon**. A document open in its own window (an assembly, for example).
- **Transparent icon**. A document open only in memory (a component of the assembly, for example).
- **Name in orange**. The document is read-only.
- **Name in bold**. The document has been modified but not saved.

Tools for **Find/Search**, **Reporting**, **Select Labels**, and **Refresh** appear at the top of the vault view. Project folders in the PDMWorks vault view appear with the icon.

### Toolbar and Menus

A PDMWorks toolbar appears in SolidWorks with the **PDMWorks Login**, **PDMWorks Logout**, **Check In (active)**, and **PDMWorks Options** tools.

**Save and PDMWorks Check In** has been added to the SolidWorks **File** menu; it saves the active files and opens the PDMWorks **Check In** dialog box.

**PDMWorks** has been added to the **Tools** menu; it replaces **File**, **PDMWorks Open** and opens the PDMWorks Standalone Client Window.

### SolidWorks Open

You can now open documents in the PDMWorks Vault with the SolidWorks **Open** dialog box. The left panel in the dialog box contains items for **My PDMWorks Documents** and **PDMWorks Vault**.

When you select a document in the Vault that is *not* owned (checked out) by another user, the button to execute the action displays **Check Out**. If another user has checked out the selected document, the button displays **Open**, which opens the document without checking it out.

To open (rather than check out) a document that is not owned by another user, clear the **Check Out** check box so the button changes to **Open**.

The **Check Out Advanced** or **Open Advanced** button opens a dialog box so you can specify reference handling, revision, working directory, note, and so on.

### Compare Documents

The **DrawCompare** utility can access drawings in a PDMWorks Vault. When you browse to a drawing, the SolidWorks **Open** dialog box appears with **My PDMWorks Documents** and **PDMWorks Vault** in the left panel. See **Comparing Drawings** on page 9-2. You can access models in a similar way with the **Compare Documents**, **Compare Features**, and **Compare Geometry** tools in SolidWorks Utilities. This allows you to browse the vault to compare different revisions of documents.

### Revision Tables

PDMWorks **Revision Scheme** is integrated with the SolidWorks Revision Table.

- A new tab in the VaultAdmin, **Revision Table**, enables handling revision tables from PDMWorks and specifies rules for revision, number of rows, and revision order.
- When revision tables are enabled in PDMWorks, the SolidWorks **Alpha/Numerical Control** options are not available for **Revision Table** in **Tools**, **Options**, **Document Properties**, **Tables**.
- If revision tables are enabled in PDMWorks, and the drawing includes a Revision Table, then the table is updated automatically.
- You can sort tables in ascending or descending order.
- When you check in a drawing, the dialog box displays the revision in the table, and the table is updated with the current revision information. The first time the document is checked into the Vault, the revision is validated at the initial level.

---

Revision schemes are case-sensitive, and only uppercase letters are allowed.

---

### Parts Not Revision-Managed

The PDMWorks Vault displays items that are not revision-managed, such as external Toolbox parts and standard parts, as references. You can specify folders for libraries of standard parts, in addition to SolidWorks Toolbox parts, on the VaultAdmin **Toolbox** tab. When you disable check-in of non-revision-managed documents, you can select an option to view referenced files in a system-controlled separate project.

A project folder is created in the vault view for each library (**Toolbox References**, for example). The name of the folder appears in blue with the number of parts in the folder displayed. Users cannot create, rename, delete, or assign ownership to the blue folders, but they can generate reports, search the project, and so on.

Parts that are not revision-managed also appear in blue in both the separate library project and in the projects where they are referenced. A shortcut menu with limited choices is available. You can right-click a document and select **Document Information** to determine where the document is used, for example.

### SolidWorks Task Scheduler (Update and Print)

**Update PDMWorks Files** and **Print PDMWorks Files** are now available in SolidWorks Task Scheduler. You must have a PDMWorks client license installed on the computer using the Task Scheduler. You log in separately for each type of operation.

**Update PDMWorks Files** converts PDMWorks Vault files that were created at an earlier SolidWorks release to the current release. You can update the complete Vault or specify a project. You can specify whether all revisions or only the latest revision are updated. A log file lists any documents that do not convert. The vault is locked automatically during the upgrade process.

**Print PDMWorks Files** schedules PDMWorks batch printing. You can select the complete Vault or just one project, filter for file type (***.drw,*.slddrw**, for example), and schedule a single job or recurring jobs (daily, weekly, or monthly). The documents are copied to a temporary folder on your local drive that you can specify.

### PDMWorks with SolidWorks DWGEditor

You can now launch a PDMWorks Standalone client from the SolidWorks DWGEditor. You can also open DXF and DWG files from a PDMWorks Vault into the DWGEditor. For more information about the editor, see **SolidWorks DWGEditor** on page 12-4.

## Check-In Enhancements

The PDMWorks **Check In** dialog box has been redesigned. Documents and references are displayed in a grid where you can edit fields (including custom properties) individually, in a group, or a whole column (click the header and enter text in the box that appears). You can apply options such as **Retain ownership** and **Delete local copy** to individual documents. Notes can be multi-line.

A VaultAdmin **Lifecycle** option allows clients to specify lifecycle status during check-in.

When you check in assembly and part documents, you can also check in their related drawing documents. A client option specifies whether to include drawings in check-in, check-out, and open operations. If the drawings are not in the same folders as their models, you can specify the folders for **Referenced Documents** in SolidWorks **File Locations** options.

You can now check in lightweight components, suppressed components, base parts, and externally referenced components (except externally referenced assemblies. Parts can be checked in without their assembly document. You can more easily specify the revision of references during check-in.

You are prompted if documents need to be saved before check-in.

## Search Enhancements

The dialog box for searching the Vault has been enhanced with a new **Advanced Search** dialog box. To access the previous **Search** dialog box, click **Simplified**.

The **Advanced Search** dialog box includes use of logical operators to create a list of search criteria, and you can now search for configuration-specific properties and search in hidden projects.

---

The numeric operators are: **=**, **not =**, **>**, **<**, **>=**, and **<=**. The text string operators are: **is (exactly)**, **is not**, **contains**, **does not contain**, **is empty**, and **is not empty**. A limitation is that although the search can compare dates or revisions for equalities (**=** and **not =**), it cannot compare them for inequalities (**>** or **<**).

---

When you generate a report of the search results, you can right-click a document in the report and select operations such as **Open Document**, **Document Information**, and so on.

***To search the vault view:***

1 Click **Find/Search** in the Task Pane, or right-click in the vault view and select **Find/Search**.

2 In the dialog box, define search criteria (as described in the next procedure) and select **Options** and **Direction**.

3 Click **Find** to find the next document in the Vault that meets the criteria, or click **Generate Report** to view a report of all documents that meet the criteria.

When you select a document in the report, the document highlights in the vault view.

***To specify search criteria:***

1 Create a search criterion in the **Advanced Search** dialog box:

a) Select a **Property** (**All Properties**, **Revision**, a custom property, and so on) to be searched.

b) Select a **Condition** (a text string condition such as **is (exactly)**, **is not**, and so on, or a numeric condition such as **=**, **not =**, and so on).

c) Type a **Value** of the property to be met.

d) Click **Add** to add the criterion to the list.

2 For subsequent criteria, select **And** or **Or** and repeat step 1.

3 To save the list of criteria, click **Save Criteria**. In the dialog box, browse to a folder, type a file name (default file name extension **.sqy**), and click **Save**.

---

To load a list of criteria saved previously, click **Load Criteria**, browse to a file, and click **Open**.

---

## Client Options

Under **Defaults**, **Check in / Check out / Open**, you can select **Include drawings**. All related drawings are selected when you execute one of these operations.

