

What's New

SOLIDWORKS 2012



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3S SOLIDWORKS
LET'S GO DESIGN

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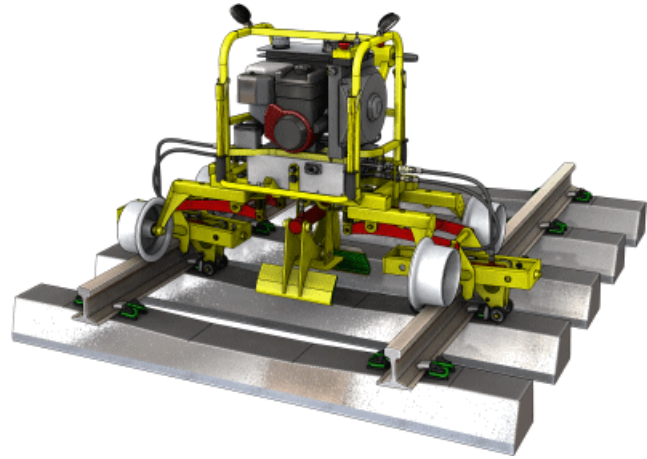
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Highlights of SolidWorks 2012

SolidWorks® 2012 includes many enhancements and improvements, most in direct response to customer requests. This release focuses on the following themes:

- Automated functionality that you use most often
- Improvements in performance and quality that speed up software operation and create a more continuous workflow
- New tools that greatly extend current SolidWorks design capabilities and enable fundamental changes in your product development process
- Extensive support for collaboration and connectivity that allows your design teams to work on joint projects across the globe



Top Enhancements

The top enhancements for SolidWorks 2012 provide improvements to existing products and innovative new functionality. Throughout this guide, look for the  symbol in these areas:

Assemblies

[Large Design Review](#) on page 33

Costing

[SolidWorks Costing Overview](#) on page 45

Drawings and Detailing

[Align Balloons with Magnetic Lines](#) on page 53

[Ordering Balloons](#) on page 67

[Punch Tables](#) on page 71

[Reuse Letters from Deleted Views](#) on page 72

SolidWorks Enterprise PDM

[Customized Enterprise PDM Menus](#) on page 98

[Browse to Option](#) on page 101

[Searching Within File-based Dialog Boxes](#) on page 102

[Version Toggle on Where Used Tab](#) on page 103

Motion Studies

[Motion Optimization](#) on page 153

Parts and Features

[Feature Freeze](#) on page 119


[Exploded Views of Multibody Parts](#) on page 121

Sheet Metal	Forming Tools on page 137 Swept Flanges on page 143
Simulation	Beams in Linear Dynamic Studies on page 145 Beam Results on page 148 Motion Optimization on page 153
SolidWorks Fundamentals	Equations on page 78
Toolbox	Performance and Customization Enhancements on page 173
User Interface	Search Commands on page 16

All features are available in SolidWorks Standard, SolidWorks Professional, and SolidWorks Premium unless otherwise noted.

For More Information

Use the following resources to learn about SolidWorks:

What's New in PDF and HTML	This guide is available in PDF and HTML formats. Click: <ul style="list-style-type: none"> • Help > What's New > PDF • Help > What's New > HTML
Interactive What's New	In SolidWorks, click the  symbol to display the section of this manual that describes an enhancement. The symbol appears next to new menu items and the titles of new and changed PropertyManagers. To enable Interactive What's New, click Help > What's New > Interactive .
What's New Examples	What's New Examples are updated at every major release to provide examples of how to use most top enhancements in the release. To open What's New Examples click Help > What's New > What's New Examples .
Online Help	Contains complete coverage of our products, including details about the user interface, samples, and examples.
Release Notes	Provides information about late changes to our products.

User Interface

Enhancements to the SolidWorks user interface include a completely redesigned Equations dialog box, a powerful Search Commands feature, quick ways to span two displays and fit document windows into display areas, and several other changes that simplify or speed up your workflow.

For information on Equations, see [Equations](#) on page 78.

This chapter includes the following topics:

- [Mouse Gestures and Shortcuts for Macros and Commands](#)
- [Recent Documents and Open Documents](#)
- [Search Commands](#)
- [Select All](#)
- [Spanning and Fitting to Displays](#)
- [Unit System in the Status Bar](#)

Mouse Gestures and Shortcuts for Macros and Commands

You can assign macros and the **OK**, **Cancel**, and **Escape** commands to mouse gestures. You can also assign the **Cancel** and **Escape** commands to shortcut keys.




This release of the SolidWorks software does not require a `\Macros` folder in the SolidWorks installation directory to display and manage macros that have keyboard shortcuts. Customized macros (those assigned to keyboard shortcuts or mouse gestures as well as those added to toolbars) are now managed internally. If you used a `\Macros` folder for macros that have shortcut keys in SolidWorks 2011 and earlier, you need to copy the folder to your SolidWorks 2012 installation directory. You can store macros that you customize in SolidWorks 2012 anywhere on your system.

To assign a macro to a mouse gesture:

1. With a document open, click **Tools** > **Customize**, and select the Mouse Gestures tab.
2. In **Category**, select **Macros**.

The list includes macros you have previously assigned to a shortcut key or toolbar button. It also includes a **New Macro Button** row.



To assign a mouse gesture to a macro that is not associated with a shortcut key or toolbar button, click  in the **New Macro Button** row, then navigate to and set up the macro in the Customize Macro Button dialog box. See *SolidWorks Help: User Interface Customizations for Macros*.

3. Assign a mouse gesture to the macro. Mouse gesture assignment works the same way as it does for other commands. See *SolidWorks Help: Reassigning Mouse Gestures*.

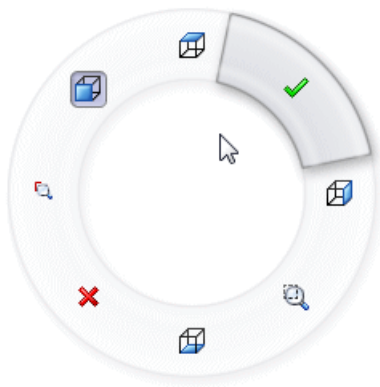
To assign the **OK**, **Cancel**, or **Escape** commands to a mouse gesture:

1. With a document open, click **Tools** > **Customize**, and select the Mouse Gestures tab.
2. In **Category**, select **Others**.
3. Scroll to the command to which you want to assign a mouse gesture, or use the **Search for** box to find the command.

For example, type **OK** in **Search for** to find the **OK** command.

4. Assign a mouse gesture to the command. Mouse gesture assignment works the same way as it does for other commands. See *SolidWorks Help: Reassigning Mouse Gestures*.

This illustration shows mouse gestures assigned for **OK** (active), **Cancel** (opposite OK), and a macro with a custom icon (above **Cancel**).