Under **Defaults**, **Check out / Open**, you can choose **Select all references**. Then when you check out or open an assembly with the SolidWorks **Open** command, for example, the reference files are checked out or opened without clicking the **Check Out Advanced** or **Open Advanced** button to specify references. See **SolidWorks Open** on page 13-10.

Under **Editing Files**, you can select **Edit .dwg/.dxf files using default program instead of DWGEditor**. For more information on the DWGEditor, see **SolidWorks DWGEditor** on page 12-4.

Under **My PDMWorks Documents**, you can select **Use most recently used SolidWorks or PDMWorks folder** or browse to specify a folder. The folder is displayed in the local view in the Task Pane **File Explorer**.

The folder specified in **Temporary viewing files** is displayed in the local view.

## API Access to Vaults

Client licenses now include API access to a Vault even if the Vault does not include the Advanced Server. In PDMWorks 2004, the Advanced Server was required for API access. The files necessary for the API operations, including operations involving SolidWorks Task Scheduler (see **SolidWorks Task Scheduler (Update and Print)** on page 13-12), are installed automatically with the client.

PDMWorks Triggers and Web Portal clients still require the PDMWorks Advanced Server. Check-in APIs require client licenses.

## Administration Enhancements

Vault passwords are now encrypted.

Shortcut menus now include standard Windows commands in addition to PDMWorks items.

The Vault locks automatically during file updating operations. See **SolidWorks Task Scheduler (Update and Print)** on page 13-12. In addition, you can specify in the VaultAdmin **Vault Settings** for the Vault to be unlocked for one user, typically the vault administrator.

### Delete Documents

When deleting documents from the Vault, you can specify:

- Scope (children)
- Roll back to a specified revision (You are warned if a revision is referenced.)

### Copy Projects

You can copy projects. On the VaultAdmin **Projects** tab, select a project and click **Copy Project**.

A grid displays the documents in the project so you can specify which documents to copy.

You must create new names so the document names are unique. You can append a prefix or suffix to all the document names in a single operation.

You can also change the columns of data (**Document**, **Number**, **Description**, **Owner**, **Status**) by selecting and editing individual cells, or by clicking a column header to edit all items in the column.

With **Advanced Select / Replace**, you can edit the fields. Specify text to search for, add or replace text, and select or clear documents for copying based on specified criteria.

You can specify the name of the new project folder, its description, and its parent. Only the latest revision of the documents is copied to the new project.

Any information that appears in red is not valid because it duplicates information already in the Vault.

### Bulk Check-In

The **Bulk Check In** dialog box includes a filter for document types (**SolidWorks Files (*.sldprt, *.sldasm, *.slddrw)**, **eDrawings Files (*.eprt, *.easm, *.edrw)**, and so on). For file types not in the list, you can select **Other Types of files As Specified Below** and enter a file type.

You can double-click headers to sort the documents.

Bulk check-in now allows you to specify lifecycle status and create new subproject folders. Lifecycle status is validated.

## Triggers

The following Triggers have been added:

- **Bump Revision**
- **Modify Custom Property**
- **Detailed Modify Project**
- **Open Document**
- **Rename Document**

## Web Portal

Each time you open a new Web Portal, a separate browser window is launched so you can view multiple documents simultaneously.

# PhotoWorks

## General Enhancements

Some enhancements to the PhotoWorks™ add-in include:

❑ General

- All the icons in the toolbar, the dialog boxes, and the RenderManager tab now follow the Windows XP look.
- An **OpenGL** button has been added to the **Preview** dialog box to switch between **OpenGL** and **Ray Trace** previews.
- The default **Screen image gamma correction** value is **1**. In earlier versions, this was **1.5**. Images rendered to screen appear darker in SolidWorks 2005.
- You can view a full size image of a decal from the **Decal Editor** dialog box. Right-click a decal on the **Manager** tab and select **View Image File**.

❑ Indirect Illumination

- The indirect illumination options have moved to the **Illumination** tab of the **Options** dialog box from the **Advanced** tab.
- The number of indirect illumination rays is shown on the **Illumination** tab of the **Options** dialog box. This value is read-only.
- You can set the radius for indirect illumination. The radius is the maximum distance from where a photon hits a surface to where the rendering engine stops calculating indirect illumination.

❑ Lights

- PhotoWorks lighting options are available by clicking **PhotoWorks Properties** in a lighting PropertyManager. See **Lighting** on page 4-9.
- You can turn lights on and off for your PhotoWorks scene via the RenderManager tab. Right-click a light and toggle **On in PhotoWorks**. This does not affect the visibility of the lights in the SolidWorks modeling environment.

- Materials

  There are five new materials. The folder **\miscellaneous\Studio Materials** holds four materials used by **PhotoWorks Studio**. The folder **\miscellaneous\water** holds the new **liquid** material that uses the enhanced dielectric materials. Note the difference between the **water** material and the **liquid** material:

water

liquid

## PhotoWorks Studio

The new PhotoWorks Studio allows you to render a model in an existing scene with lights. You select from one of the studios and the scene and lights are automatically added.

You can change the quality and brightness of the image without customizing settings in the **Options** dialog box or lighting PropertyManagers. Move the sliders in the **PhotoWorks Studio** PropertyManager to change the quality and brightness. Adjusting the brightness slider does not change the properties of the lights in the scene.

***To render a model:***

1. Open **indent.sldprt** from the **File Explorer**.
2. Click **Edit Material** on the Standard toolbar, or click **Edit**, **Appearance**, **Material**.
3. In the PropertyManager, under **Materials**, expand **Iron** and select **Malleable Cast Iron**.
4. Click **OK**.
5. Click **PhotoWorks Studio** on the PhotoWorks toolbar, or click **PhotoWorks**, **PhotoWorks Studio**.
6. In the PropertyManager, under **Scenery**, select **Infinite White Floor** in **List of Available Studios**.

Click the preview of the scene in the PropertyManager to view a larger preview.

7 Under **Scene settings**, move the **Render Quality** slider to **low**, and click **Render**.

The model is rendered with the material that you set in SolidWorks and the scene that you set in the **PhotoWorks Studio** PropertyManager. Lights are automatically added to the scene. With the **Render Quality** slider set to **low**, no shadows are cast.

8 Under **Scene settings**, move the **Render Quality** slider to **medium**, and click **Render**.

9 Click **OK** .

Note one difference is that opaque shadows are in the image. If you continued to increase the quality, transparent shadows and high anti-aliasing quality would be used to render the image.

## Caustics

Caustic effects are the result of indirect illumination. Light is emitted from a light source, goes through one or more specular reflections or transmissions, hits a diffuse object, and reflects to the viewer. You can enable caustic effects independent of indirect illumination effects on the **Illumination** tab of the **Options** dialog box.

For example, consider light patterns on the bottom of a swimming pool. Light is emitted from a light source, the sun. The light goes through one or more specular transmissions as it passes through the water. The light hits a diffuse object, the bottom of the pool. Finally, the viewer sees the caustic effects on the bottom of the pool.

For caustic effects to appear in PhotoWorks, the following must be true:

- Caustic effects must be turned on in the **Options** dialog box.
- One or more specular materials must be set to cast caustic photons.
- One or more diffuse materials must be set to receive caustic photons.
- A light source must emit caustic photons. Directional, point, and spot lights can emit caustic photons.

***To set the caustic options:***

1. Open **\SolidWorks_Office\caustics.sldprt** from the **File Explorer** .
2. Click **Options** on the PhotoWorks toolbar, or click **PhotoWorks**, **Options**.
3. On the **Illumination** tab, under **Caustics**, select **Enable caustic**.
4. Click **Apply**, then **Close**.

***To set the caustic options on the materials:***

1. On the RenderManager tab, expand **Materials**. Right-click **polished brass** and select **Edit**.

   This is the material on the ring in the model.
2. On the **Illumination** tab, select **Cast**. Click **Apply**, then **Close**.

   This specular and reflective material now casts photons.
3. Right-click **injected plastic** and select **Edit**.

   This is the material on the base of the model.
4. On the **Illumination** tab, select **Receive**. Click **Apply**, then **Close**.

   This diffuse material now receives photons.

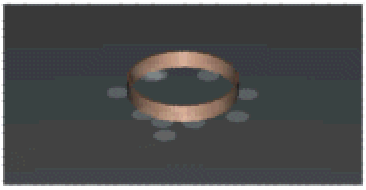
***To set the caustic options on the light source:***

1. On the FeatureManager design tree tab, expand **Lighting**, then right-click **Spot1** and select **Properties**.
2. Under **Basic**, click **PhotoWorks Properties**.
3. Under **Advanced**, set **Energy** to **30** and **C photons** to **10**.
4. Click **OK** .

***To render the model:***

1 Click **Render** on the PhotoWorks toolbar, or click **PhotoWorks**, **Render**.

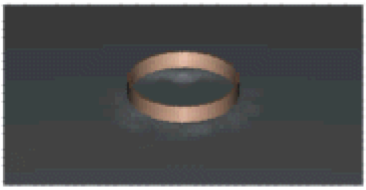
Note the distinct dots on the base from the photons emitted by the spot light that reflect off the ring.



2 Edit the spot light and change **C photons** to **100**.

3 Render the model again.

Note the dots on the base are less clear as more than one photon is reflected on top of another. Also, the dots are fainter because the energy of the spot light is shared among ten times as many photons.



## Global Illumination

Global illumination in PhotoWorks includes all forms of indirect illumination other than that caused by caustic effects. Indirect illumination comes from an object in the scene other than a light source. Global illumination typically affects most objects in the scene.

For global illumination effects to appear in PhotoWorks, the following must be true:

- Global illumination effects must be turned on in the **Options** dialog box.
- One or more specular materials must be set to cast global illumination photons.
- One or more diffuse materials must be set to receive global illumination photons.
- A light source must emit global illumination photons. Directional, point, and spot lights can emit global illumination photons.

***To set the global illumination options:***

1 Open **\SolidWorks_Office\global_illumination.sldprt** from the **File Explorer**.

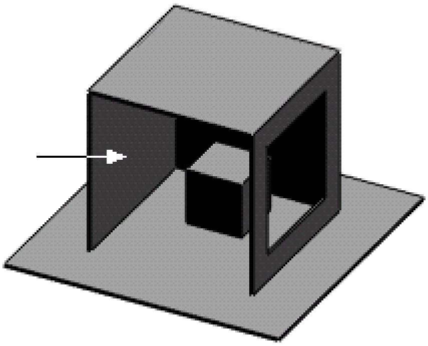
2 Click **Options** on the PhotoWorks toolbar, or click **PhotoWorks**, **Options**.

3 On the **Illumination** tab, under **Global Illumination**, select **Enable global illumination** and **All materials cast and receive illumination by default**.

Global illumination now affects all materials on the model and in the scene.

4 Click **Apply**, then **Close**.

***To add a material:***



1 Select the face shown.

2 Click **Materials** on the PhotoWorks toolbar, or click **PhotoWorks**, **Material**.

3 Expand **plastics**, **miscellaneous** and select **blue polished plastic**.

4 On the **Illumination** tab:

- Select **Conductor** in **Material type**.
- Set **Diffuse** to **0.9**.
- Set **Specular** to **0.1**.

A higher **Diffuse** value increases the effect of indirect rays cast onto the material.

5 Click **Apply**, then **Close**.

***To set the global illumination options for the light:***

1 In the FeatureManager design tree, expand **Lighting**, then right-click **Directional1** and select **Properties**.

2 Under **Basic**, click **PhotoWorks Properties**.

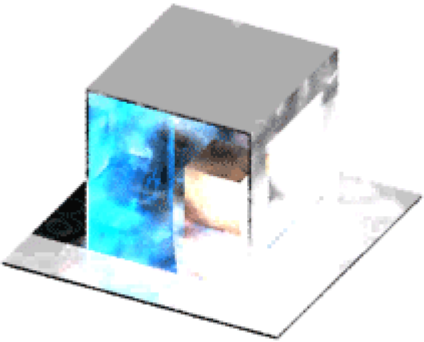
3 Under **Advanced**, set **Energy** to **60** and **G photons** to **10000**.

4 Click **OK**.

***To render the model:***

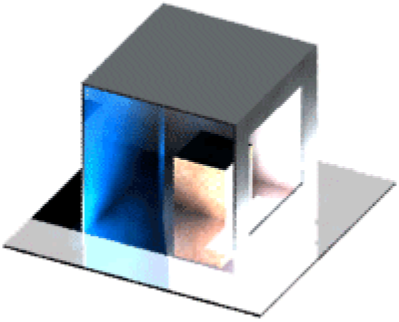
1 Click **Render** on the PhotoWorks toolbar, or click **PhotoWorks**, **Render**.

Note the effect of the global illumination on the blue wall. The dots from the photons are visible and the wall appears splotchy.

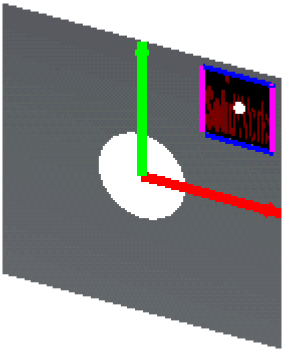


2 Edit the directional light and change **G photons** to **100000**.

3 Render the model again. Note that it takes longer because of the increased photons. Note the blue wall looks much smoother now.



## Decal and Texture Mapping

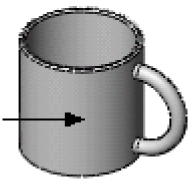


Improvements have been made to how you add decals and 2D texture mapped materials. When the **Preview** dialog box renders using **Open GL**, you see feedback on the size and orientation of the decal or texture. Arrows and lines appear in the preview to indicate the origin, axis of rotation, height, width, and so on. The image shows an example of planar mapping.

Additional enhancements include:

- **Reset aspect ratio**. Click **Use Image Ratio** to reset the decal or texture to its original size.
- **Fit width or fit height to selection**. You can fit the decal or texture to the height of the selected face, the width of the selected face, or both the height and width. In earlier versions, you could only fit to the entire face.

***To add a decal:***



1 Open **\SolidWorks_Office\coffeecup.sldprt** from the **File Explorer**.

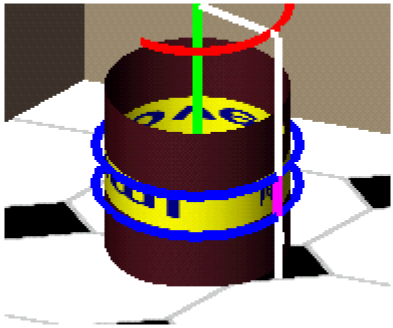
2 Select the face shown, then click **New Decal** on the PhotoWorks toolbar, or click **PhotoWorks**, **Decal**.

3 In the **Preview** dialog box:

- Click **Render**, **Immediate** to immediately see the effects of changes you make in the **Decal Editor** dialog box.
- Click **OpenGL**, or click **Render**, **Opengl**.

4 In the **Decal Editor** dialog box, on the **Manager** tab, select the decal shown. Note the preview.


Decal



Preview

5 On the **Texture** tab:

- Set **About axis** to **115°** to rotate about the axis of the selected cylindrical face. The axis is represented by the green line in the preview and the angle is represented by the red curved arrow.
- Set **Rotation Angle** to **180°** to flip the decal.
- Clear **Fixed aspect ratio**.
- Set **Width** to **30**. The width is represented by the distance between the pink lines in the preview.
- Set **Height** to **45**. The height is represented by the distance between the blue lines in the preview.