Recent Documents and Open Documents

The Recent Documents list can now hold up to 16 items, and you can pin favorite documents to the list. You can close open documents from the Open Documents list. You can open the folder that contains an item from either the Recent Documents or the Open Documents list.

Items that you highlight from the keyboard are distinct from those you highlight with the mouse in the Recent Documents and Open Documents browsers.





In this illustration, the item on the left is selected with the keyboard and opens if you press **Enter**. The item on the right is selected with the mouse by hovering over the item, and opens if you click the item.



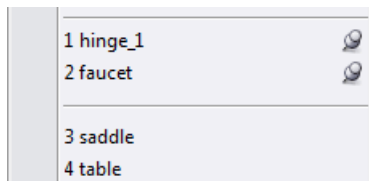
Pinning and Unpinning Recent Documents

You can pin a document to the Recent Documents list. This lets you create a set of favorite documents.

Do one of the following:

- Click **File**, hover over the item for the document, and click **Pin**  or **Unpin** .
- Click **File** > **Browse Recent Documents** or press **R**, hover over the item for the document, and click **Pin**  or **Unpin** .

A pinned document remains at the top of the Recent Documents list until you unpin it.





Closing Documents from the Open Documents List

You can close documents from the Open Documents list. You can close more than one document at a time from the Open Documents browser.

The Open Documents browser is a convenient way to close multiple documents because you do not have to locate each document window individually.



Do one of the following:

- Click **Window**, hover over the item for an open document, and click **Close** .
- Click **Window** > **Browse Open Documents** or press **Ctrl + Tab**, hover over the item for each open document you want to close, and click **Close** . Click in the graphics area to close the Open Documents browser.

Opening Containing Folders for Documents

You can open the containing folder for a document from the Recent Documents list or the Open Documents list.

Do one of the following:

- Click **File**, hover over the item for the recent document, and click **Show in Folder** .
- Click **Window**, hover over the item for the open document, and click **Show in Folder** .
- Click **File** > **Browse Recent Documents** or press **R**, hover over the item for the recent document, and click **Show in Folder**.
- Click **Window** > **Browse Open Documents** or press **Ctrl + Tab**, hover over the item for the open document, and click **Show in Folder**.

Search Commands

Search Commands lets you find and run commands from SolidWorks Search or locate a command in the user interface.

These features make it easy to find and run any SolidWorks command:

- Results are filtered as you type and typically find the command you need within a few keystrokes.
- When you run a command from the results list for a query, **Search Commands** remembers that command and places it at the top of the results list when you type the same query again.
- Search shortcuts let you assign simple and familiar keystroke sequences to commands you use regularly.

For an example of how to use **Search Commands**, see [Search Commands Example](#) on page 17.

Searching for Commands

To search for a command:

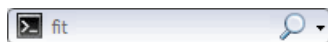
1. If **Search Commands** is not already active, in the SolidWorks Search list, select **Commands**.



To activate **Search Commands** from the keyboard, press **W** or **S**. You can also assign another keyboard shortcut.


If you do not want the **S** key associated with **Search Commands**, click **Tools > Customize**. In the Customize dialog box, on the **Toolbars** tab, clear **Activate Command Search when the shortcut bar is launched**.

2. Start typing a search query in the SolidWorks Search box.



As you type, **Search Commands** displays a results list and updates it in response to your typing. For example, when you type `fit`, the results list displays the commands that include that sequence of characters.

3. Do one or more of the following:

- To locate a command in the SolidWorks user interface, hover over the command in the results list and click **Show Me** .
- To run a command, click the command, or highlight the command and press **Enter**.



If a command cannot run, it is unavailable in the results list and appears lower in the list. For example, many commands cannot run unless a document is open.

- To add a command to a toolbar, drag the command from the results list and drop it on the toolbar.

Assigning Search Shortcuts to Commands

To assign a search shortcut to a command:

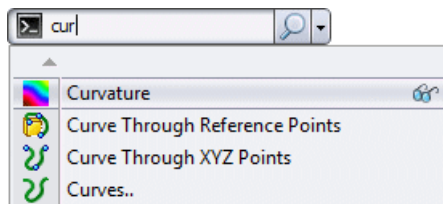
1. Click **Tools > Customize**, and select the Keyboard tab.
2. Navigate to the command to which you want to assign a search shortcut.
See *SolidWorks Help: Customize Keyboard*.
3. In the **Search Shortcut** column for the command, type the shortcut you want to use, then click **OK**.

Search Commands Example

This example shows how you might use Search Commands to find and run the **Curve Through Reference Points** command.

For full instructions on using Search Commands, see [Search Commands](#) on page 16.

With a part or assembly open, start typing `curve through reference points` in Search Commands. As soon as you type `cur`, the results list displays only those commands that include the character sequence "cur," and **Curve Through Reference Points** appears near the top of the results list.

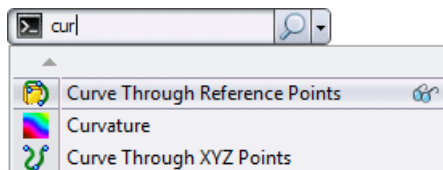


If you click Show Me  for **Curve Through Reference Points**, a red arrow indicates the command in the user interface.



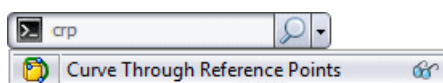
If it is not already visible, the toolbar or menu containing the command opens.

If you run **Curve Through Reference Points** from the results list, the command appears at the top of the list the next time you search using `cur`.



If you assign **Curve Through Reference Points** the search shortcut `crp`, the command is the only one found in the results list when you search by typing `crp`.

Because you can activate Search Commands using the **W** or **S** key, and the first item in the results list is automatically highlighted, you can run **Curve Through Reference Points** without leaving the keyboard. Press **W** or **S**, then type `crp`, then press **Enter**.




Select All

Select All selects everything in the graphics area, as if you made a box selection covering the entire area.

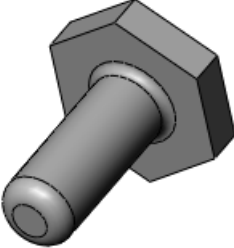
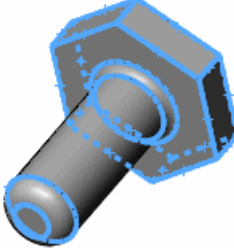
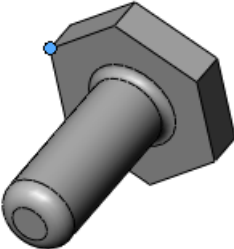
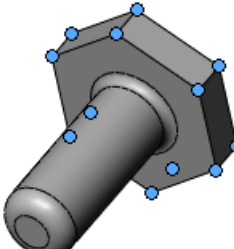

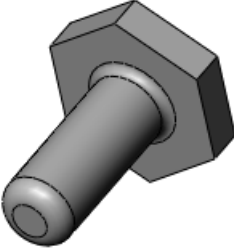

For parts, **Select All** respects any active selection filters when it makes the selection. For assemblies, **Select All**, like box selection, selects all components.