6 Click **Apply**, then **Close**.

7 Click **Render** on the PhotoWorks toolbar, or click **PhotoWorks**, **Render**.

## Environment Scenes

Enhancements to environment scenes include:

- **Floors**. Floors have been added to spherical environment scenes.
- **Size/Alignment**. A single value for the radius of a spherical environment replaces the values for length, width, and height.
- **Single image**. A single 2D image is used for a spherical environment. In earlier versions, two copies of the same image were used for the top and bottom of the sphere resulting in a seam where the images met.
- **Background image as environment**. You can use a 2D image as a reflective spherical or cubical environment. The 2D image appears only as a reflection on the model and is not visible in the background. Select a **Scaled image** on the **Back/ Foreground** tab of the **Scene Editor** dialog box. Then, select **Enable spherical environment** or **Enable cubical environment**.

## SolidWorks Animator

SolidWorks Animator has shifted from a path-based interface to a key points-based interface, in which you decide how your assembly should look at various times, and then the SolidWorks Animator application computes the sequences needed to go from one position to the next. Key points-based animation uses three basic user interface elements:

- **Timeline**. The area that displays the times and types of animation events.
- **Timebar**. The entity you position along the timeline to indicate the time in the animation you are viewing or editing.
- **Key points**. The entities that correspond to assembly components, visual properties, and so on.

Using key points-based animation means:

- You position a timebar along the timeline to define where you want a change to end. Changes can include the motion of an assembly component, different viewpoints, and so on.
- You position the assembly components in the graphics area where you want them to be at the time indicated by the position of the timebar.
- SolidWorks Animator positions the appropriate key points, based on the position of the timebar, and solves the required animation, so that the components reach the positions you designated.

You can animate the following:

- Component position
- Value of a distance mate
- Value of an angle mate
- Component properties
- Viewpoint orientation
- Component color
- Component texture

You can continue to reposition components, viewpoint orientation, texture, and so on as part of the same animation, or you can create multiple animations each with its own key points, component positions, viewpoints, and so on. Animations are no longer configuration specific.

## Animation Tabs

To access SolidWorks Animator, use the tabs located at the bottom left of the graphics area. Tabs can include any of the following types:

- **Model**. Selected by default when you open a model. When selected, the graphics area appears normally.
- **Animation1**. Always displays whenever you open a model. The tab may have an empty animation with no sequences, or it may have an animation that includes multiple sequences.
- **Animation -** [*identifier*]. Displays when animations created using previous releases of SolidWorks Animator are included with the assembly. The appended name identifies the configuration, if any, used to create the animation.
- **Animation2**. Displays if you created a new tab (see **Key Points** on page 13-28). Each incremental tab can include one or more animated sequences.

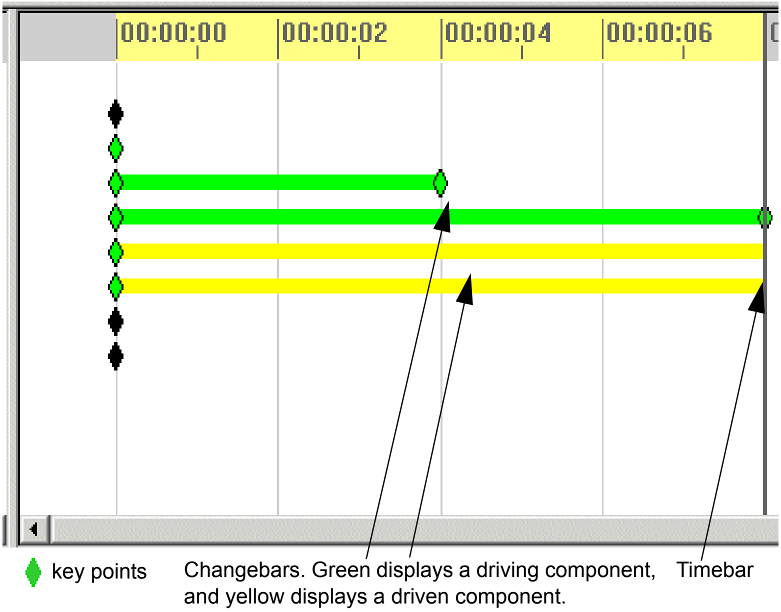
Animation tabs appears for parts or assemblies. You can rename any tab by right-clicking the tab and selecting **Rename**.

## User Interface

When you select an **Animation** tab, the graphics area splits horizontally. The upper area displays the model, and the lower area splits vertically into two sections:

- Lower left section displays the SolidWorks Animator tools and a duplicate of the FeatureManager design tree so that the assembly components align with the timeline. You can select a component using either FeatureManager design tree.

- Lower right section displays the timeline. You use the timeline to position the timebar so that SolidWorks Animator can assign the corresponding key points .




### Driving and Driven Components

When you move a component that drives the motion of related components, the system animates the driven components as well. Along the timeline, the motion of driving components is represented by a green bar, whereas the motion of driven components is represented by a yellow bar (see **Key Points** on page 13-28).

### Timeline

The timeline is the temporal interface for the animation. It works in association with positioning assembly components in the graphics area, or adding visual changes to the components. Visual changes to components include altering the model view and the appearance (color, texture, and display properties) of the model.

The timeline is divided by vertical grid lines corresponding to numerical markers showing the time. The numerical markers start at **00:00:00**, and by default, are spaced depending on the size of the window. For example, there may be one marker every one, two, or five seconds along the timeline.

### Timebar

The solid black vertical line on the timeline, is the timebar It represents the animation's current time. You can move the timebar by dragging it or by clicking anywhere in the timeline except over a key. When you move the timebar, you change the current time in the animation, and the model updates accordingly.

When the pointer is on the timeline, pressing the space bar advances the timebar to the next increment.

### Using the Timeline

To use the timeline to create the animation, first position the timebar where you want the event to end. Then move the components and the viewpoint in the graphics area. Repeat this process for each additional animation sequence.

## SolidWorks Animator Tools

The SolidWorks Animator toolbar appears above the SolidWorks Animator FeatureManager design tree, and displays the following tools:

| Tool | Function |
|---|---|
| | **Play from start** |
| | **Play** |
| | **Stop** |
| | Display animation mode:<br>**Normal**<br>**Loop**<br>**Reciprocate** |
| | **Record**. Records the animation so you can save it to an **.avi** file. |
| | **Animation Wizard**. Starts the wizard (see **Key Points** on page 13-28). |
| | **Place New Key**. Inserts a **Key point** at the position you select. Different colors represent different functions. For example, a green key, represents a component which you can animate (see **Key Points** on page 13-28). |
| | **Delete**. Deletes a **Key point** . |
| | **Zoom In**. Enlarges the size of the timeline to increase space between the timebars. |
| | **Zoom Out**. Reduces the size of the timeline to display a longer time span. |

## Key Points

Each **Key point** on the timeline represents a beginning or end of motion in time. Whenever you position a new **Key point**, it corresponds to motion or changes to visual attributes. Some types of key points include:

### Key Points

Black key points represent all non-movement attributes, including:

- **Viewpoint Orientation**. You can change the view of the model as part of the animation. Use the tools from the View or Standard Views toolbars, and position the model, relative to the timebar.
- **Visual Attributes**. These key points include editable attributes such as **Appearance** and **Texture**. Dragging the component key point also moves any associated component attributes.
- **Unconstrained Move**. Indicates movement that is not restricted.

Green key points represent any selected key.

Green key with black border represent component.

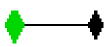
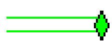
**Bad Solve**. When constraints in the model prevent a component from moving where you had placed it earlier, the green key will turn red. For example, if you animate a component and then later fix the component or add a mate that restricts its movement, the keys you defined earlier will turn red.

**Interim key points**. These are added by Animator to prevent conflicts. Conflicts can occur with components that operate in tandem, such as pistons driven by a crankshaft in an engine. If you move the crankshaft, position the appropriate key point, and then position the key point for a piston further down on the time bar, SolidWorks Animator adds interim key points for the piston to account for their motion.

Blue key points represent distance or angle mate movement.

Changebars are horizontal lines that sometimes connect **Key points**. Changebars serve as visual cues to the component, and to the type of motion. Some types of motion include:

### Changebars between key points

| | |
|---|---|
| Movement of driving component |  |
| Change to visual attribute of component |  |
| Bad solve movement of a driven component |  |
| Interim key point for a driven component |  |
| Movement of distance or angle mate of driving component |  |

## Key Properties

When you move the pointer over any key point, a balloon displays the basic properties for that key point. The example displays the key properties for **Slide2<1>** at two different instances in the animation.


Some of the key properties include:

- Name of the component, or name of the component the key property is related to.
- Position of the key point along the timeline (the example is on the timebar located at **00:00:00**).
- Visual attributes such as color of the component (the example is mustard color).
- View, such as shaded, wireframe, and so on.
- Interpolation mode (see **Shortcuts from the Timeline** on page 13-30).
- Encoding, whether the key point encodes viewpoint, component movement, or both.

When you move the pointer over any key point, the graphic area updates to display the corresponding position of the component.

## SolidWorks Animator Shortcut Menu

The SolidWorks Animator shortcut menu is dynamic. You can right-click entities from the Animator FeatureManager design tree, from the timeline area, or an **Animation** tab to display a shortcut menu.

## Shortcuts from the Animator FeatureManager Design Tree

**Component or other assembly item**. List component display options (see **Component Display Animation** on page 13-31).

**View Orientation**. Allows you to select **Lock** and **Suppress** view orientations.

- **Suppress**. Select to suppress viewpoint orientation keys in the animation, so that the model view does not change during playback, or while editing the animation. Clear to allow the animation to set the model view.
- **Lock**. Select during editing to prevent adding new key points as you spin the model. Clear to include changes in viewpoint orientation as part of the animation.
- **Suppress** is selected.
- **Lock** is selected.
- **Suppress** and **Lock** are both cleared.

When **Suppress** is selected, you cannot clear **Lock**. However, when **Suppress** is cleared, you can select or clear **Lock**.

## Shortcuts from the Timeline

**Key point**. Manipulate existing key points positioned when you placed the timebar during animation editing (see **Key Points** on page 13-28).

**View Orientation** ◆. Right-click and select any of the following:

- **Cut** or **Copy**. Then move your pointer, and select **Paste** where you want to position the new **key point**.
- **Place key point**. Add a new **key point** ◆, or a set of associated key points at the pointer location, then drag the key point left or right to adjust the position.

When you display the shortcut menu and select **Place Key**, the vertical position of your pointer determines which key point or series of associated key points is affected (see **Key Points** on page 13-28).

- **Select All**. Select all the key points. You can select a key point, and then drag to position all the key points where you place your pointer, or choose **Select All** and then choose **Delete** to return the animation to its pre-animation state.

You can select more than one key point from the timeline and choose **Reverse Path** to reverse the animation of entities that correspond to the key points you selected. The position and any visual properties are reversed. No additional reverse animation sequence is created.

**Component** ◆. Right-click, select all functions available with a view orientation key point ◆, plus any of the following:

**Interpolation modes**

This enables you to control the acceleration or deceleration of a component or a visual property during the playback. For example, if a component changes from time **00:00:02** to time **00:00:06**, you can adjust the playback motion from A to B, by selecting any of the following:

- **Linear**. The default setting moves the component from time **00:00:02** to time **00:00:06** at a uniform velocity from position A to position B.
- **Snap**. The component remains in a stationary position A until time **00:00:06** is reached, then it snaps to position B.
- **Ease in**. The component starts moving uniformly, buts accelerates towards position B as it approaches time **00:00:06**.
- **Ease out**. The component starts accelerating at time **00:00:02**, but as it approaches position B, it decelerates towards time **00:00:06**.
- **Ease in/Ease out**. Combines the motion so that the component accelerates to the halfway time between position A and position B, and then decelerates as it approaches time **00:00:06**.

### Animation Wizard

Start the Animation Wizard. The functions in the Animation Wizard have not changed. You can select **Rotate model**, **Explode**, **Collapse** or **Physical Simulation** if a physical simulation was previously enabled.

## Shortcuts from the Animation Tab

**Animation**. Select **Insert**, **Rename**, or **Delete** to insert a new **Animation** tab, rename the existing tab, or delete an **Animation** tab.

# Component Display Animation

Right-click any component in the Animator FeatureManager design tree, or in the graphics area to modify the display for that component.

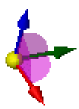
You can animate the display of single or multiple components, and combine different display options within the same or different assembly components.

***To use component display options:***

1 Right-click a component in the Animator FeatureManager design tree.

2 Select from the menu items:

- **Hide** . Hide or show the component.
- **Change Transparency** . Add transparency to the component. If transparent, select **Change Transparency** again to remove transparency.
- **Component Display**. Change how a component displays. Select from SolidWorks display options such as **Wireframe** , **Shaded** , and so on. **Default Display** returns the component display to its original state.



**Move with Triad**. Add a reference triad to the graphics area that you can position anywhere in the graphics area.

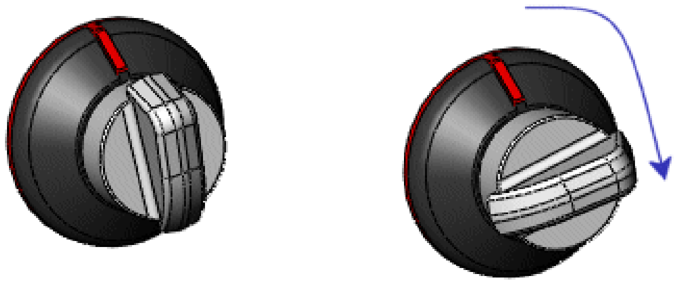
The triad facilitates the movement and the orientation of the assembly based on the X, Y, and Z axes.

- **Appearance**. Select **Color**  or **Texture**  to modify either of these visual attributes on the selected component.
- **Properties** . Display the **Component Properties** dialog box.
- **Zoom to selection** . Display an enlarged view of the component. The size of the view is relative to the size of the assembly, and to the number of components. You can also use any of the tools on the View toolbar.

With the **Animation** tab selected, **Collision Detection** and **Physical Dynamics** are disabled if you try to move a component with the **Move Component**  tool on the Assembly toolbar.

## Limit Mates

You can add a limit mate (**Distance** or **Angle**) to an assembly, and create the animation based on the movement specified by the mate.

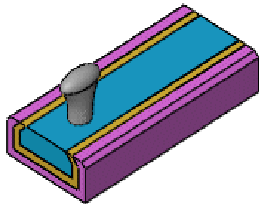


Set an Angle type of limit mate, then animate the rotation. The degree of rotation is defined by the maximum and minimum values you set in the Angle mate.

Once you create the animation, you can click on any section of the timeline to display a preview of the position of the components at that time increment.

***To create an animation:***

1 Open **animator\nested_slides.sldasm** from the **File Explorer** .



2 Select **Animation1** at the bottom of the graphics area.

The graphics area splits horizontally. The top area displays the model. The bottom area splits vertically into two sections:

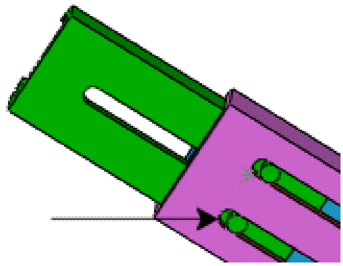
- The Animator toolbar and Animator FeatureManager design tree on the left.
- The timeline with the key points and the timebar on the right.