To select everything in the graphics area:

- Click **Edit > Select All**, or press **Ctrl+A**.

 To limit the selection to specific entities in a part, pre-select one or more entities in the graphics area. For example, pre-select an edge and a vertex to limit the selection to all edges and vertices.

This table provides examples of **Select All** performed in different circumstances on a part.

Description	Before	After Select All
No filters active, and no pre-selection		
No filters active, and a vertex pre-selected		
Filter Faces  active, and no pre-selection		

Spanning and Fitting to Displays

You can automatically position and size the main SolidWorks window to span two displays. You can also position and size two document windows conveniently across one or two displays.

Understanding Span Displays

If you have multiple displays, you can use **Span Displays** to span two displays without manually resizing the SolidWorks window.

If you have two displays configured side-by-side, **Span Displays** spans the two displays using the height of the display with the smallest vertical resolution, as shown.



If you have two displays configured one above the other, **Span Displays** spans the two displays using the width of the display with the smallest horizontal resolution, as shown.



If you have more than two displays, **Span Displays** spans only two of them, and uses the following priority order to determine which additional display to span: right, left, bottom, top. For example, if a display is available to the right of the current display, **Span Displays** spans that display. If displays are available above and below the current display, **Span Displays** spans the display below.

When the SolidWorks window partially or fully spans two monitors:

- A document that you drag from Windows Explorer opens on the monitor where you drop it.
- Open documents stay on their current monitor when you use **Windows > Tile** or **Windows > Cascade** to organize them.



You can also **Alt** + click one of the **Tile** buttons in a document window to span two displays and tile the document window at the same time.


Spanning Displays

To span two displays:

Click **Span Displays**  or **Window > Span Displays**.

The **Span Displays** button is on the right side of the SolidWorks window's title bar.



The icon changes to  if the only available displays are configured one above the other.

Restoring Spanned Displays

To restore the SolidWorks window to its previous size and position when it spans two displays:

Click **Restore**  or **Window > Restore**.

The **Restore** button is on the right side of the SolidWorks window's title bar.



Tiling Windows in Displays

You can tile a document window so it occupies half of one display, or all available space on one of two displays. You can also resize the SolidWorks window to fill a single display or to span two displays at the same time as you tile a document window.

This functionality is useful for arranging two document windows to take advantage of the available display area. For example, you can tile a document window containing an assembly on one display and a document window containing a part from that assembly on the other. This is also a good way to arrange two document windows when you have more than two document windows open. (**Windows > Tile**, in contrast, arranges all the open documents.)

To tile a document window in the display:

Click or **Alt** + click **Tile Left** , **Tile Right** , **Tile Top** , or **Tile Bottom**  in the document window's title bar.

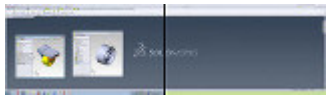
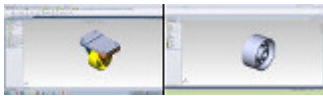

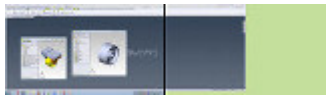










If two displays are configured side-by-side, or if only one display is connected, the available buttons are **Tile Left** and **Tile Right**. If two displays are configured one above the other, the available buttons are **Tile Top** and **Tile Bottom**.

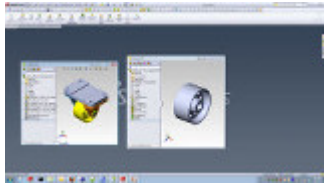
When you click a **Tile** button, the document window is resized and repositioned but the SolidWorks window is not changed.

When you **Alt** + click a **Tile** button, the document window and the SolidWorks window are resized and repositioned. If multiple displays are connected, the SolidWorks window spans two displays. If only one display is connected, the SolidWorks window is maximized.

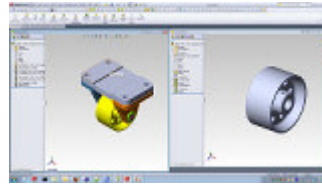
The following table describes the results for different display configurations and illustrates the results when you click or **Alt**+click **Tile Left** and **Tile Right** to tile two documents.

Display Configuration	Result When Clicking	Result When Alt+clicking
<p>Two or more displays available. The SolidWorks window fully spans two displays.</p>	<p>The document windows tile to the appropriate display.</p>	<p>The document windows tile to the appropriate display.</p>
		
<p>Two or more displays available. The SolidWorks window spans one display plus a substantial part of a second display.</p>	<p>The document windows tile to the appropriate display.</p>	<p>The SolidWorks window resizes to span two displays and the document windows tile to the appropriate display.</p>
		
<p>Two or more displays available. The SolidWorks window spans one display plus a small part of a second display.</p>	<p>The document windows tile to occupy 50% of the available display area. (The window on the right does not attempt to tile to the small area occupied by the SolidWorks window on the second display.)</p>	<p>The SolidWorks window resizes to span two displays and the document windows tile to the appropriate display.</p>
		
<p>Two or more displays available. The SolidWorks window occupies all or part of one display only.</p>	<p>The document windows tile to the appropriate half of the display.</p>	<p>The SolidWorks window resizes to span two displays and the document windows tile to the appropriate display.</p>
		

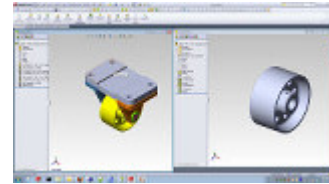
One display available. The SolidWorks window is maximized on the display.



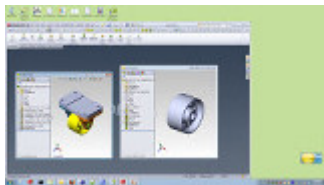
The document windows tile to the appropriate half of the display.



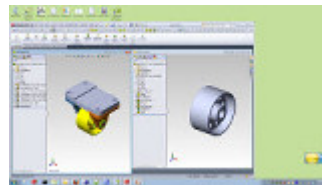
The document windows tile to the appropriate half of the display.



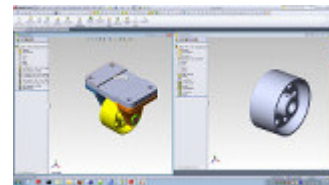
One display available. The SolidWorks window is not maximized on the display.



The document windows tile to the appropriate half of the display.