3 Click **Rotate View**  on the Views toolbar, and spin the model to display the bottom.

4 On the timeline, click at the **5** second mark.

The timebar is inserted at the **5** second mark.

5 Drag the component, **Slide2<1>**, into position, as shown.



The retaining pins on **Slide2<1>** should be almost adjacent with the ends of the slots. Note the following:

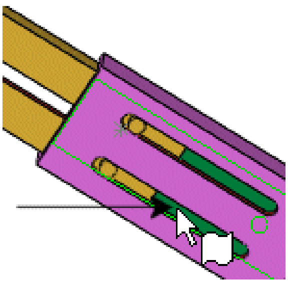
- All key points that belong to **Slide2<1>** move to the **5** second mark.
- The green changebar indicates a driving component.

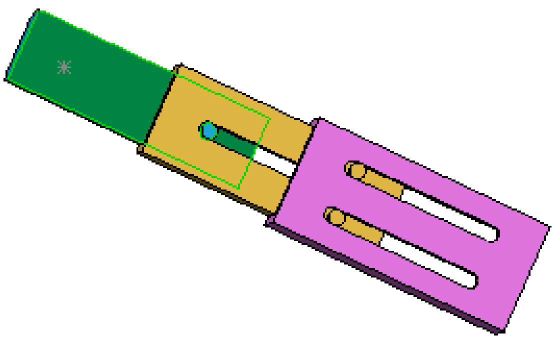
If you were to play the animation, the model would return to its original isometric view, because **View Orientation** is set to **Lock**.

6 On the timeline, click at the **9** second mark.

7 Drag the component **Slide1<1>**, as shown.

The single retaining pin should be almost adjacent with the end of the slot.





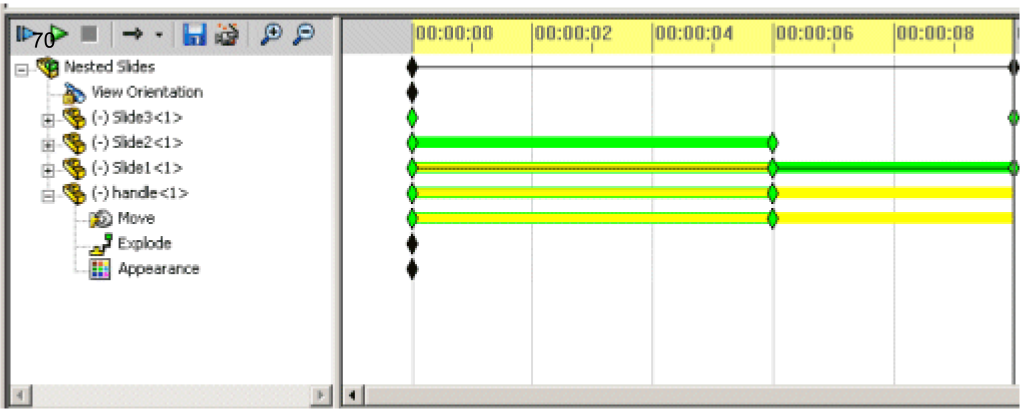
The timebar is inserted at the **9** second mark.

8 In the Animator FeatureManager design tree, right-click **Slide1<1>**, and select **Component Display**, **Wireframe**.

9 Click **Play from start** on the SolidWorks Animator toolbar.

**Slide2<1>** begins the animation, and towards the **5** second mark, **Slide2<1>** starts to move, changing to wireframe as it approaches the **10** second mark.

The timeline appears as follows after you play the animation:




## Save Animation to File

You can save a partial animation by selecting a start and end frame value.

***To save a partial animation:***

1 Create the animation.

2 Select **Record** on the Animator toolbar to save the animation an **.avi** file.

3 In the dialog box:

a) Browse to a folder.

b) Type a name for **File name**.

c) Type a value for **Frames per second** (**7.5** is the default).

d) Select **Enter start and end frame**.

e) Enter values for a starting frame and an ending frame.

4 Click **Save**.

# SolidWorks Toolbox

## Access Control

You can add administrative access to the standards and options of SolidWorks Toolbox. An administrator selects a password and forces settings to be view only. Individual users will be able to view, but not modify, the settings without the administrative password. You can restrict the following settings:

- Standards data. The data in the **Edit Data** tab of the **Configure Data** dialog box
- Toolbox options. The options on the **Browser** tab of the **Configure Browser** dialog box
- Add My Parts Wizard. The **Add My Parts Wizard** dialog box
- Smart Fasteners options. The options on the **Smart Fasteners** tab of the **Configure Browser** dialog box

***To set administrative access:***

1 Click **Options** on the Standard toolbar, or click **Tools**, **Options**.

2 On the **System Options** tab, click **Data Options**.

3 Click **Edit Standards Data**, then click the **Access Control** tab.

There you create a password and change the settings under **User Access Control**. When a check box is cleared by the administrator, a user can view, but not modify, those settings.

## Configure Browser

### Read-Only Documents

The read-only options under **Inserting documents into SolidWorks** of the **Document Properties** portion of the **Configure Browser** dialog box have been removed. All Toolbox documents now follow the read-only attributes of the file.

### Access to Toolbox Library Data

The information on standards, catalogs, documents, and alternate file names is now accessed from the **Options** dialog box within the SolidWorks software. In earlier versions, you access the information from the **Configure Browser** dialog box.

This redesign facilitates the sharing of the information with Hole Wizard. If you disable an item that is shared by Toolbox and Hole Wizard, the item is disabled for both functions. If you turn off the Toolbox Browser add-in, only the catalogs, chapters, and pages appropriate for Hole Wizard appear in the tree on the **Edit Data** tab of the **Configure Data** dialog box.

## Configure Data

There have been many enhancements to the editing of standards, catalogs, and the components in the SolidWorks Toolbox library. Click **Data Options**, **Edit Standards Data** on the **System Options** dialog box to open the **Configure Data** dialog box.

### User-defined Standards

User-defined standards appear with a new icon before the standard's name.

***To create a new standard:***

1 Click **Options** on the Standard toolbar, or click **Tools**, **Options**.

2 On the **System Options** tab, click **Data Options**.

3 Click **Edit Standards Data**.

4 On the **Edit Data** tab, select **Standards** at the top of the tree.

5 Type **whatsnew** for **New Standard Name**.

6 Select **ISO** in **Derived From** and click **Create new standard**.

A progress bar appears, then a new standard is added to the Toolbox library.

7 Keep the **Configure Data** dialog box open for the next procedure.

### Enable/Disable Item

Now, you select a standard, catalog, page, or document in the tree and select **Disabled**. This disables the selected item and all items below it in the hierarchy. The light bulb icon changes to when you disable an item. The behavior of a disabled item is unchanged from earlier versions of SolidWorks Toolbox.

***To disable items in the Toolbox library:***

1 If you have not completed the **User-defined Standards** section, you must do so before proceeding.

You can enable and disable items in default standards or user-defined standards. A user-defined standard is used for this example.

2 On the **Edit Data** tab of the **Configure Data** dialog box, expand **whatsnew**, then **Nuts**, then **Hex Nuts**.

3 Select **Hex Nuts**, then select **Disabled**.

**Hex Nuts** and all of the documents under **Hex Nuts** are now disabled. They will not appear in the Toolbox Browser in the **Task Pane**.

4 Keep the **Configure Data** dialog box open for the next procedure.

## Enable/Disable Values

You can now disable individual sizes, lengths, finish types, and other values for the properties of documents. In earlier versions, you could not disable individual values.