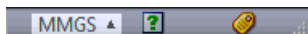
The SolidWorks window maximizes to fill the display and the document windows tile to the appropriate half of the display.



Unit System in the Status Bar

Unit System shows the unit system for the active document in the status bar and lets you change or customize the unit system.

Unit System is located next to **Quick Tips** in the status bar.



For example, **MMGS** is displayed when the unit system for the active document is millimeters, grams, seconds.

To view the full unit system name in a tooltip, hover over **Unit System** with the pointer.

To select a different unit system or to customize the unit system, click **Unit System**, then select a unit system, or click **Edit Document Units** to open the Document Properties - Units dialog box.

SolidWorks Fundamentals


New features include display improvements, better low-resource warnings, and better autosave and document recovery processes.


This chapter includes the following topics:

- **Application Programming Interface**
- **Documentation**
- **Document Recovery**
- **Blank Custom Properties**
- **Display**
- **SolidWorks Resource Monitor**

Application Programming Interface

Enhancements in the SolidWorks 2012 Application Programming Interface (API) are the ability to:

- Determine whether any buttons of a CommandGroup are enabled.
- Get and set the faces to exclude from **Flat-Pattern**  features.
- Add objects to selection lists without preselecting them through the user interface.
- Sort bill of materials, hole, weldment cut list, and general tables.
- Specify equal spacing between pattern instances in circular pattern features.
- Insert DXF/DWG image features into a part or drawing.
- Select chains of entities attached to sketch segments.
- Close drawings without unloading their references from memory and reopen them.
- Isolate changed dimensions.
- Specify configuration names when setting the files to open in response to SolidWorks events.
- Specify a rebuild option when activating documents.
- Get system information.
- Suppress multiple notifications while dragging or spinning the slider of number boxes on PropertyManager pages.
- Anchor notes.
- Automatically insert center marks.
- Specify the math transform to use in interference detection.
- Get and set display modes, visual properties, transparency states, and visibility states of specified components in specified display states for a specified scope.
- Specify balloon options when creating balloon, auto balloon, and stacked balloon annotations.
- Get the number of virtual sharp witness lines and their geometry data.
- Insert magnetic lines in drawing sheets.

- Insert bend table annotations in drawing views.
- Modify, insert, and delete the features, components, and reference entities of Smart Components.
- Specify whether to color section view caps.
- Get the fixed face of **Flatten-Bends**  features.


See *SolidWorks 2012 API Help Release Notes* for late-breaking updates.

Documentation

New Tutorials

SolidWorks 2012 includes a new PhotoView 360 and Appearances tutorial and an enhanced Animation tutorial.

To access these tutorials:

1. Click **Help**  > **SolidWorks Tutorials**.
2. Click one of the following:
 - **All SolidWorks Tutorials (Set 1)** (Animation Tutorial)
 - **All SolidWorks Tutorials (Set 2)** (PhotoView 360 and Appearances)
3. Select the tutorial from the list.

Routing

Changes to routing documentation include a new Getting Started section. This section describes best practices for setting up and managing your routing projects.

Document Recovery

Auto-recover is simpler and more reliable. Document recovery information in the SolidWorks Recovery tab of the **Task Pane** is easier to understand and respond to.

You can determine how often auto-recover information is saved by setting an interval in minutes.



In SolidWorks 2011 and earlier, you can control how often auto-recover information is saved based on the number of changes you make to a document. This option is not available in SolidWorks 2012.

You can also determine where auto-recover information is saved on your system.



The **Auto-recover folder** must be a folder on your local system and cannot be a network location.

See *SolidWorks Help: Backup/Recover Options*.

When you start SolidWorks after the system has terminated unexpectedly, any available auto-recovered documents are listed under **Document Recovery** in the SolidWorks Recovery tab of the **Task Pane**.

To open an auto-recovered document, double-click the document in the **Document Recovery** list, or right-click the document and click **Open**.

You can then choose to save the document either by using **Save** to replace the original file or by using **Save As** to create a new file.

To delete an auto-recovered document, right-click the document and click **Delete**.

Blank Custom Properties

You can create blank custom or configuration-specific properties in the Summary Information dialog box.

This functionality is useful, for example, when you want to create a template that contains custom properties but you do not want to give those properties specific starting values.

To add a blank custom or configuration-specific property:

1. Click **File > Properties**.
2. On the Custom tab or the Configuration Specific tab:
 - a. Click <Type a new property> in the **Property Name** column for the blank property row, then type a custom property name.
 - b. Select a property type in the **Type** column for the new property.
3. If necessary, repeat step 2 to add more custom properties.
4. Click **OK**.

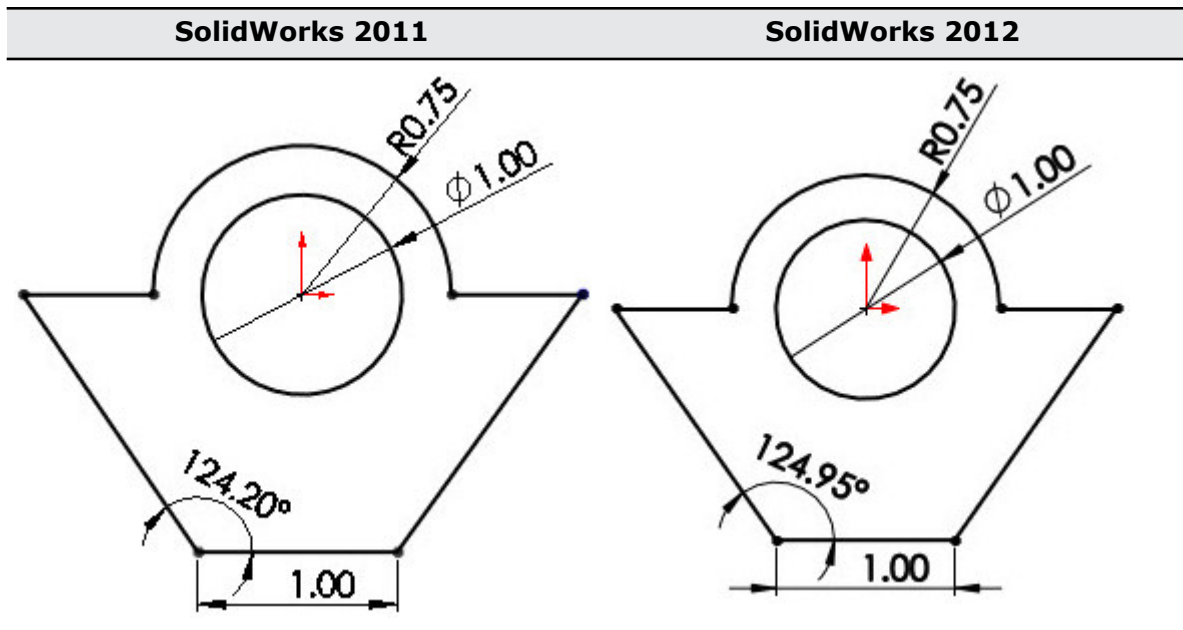
Display


Enhancements to display capabilities include improvements to the look of the graphics area, changes to section display and coloring, better images for curvature analysis, and the wider availability of ambient occlusion.