***To disable a value:***

1. If you have not completed the **User-defined Standards** section, you must do so before proceeding.
2. On the **Edit Data** tab of the **Configure Data** dialog box, expand **whatsnew**, then **Nuts**, then **Hex Nuts - Fine Pitch**.
3. Select **Hex Nut Style 1 Fine ISO - 8673**.
4. On the **Finish** tab, select **Disabled** next to **Washer-face**.

   The value, **Washer-face**, is now disabled and will not appear when you add a **Hex Nut Style 1 Fine ISO - 8673** to an assembly.
5. Keep the **Configure Data** dialog box open for the next procedure.

## Shared Properties

The disabling of some properties is shared within a standard or catalog. This occurs for properties that are common to more than one type of fastener. Shared properties are denoted by on the tab. For example, if you disable **Schematic** threads for a countersunk bolt, then **Schematic** threads are disabled for all other bolts and screws within the standard or catalog. Some shared properties include: **Finish**, **Thread Display**, and **Drive Type**.

## Custom Properties

Custom properties that you add to SolidWorks Toolbox documents appear as a tab in the **Configure Data** dialog box. You must select **Each value for this property requires a new configuration name** when you add the custom property. Custom properties are denoted by on the tab.

## Alternate File Name

Select a catalog document and add an alternate file name on the **Properties** tab. The behavior of an alternate file name remains unchanged from earlier versions.

## Access to More Properties

In earlier versions, you could view and edit the size of the components including diameter, length, thread length, and so on in the **Configure Data** dialog box. Now, you can also view and disable the thread display, finish, hub style, keyway, and many other properties in this dialog box. This allows you to view the available properties of many components in one location before you insert the component into an assembly.

## Part Numbers and Descriptions

You can now add the part numbers and descriptions for every configuration of a document. After generating all configurations, you can optionally export the data to Microsoft Excel, enter the part numbers and descriptions, then re-import the data. In earlier versions, you had to enter the part numbers and descriptions one configuration at a time as you added the document to an assembly.

***To enter part numbers and descriptions:***

1 If you have not completed the **User-defined Standards** section, you must do so before proceeding.

2 On the **Edit Data** tab of the **Configure Data** dialog box, expand **whatsnew**, then **Washers**, then **Plain Washers**.

3 Select **Washer - ISO 8738 Clevis Pin**.

4 Click the **All Configurations** tab.

The software generates and displays all the configurations for the washer.

5 Click **Export** and save the spreadsheet as **whatsnew washer.xls**.

6 Open the spreadsheet in Excel and enter part numbers and descriptions in the appropriate column.

The values in the **Part Numbers** column must be unique. Also, do not delete any columns from the spreadsheet.

7 Save and close the spreadsheet.

8 In the **Configure Data** dialog box, click **Import** and open **whatsnew washer.xls**.

The part numbers and descriptions appear in the dialog box and appear when you insert the documents into an assembly.

9 Click **OK**.

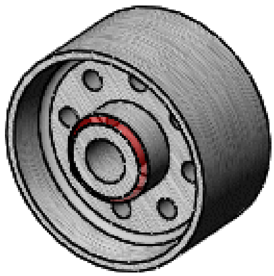
## SolidWorks Utilities

### Compare Features

The **Compare Features** tool now supports appearance properties, including colors, optics, and textures.

***To compare the appearance properties using Compare Features:***

1. Open **SolidWorks_Office****wheel_A.sldprt** and **SolidWorks_Office****wheel_B.sldprt** from the **File Explorer**.



wheel_A.sldprt

wheel_B.sldprt

2. Click **Compare Features** on the Utilities toolbar, or click **Utilities**, **Compare Features**.
3. In the dialog box:
   a) Select **wheel_A.sldprt** in **Reference Part**.
   b) Select **wheel_B.sldprt** in **Modified Part**.
   c) Click **Compare**.
4. In the **Compare Features: Results** dialog box, expand **Modified Features**.
5. Click **Pair 1**.

   The **Color**, **Color values**, and **Transparency** values for both parts are listed because they differ in both parts.
6. Click **Pair 2**.

   The textures for both parts are listed because they each have a different texture.
7. Click **Close**.

## Compare Geometry

The **Compare Geometry** utility now supports assemblies when evaluating volume comparison only.

The definition of a "modified face" has been updated for the **Compare Geometry** tool. A "modified face" now includes overlapping faces if the intersection area of two faces is more than 50% of the area of the smaller face.

***To determine modified faces with the Compare Geometry tool:***

1 Open **SolidWorks_Office****block_A.sldprt** and **SolidWorks_Office****block_B.sldprt** from the **File Explorer** .

   **Block_B.sldprt** is a modified version of **block_A.sldprt**.

2 Click **Compare Geometry** on the Utilities toolbar, or click **Utilities**, **Compare Geometry**.

3 In the dialog box:

   a) Select **block_A.sldprt** in **Reference Document**.

   b) Select **block_B.sldprt** in **Modified Document**.

   c) Click **Compare**.

4 In the **Compare Geometry: Results** dialog box, expand **Face Comparison** and **Modified Faces**.

   **Pair 1** and **Pair 2** are listed as **Modified Faces**. In previous versions of SolidWorks Utilities, these faces would have been **Unique Faces**.

5 Click **Close**.

6 Click **No** when you are prompted to convert the volume comparison results into solid bodies.

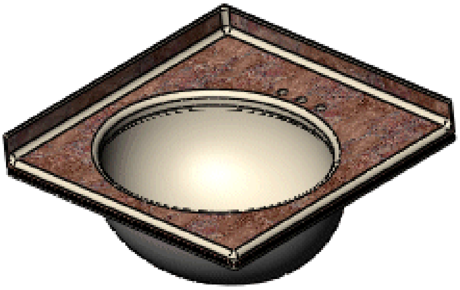
## Feature Paint

The **Feature Paint** tool allows you to paint texture properties to other features. Additionally, when you paint optical properties, the properties are broken down into transparency, ambience, diffusion, specularity, shininess, and emissivity. In previous versions of SolidWorks Utilities, these properties were contained under **Advanced Color Properties**.

> If texture properties are applied to the target feature but not to the source feature, the texture properties are not removed from the target feature.

***To paint texture and optical properties to other features:***

1 Open **SolidWorks_Office****countertop.sldprt** from the **File Explorer**.



The countertop has a granite texture and the inside of the sink is beige.

2 Click **Feature Paint** on the Utilities toolbar, or click **Utilities**, **Feature Paint**.

3 In the dialog box, make sure **Paste appearance properties** is selected.

4 In the FeatureManager design tree:

a) Select **Base-Extrude** for **Copy properties from**.

b) Select **Cut-Extrude1**, **Cut-Extrude7**, **Fillet1**, **Fillet2**, and **Fillet3** for **Paste properties to**.

5 Click **Apply**, but do not close the dialog box.

The granite texture is applied to the selected features.

6 In the FeatureManager design tree:

a) Select **Shell1** for **Copy properties from**.

b) Select **Boss-Loft1** for **Paste properties to**.

7 Click **Apply**.

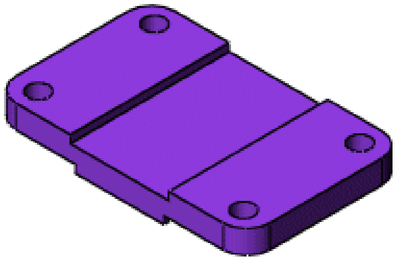
The color and optical properties are applied to the outside of the sink.

8 Click **Close**.

## Find/Modify/Suppress/Simplify

The **Simplify** tool has been added to the **Find/Modify/Suppress** utility. For every part, SolidWorks Utilities determines an internal calculation of "insignificant volume" based on the size of the part. You can enter a simplification factor that increases or decreases the insignificant volume factor. Supported features below the insignificant volume can be suppressed to a derived configuration so you can perform analysis (using COSMOSXpress) on the simplified part.

***To create a simplified part:***



1 Open **detailing****Top_plate.sldprt** from the **File Explorer** .

2 Click **Simplify part** on the Utilities toolbar, or click **Utilities**, **Simplify**.

3 In the dialog box:

   a) Select **Fillets** in **Simplify**.

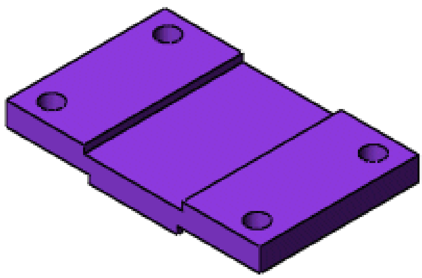
   b) Set **Simplification factor** to **1**.

   c) Click **Find Now**. You must perform the **Find** utility before the **Simplify** utility.

   A fillet feature is found as insignificant geometry based on the simplification factor. The following attributes are listed in the dialog box:

   - **Feature name**. The name of the feature as shown in the FeatureManager design tree.
   - **Feature type**.
   - **Defining parameter**. The parameter that indicates why the feature was found.
   - **Value**. The value of the **Defining parameter**.
   - **Limiting value**. The **Simplification factor** multiplied by the internal calculation for insignificant volume.
   - **State**. This displays the text, **Suppressed**, if the feature is suppressed.

4 In the dialog box:



   a) Under **Feature name**, select **Fillet1**.

   b) Make sure **Create derived configuration** is selected. When the part is simplified, the fillets are suppressed in a derived configuration.

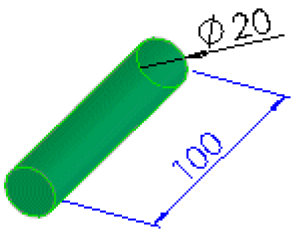
   c) Click **Suppress**.

   A derived configuration is created, called **Simplify_1**.

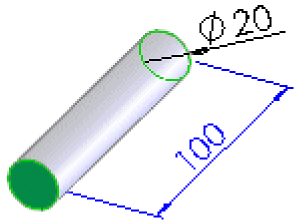
5 Click **Close**.

## Geometry Analysis

The definition of a “small face” has been updated for the **Geometry Analysis** tool. A “small face” is one where all edges of the face are smaller than the specified length *and* the area of the face is less than the square of the specified length of an edge. Previously, a “small face” was one where all of its edges were smaller than a specified length.



SolidWorks Utilities 2004:
three small faces



SolidWorks Utilities 2005:
two small faces

## Power Select

The **Power Select** tool now supports new feature types including sheet metal features and body features (Curve Driven Patterns, Deform features, Structural Members, and so on). Additionally, the **Power Select** tool uses a new interface to select feature types.

#### *To use the Power Select tool:*

1 Open **detailing****tree_gate_simplified.sldprt** from the **File Explorer** .

2 Click **Power Select**  on the Utilities toolbar, or click **Utilities**, **Power Select**.

3 In the dialog box:

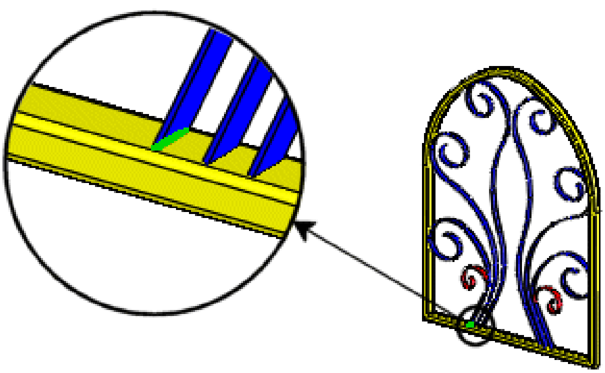
a) Under **Select what**, clear all items except **Features**.

b) Under **Filters and parameters**, clear all items except **Feature type**.

c) Expand **Weldments** in the features list, then select **Trim/Extend**.

d) Click **Search**.

The utility finds one feature, **Trim/Extend5**, and highlights it in the graphics area.



4 Click **Close**.

## Reports

When you create a report in SolidWorks Utilities, you can save the report to the Design Binder. You can create reports with the following utilities: **Compare Features**, **Compare Geometry**, **Compare Documents**, **Geometry Analysis**, and **Thickness Analysis**. See **Design Journal** on page 1-7.

***To save a report to the Design Binder:***

1 Open **SolidWorks_Office****bucket.sldprt** from the **File Explorer** .

2 Click **Geometry Analysis** on the Utilities toolbar, or click **Utilities**, **Geometry Analysis**.

3 In the PropertyManager:

a) Select **Insignificant geometry** and **Small faces**.

b) Set **All Edge Lengths** to **50**.

c) Clear all other check boxes.

d) Click **Calculate**.

4 Click **Save Report**.

5 In the dialog box:

a) Type a name for **Report name**.

b) Select **Add report to Design Binder**.

6 Click **Save**.

7 Click **Cancel** to close the PropertyManager.

The report appears in the FeatureManager design tree in the **Design Binder** folder with a **.utl** extension. It is embedded in the part, not linked. If you update the part, the report does not update automatically.

8 Double-click the report to view it.

9 Keep **bucket.sldprt** open for the next procedure.

## Thickness Analysis

You can now create reports of the data collected with the **Thickness Analysis** tool. The reports contain extra information to supplement the data included in the **Thickness Analysis** PropertyManager. Additionally, you can use the **Report Manager** utility with reports generated from **Thickness Analysis**.

***To save a report with data from the Thickness Analysis tool:***

1 Open **SolidWorks_Office****bucket.sldprt** from the **File Explorer** if you do not have it open from the previous procedure.

2 Click **Thickness Analysis** on the Utilities toolbar, or click **Utilities**, **Thickness Analysis**.

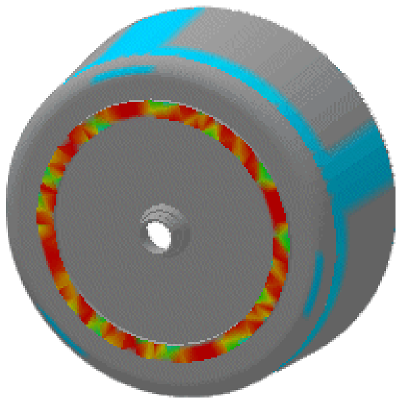
3 In the PropertyManager, under **Analysis Parameters**:

a) Set **Target thickness** to **3**.

b) Select **Show thick regions**.

c) Set **Thick region limit** to **6**.

d) Select **Treat corners as zero thickness**.



4 Click **Calculate**.

5 Click **Save Report**. You can create views in the **Orientation** dialog box before clicking **Save Report** if you want to include custom views in the report.

6 In the dialog box:

a) Type a name for **Report name**. This is the name of the folder where the report is saved. All reports generated by this utility are named **gtReportIndex.htm**.

b) Click **Browse** and navigate to where you want to save the report, then click **OK**.

c) Select **View report on save**.

d) Click **Save**.

The report is generated and opens with the following information:

**Options Used**. Lists the options you selected for the analysis.

**Summary**. Displays an abridged version of the **Analysis Details**.

**Analysis Details**.

- Critical thickness range. Lists four ranges of critical thickness, with the number of faces and surface area of the model that falls into each category. The **% of analyzed area** is the total area analyzed that falls into the associated **Thickness range**.
- Critical features. Lists all features that violate the **Target thickness**.

**Mass Properties**. Lists the **Surface Area**, **Volume**, and **Mass** of the part.

**Model View(s)**. Displays the **Thickness scale** and the **Current** view of the model.

# Index

**Q**



**R**



**S**

**T**