Visual Improvements to the Graphics Area

The graphics area, and sketch mode in particular, has an improved appearance.

Lines, arrows, text, and numbers are anti-aliased. Items such as dimension leader lines, balloons, origins, points, and spline controls have an improved look.





Additionally, the list of scenes available from **Apply scene**  (Heads-up View toolbar) is organized into categories that match the structure of the **Scenes** folder.

Section Display and Coloring

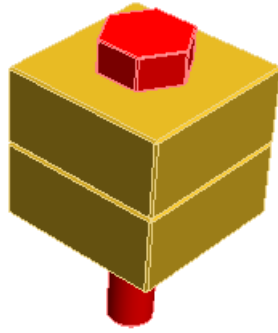
Section views are available in a part or assembly when the Section View PropertyManager is not open, and persist when you save and re-open the document. You can display a part or assembly with or without section caps. If you display section caps, you can turn the cap color on or off.

To control how section views display:

1. Click **Section View**  (View toolbar) or **View > Display > Section View**.
2. In the PropertyManager, select options to define your section views. See *SolidWorks Help: Section View PropertyManager (Models)*.
3. Select or clear **Show section cap**.
4. If you selected **Show section cap**, select or clear **Keep cap color**.
5. Click .

The section view remains visible for the part or assembly. The following illustrations show the possible section view displays for an assembly.

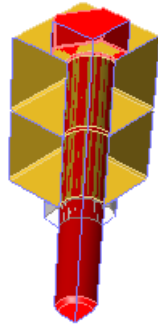
Section views hidden.



Show section cap selected and **Keep cap color** cleared. (Previously this was the only available display after you close the Section View PropertyManager.)



Show section cap and **Keep cap color** cleared.

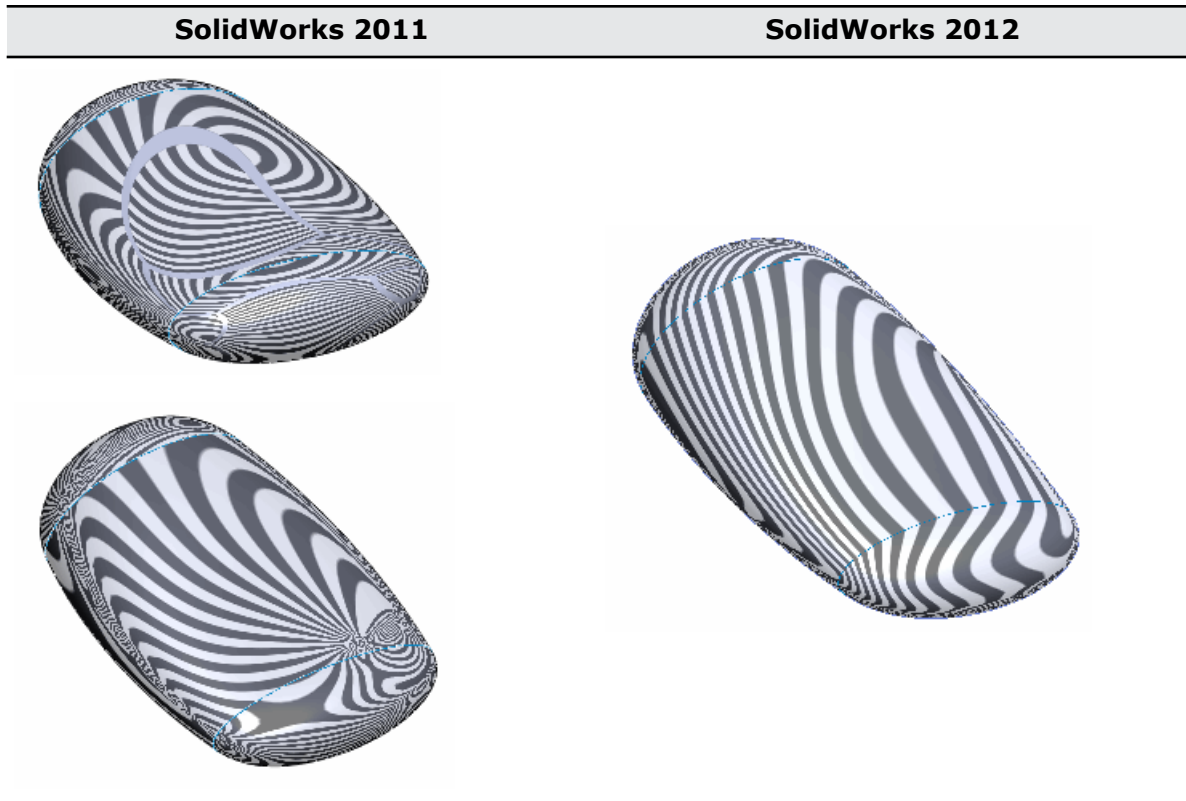


Show section cap and **Keep cap color** selected.



Zebra Stripes and Curvature Analysis

Improved zebra stripe images let you perform curvature analysis more easily. Stripes no longer converge at the poles and are less visually distracting.



The Zebra Stripes PropertyManager has these new options:

- You can orient the zebra stripes horizontally or vertically by selecting either **Horizontal Stripes** or **Vertical Stripes**.
- You can use an alternative environment image instead of zebra stripes by selecting **From file** in the **Environment** list and browsing to a file.

Ambient Occlusion

Ambient occlusion is available in all scenes when you use RealView graphics. Objects appear as they would on an overcast day.



Ambient occlusion is a global lighting method that adds realism to models by controlling the attenuation of ambient light due to occluded areas.

The effects of ambient occlusion depend on other factors such as the nature of the model itself, the appearances you apply to the model, the scene, and the lighting. You might need to adjust some of these other variables to get the desired result.

- Ambient occlusion often looks best on solid color or non-reflective surfaces. Highly reflective surfaces, in contrast, can diminish the shadows in occluded areas. However, the methods used to render ambient occlusion can sometimes result in artifacts, and these can be more visible on solid color surfaces.
- Too many lights, or lighting that is too bright, might reduce the impact of ambient occlusion by hiding the shadows it creates.

You can use two different quality levels for ambient occlusion: draft and default. Draft displays models faster but with less visual fidelity.

To use ambient occlusion:

1. Click **View Settings**  (Heads-up View toolbar) or **View > Display**, then click **RealView Graphics**.
2. Click **View Settings**  (Heads-up View toolbar) or **View > Display**, then click **Ambient Occlusion**.



RealView Graphics is only available with graphics cards that support RealView Graphics display.



To change the display quality level for **Ambient Occlusion**, click **Tools > Options**. On the System Options tab, click **Display/Selection**, then select or clear **Display draft quality ambient occlusion**.

Ambient Occlusion Examples

Model with solid color surfaces

Model with appearances



Ambient Occlusion off



Ambient Occlusion on and using default quality




Ambient Occlusion on and using draft quality




SolidWorks Resource Monitor

The SolidWorks Resource Monitor utility monitors resources that your SolidWorks software or your system is using. When you are running low on resources, the SolidWorks Resource Monitor provides messages in the notification area of the Windows taskbar that help you take action to avoid a system failure or a loss of data.

The SolidWorks Resource Monitor monitors these resources:





Resource	Description
System memory	The combined total of physical RAM and page file size. In Windows, this combined total is known as the Commit Charge.
GDI handles	A finite resource used to draw user interface items on the screen.
Process memory (SolidWorks 32-bit only)	The maximum amount of memory available to the SolidWorks software (2GB).  There is no equivalent for SolidWorks 64-bit as the memory available to a process is only limited by the system memory.


The SolidWorks Resource Monitor's icon in the notification area is color-coded to indicate the availability of resources:

Color	Description
 Green	Sufficient resources of all types are available.
 Yellow	One or more resources are running low. You should take action to free resources.
 Red	One or more resources are almost completely consumed. If you continue to work you are likely to experience a system failure or loss of data, so you must take action to free resources. In addition to the messages provided by the monitor in the notification area, messages appear in the SolidWorks software if you attempt to execute any commands.

The SolidWorks Resource Monitor displays messages that are specific to the type of resource that has run low. Follow the directions in the message to correct the resource problem.

For example, if system memory is low, close non-SolidWorks programs or close documents within SolidWorks. If the number of available GDI handles is low, close documents within SolidWorks.

 To display the most recent message again, click the SolidWorks Resource Monitor icon - , , or  - in the notification area.

 The SolidWorks Resource Monitor has no user interface other than the icon and messages. Nothing appears on the screen if you double-click the icon, or click it before a message has appeared.

This chapter includes the following topics:

- **Installation Manager**
- **Administrative Image Option Editor**

Installation Manager

SolidWorks Installation Manager is enhanced.

- You can completely uninstall the SolidWorks software or customize the software removal by removing one or more of the following items: Program Files and Folders, registry entries, data folders such as SolidWorks Toolbox, and previously downloaded installation files and folders. By default, the SolidWorks Installation Manager removes Program Files and Folders.
 1. To uninstall SolidWorks in Windows Vista and later, click **Start > Control Panel > Programs > Programs Features**. For Windows XP, click **Start > Control Panel > Add or Remove Programs**.
 2. From the Summary screen in the SolidWorks Installation Manager, click **Change**. Select items to remove from the Advanced Options screen.
- You can specify which standards to support before installing the SolidWorks Toolbox. By default, all standards are selected for a new installation of the Toolbox. To select standards:
 1. In the Summary screen, expand **Products** and click **Change**.
 2. In the Product Selection screen, expand **SolidWorks > SolidWorks Toolbox** and select standards from a list.
- The installation of the Workgroup PDM Server requires a validation code. The Installation Manager can obtain the code automatically. If an Internet connection is not available, you can obtain the code by email.

Administrative Image Option Editor

SolidWorks Administrative Image Option Editor is enhanced.

- Administrators can use new uninstall options to perform a complete or custom uninstall on client computers. The uninstall options are available in the Client Installation Options section of the Option Editor. To view the options, click **Yes, perform a custom uninstall** and then select items for removal.
- Administrators can also preselect SolidWorks Toolbox standards from a list.

This chapter includes the following topics:


- **Large Design Review**
- **Usability and Workflow Improvements**

Large Design Review

Large Design Review lets you open very large assemblies quickly, while still retaining capabilities that are useful when conducting design reviews of assemblies.

When you open an assembly in Large Design Review mode, you can:

- Navigate the FeatureManager design tree
- Measure distances
- Create cross sections
- Hide and show components
- Create, edit, and play back walk-throughs

To open an assembly in Large Design Review mode, click **Open**  (Standard toolbar) or **File > Open**. In the dialog box, select the assembly you want to open, and then, in **Mode**, select **Large Design Review**.

Assembly and Component Information

Information about the assembly and its components is limited in Large Design Review.

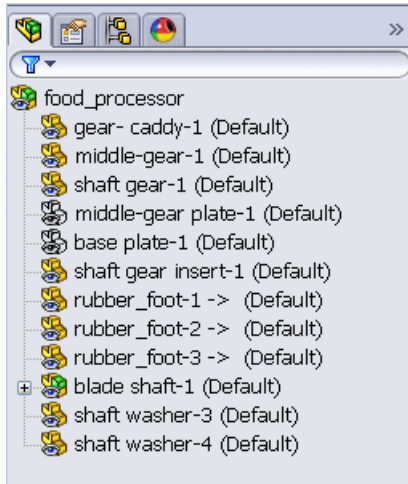
FeatureManager Design Tree

The FeatureManager design tree displays information about the assembly components and hierarchy.

Information displayed in the tree includes:

- Component names
- Assembly and subassembly hierarchy

An eye overlay appears on the icons of all components when the assembly is opened in Large Design Review mode.




Limitation Information about the following items is not shown in the FeatureManager design tree:

- Assembly features
- Component patterns
- Mates

Assembly Statistics

In AssemblyXpert, you can see assembly statistics such as number and type of components, depth of the assembly hierarchy, etc.

Click **AssemblyXpert**  (Large Design Review tab of the CommandManager) or **Tools > AssemblyXpert**.

Limitation The reported assembly statistics are approximate. They can differ from the numbers reported when the document is opened in resolved or lightweight mode.

AssemblyXpert diagnostic tests are not available in Large Design Review.

Component Properties

You can view component properties, but you cannot change them.

Right-click a component and click **Component Properties** .

You can see information such as:

- Model Document Path
- List of configurations

Limitation You cannot change the properties. For example, you cannot:

- Switch configurations
- Switch between flexible and rigid


Reviewing Assemblies

Most of the capabilities in Large Design Review mode are limited versions of existing SolidWorks capabilities. Snapshots are a new capability available only in Large Design Review.

Most available tools appear on the Large Design Review tab of the CommandManager.

Measurements


You can measure the distance between entities in Large Design Review.

Click **Measure**  or **Tools > Measure**. When you select two entities (vertices, edges, or faces) the software reports the *approximate* distance between them.

Limitation Measurements reported in Large Design Review are approximate. To obtain accurate measurements, you must resolve the components.

Section Views

You can create section views in Large Design Review.

Click **Section View**  or **View > Display > Section View**. You can select one or more of the three standard planes to section the model. You can cap the ends of the section view.

Limitation You cannot select other planes to section the model.


Isolate

You can use **Isolate** to focus on selected components.

Select one or more components, and then right-click and click **Isolate**.


Walk-throughs

You can create Walk-throughs in Large Design Review just as you do in resolved mode.

Click **Add Walk-through**  or **View > Lights and Cameras > Add Walk-through**. Walk-throughs can be played back in both Large Design Review and resolved modes, regardless of which mode they were created in.

Snapshots

Snapshots are a new capability available only in Large Design Review. You can take snapshots of locations that you want to return to.


A snapshot captures the state of the graphics area, similar to a custom named view, and saves it in a **Snap** feature on the DisplayManager tab  of the Manager Pane. Unlike a custom named view, if some parts are hidden when you take a snapshot, those parts are always hidden when you return to that snapshot.

Click **Take Snapshot**  or **View > Lights and Cameras > Take Snapshot**, or press **Alt + Spacebar**.

You can:

- Right-click a **Snap** feature and add a comment (140 characters maximum)
- Double-click a **Snap** feature to return to that view of the assembly
- Click-pause-click a **Snap** feature to rename it
- Hover over a **Snap** feature to see the comments and a preview




 A snapshot named **Home** is created whenever you open an assembly in Large Design Review. If you make changes such as hiding components, zooming, and rotating, you can double-click the **Home** snapshot to return the graphics area to the state it was in when you opened the assembly. You cannot remove or rename the **Home** snapshot and you cannot add comments to it.

For more information about creating snapshots, see *SolidWorks Help: Snapshots*.


Save


You can save an assembly in Large Design Review only when you create or change Walk-throughs or snapshots.

Click **Save**  (Standard toolbar) or **File > Save**.

Component Changes


While an assembly is open in Large Design Review, you can make changes to its components. New tools help you to visually indicate which components have been modified and to update the graphics data of dependent components.

To make changes to a component, right-click it in the FeatureManager design tree and click **Open Part** . The component opens in a separate window where you can modify the component and save the changes.

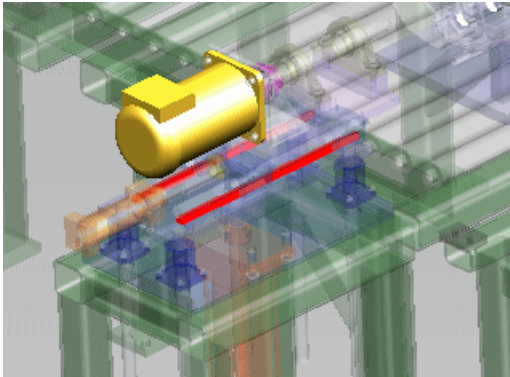
 In Large Design Review, you cannot edit components in the context of the assembly.

Filtering Modified Components

In the assembly window, you can use the new **Filter Modified Components** tool to visually indicate which components have been modified.

 **Filter Modified Components** is available only in assemblies that have been opened in Large Design Review mode, and only when one or more components have been modified.




When **Filter Modified Components** is enabled and you edit and save a component in a separate window, the changed component remains opaque in the assembly window, while unchanged components become transparent. Using **Filter Modified Components** allows you to see more easily which components have been modified.



Modified components include:

- Components that are detected as out of date when you open the assembly in Large Design Review
- Components that you modify and save in a separate window during the current Large Design Review session

To filter modified components:

1. Click **Filter Modified Components**  or **View > Modify > Filter Modified Components**.
2. In the PropertyManager:
 - a) Select **Enable**.
Modified components remain opaque; unmodified components become transparent.
 - b) Use the slider to adjust the transparency.
 -  Select **Dynamic** to see the transparency level change in the graphics area as you move the slider in the PropertyManager.
3. Click .

-  To turn off the transparency, click **Filter Modified Components**  again and clear **Enable**.

LDR Update

You can use **LDR Update** to update the graphics data of a dependent component that becomes out of date due to changes to its parent component.

If changes that you make to one component influence a dependent component, then the graphics data in Large Design Review might become out of date for the dependent component.

Examples of changes that might require **LDR Update**:


- A change to an equation in one part that influences a dimension in another part.
- A change to a part that is the seed component for a mirrored part.





To update the graphics data:

1. Right-click the assembly name at the top of the FeatureManager design tree.
2. Click **LDR Update**.

Opening and Resolving Components

Selective Open is available on the Large Design Review tab of the CommandManager. Previously it was in the Open dialog box. New tools let you selectively open as lightweight, or set all components to resolved or to lightweight.

 The following commands cause the software to exit Large Design Review. You might notice a delay while component data is loaded. To return to Large Design Review, close and reopen the assembly.

- **Selective Open** . In the Selective Open dialog box, select components to fully load. Other components are not loaded and become hidden.
- **Selective Open in Lightweight**  (on **Selective Open** flyout tool). In the Selective Open in Lightweight dialog box, select components to load as lightweight. Other components are not loaded and become hidden.
- **Set All to Resolved** . All components are fully loaded.
- **Set All to Lightweight**  (on **Set All to Resolved** flyout tool). All components are loaded as lightweight.

Usability and Workflow Improvements

Various assembly areas have been enhanced to add shortcuts, remove limitations, and streamline workflows and dialogs.

Creating Subassemblies

You can preselect components in the parent assembly's graphics area when creating a subassembly. Previously, you could preselect only components in the parent assembly's FeatureManager design tree.

In an assembly, in the graphics area, select several components. Then right-click and click **Form New Subassembly**.

Editing Flexible Subassemblies


You can edit flexible subassemblies in the context of their parent assemblies.

Face and Edge Names in Lightweight Components

When replacing components in assemblies, if you select **Re-attach mates**, the software can access previously assigned face and edge names in lightweight components.

If you have previously assigned names to mated entities (faces or edges), then when you replace one component with another, you can reattach the mates automatically without resolving the original component.

Previously, the mates would not reattach automatically because the software could not access the entity names in lightweight mode.

 You must save the assembly in SolidWorks 2012 to access the face and edge names in lightweight components.

Finding Missing References

While opening an assembly that has multiple missing references, the process of finding those references is simplified.

The software tracks each folder where you find a missing reference and checks those folders for subsequent missing references.

Previously, the software tracked only the most recent folder. If the missing references were in several folders, you often had to browse to the same folder multiple times.

Folders

When adding a folder in the FeatureManager design tree, you can preselect non-sequential components or mates to place in the folder.

In an assembly, in the FeatureManager design tree, select several non-sequential components or several non-sequential mates. Right-click and click **Add to New Folder**.

Hiding and Showing Components

In the graphics area, you can press **Tab** and **Shift + Tab** to hide and show components.

To hide a component, move the pointer over it and press **Tab**. To show a component, move the pointer over an area containing a hidden component and press **Shift + Tab**.

You can use these commands regardless of the component's state (resolved, lightweight, SpeedPak, or Quick view). The commands are not available when another command is active.

Hiding the Broken Reference Indicator

A system option lets you hide the x that appears in the FeatureManager design tree to indicate broken references.

Click **Tools > Options > System Options > External References** and clear **Show "x" in feature tree for broken external references**.

Mate Reference Propagation

When you insert a subassembly into a higher-level assembly, mate references defined in the subassembly's components are available for automatic mating.

Previously, only mate references defined in the subassembly's top level were available.

Modifying Component Dimensions

You can edit the dimensions of a multiple-configuration component in the assembly or drawing environment, regardless of which configuration is active in the component document.

Previously, in an assembly or drawing, you could not edit the configured component dimensions unless the configuration you were editing was also the active (open or last saved) configuration in the component document.

In an assembly with a component that has multiple configurations, double-click a dimension that has a different value for each configuration. In the Modify dialog box, you can set a new value for the dimension and specify to which configurations the change applies.

Opening Assemblies

In the Open dialog box, checkboxes for various assembly modes have been replaced by the **Mode** drop-down list. **Quick view / Selective open** has been moved to Large Design Review.

Click **Open**  (Standard toolbar) or **File > Open**. In the dialog box, select the assembly you want to open. The **Mode** drop-down list appears. Available modes include:

- **Resolved**
- **Lightweight**
- **Large Assembly Mode**
- **Large Design Review**

 The **Use Speedpak** checkbox is shown only for modes that allow it.

Quick view / Selective open is no longer available in the Open dialog box for assemblies. Instead, in **Mode**, select **Large Design Review**. On the CommandManager, click

Selective Open . See [Opening and Resolving Components](#) on page 38.

Reloading a Component after Discarding Changes

You no longer need to manually reload a component in the assembly file when you discard changes made in the component file. When you close the component file without saving it, a dialog box asks if you want to keep or discard the changes in the assembly.


1. Open an assembly, then open one of its components in a separate window.
2. Make changes to the component.
3. Click **File > Close**.

A dialog box asks if you want to save the changes.

4. Click **No**.

A dialog box explains that because the component is open in another window, changes you made to the component will still appear in that other window.

5. Click one of the following:

Yes	Keeps the changes in the assembly file, even though you did not save them in the component file. For example, click Yes when you want to make more changes to the component, but you need to make them in the context of the assembly.
	 When you save the assembly file, changes to the component are saved to the component file.
No	Reloads the unchanged component file into the assembly file, so that the changes no longer appear in the assembly.
Cancel	Keeps the component file open so you can make other changes or save the file.

Saving Components Edited Within Assemblies

The Resolve Ambiguity dialog box has been eliminated.

When you save a component that you are editing in the assembly window, the component is saved. You are no longer asked whether you want to save the component file or the assembly file.

SmartMates

To invoke SmartMates between two components already in an assembly, you can press and hold **Alt** *after* selecting a face to drag. Previously, you had to press and hold **Alt** *before* selecting the face.

Also, the sensitivity in SmartMates mode has been adjusted to prevent the part you are dragging from jumping excessively as you drag it across an assembly that contains multiple possible mating solutions. It now takes a few seconds for the SmartMates preview to appear when you hover the pointer over a potential mate partner.

Smart Fasteners

When inserting Smart Fasteners, the software recognizes more holes that were not created with the Hole Wizard or Hole Series tools, such as holes created by various mirroring and patterning operations.

Available in SolidWorks Premium.

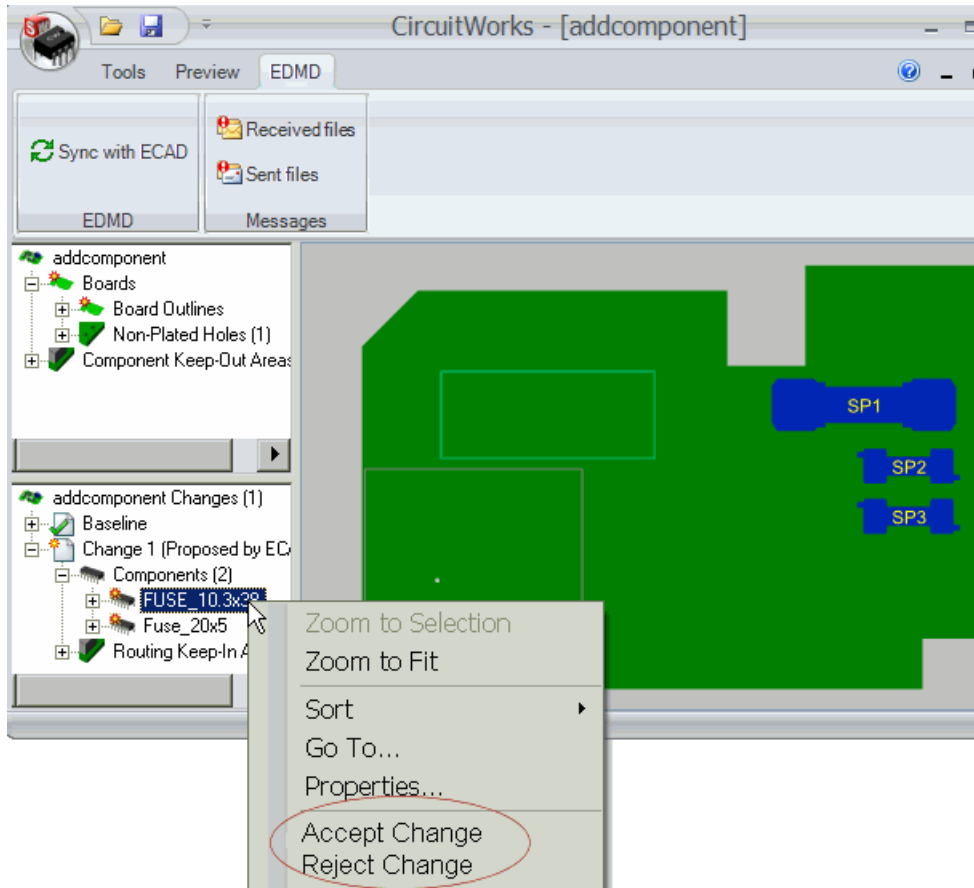
This chapter includes the following topics:

- **ProStep EDMD Import and Export**

ProStep EDMD Import and Export

ProStep EDMD (*.idx) files are supported by major ECAD vendors. Use this file format to collaborate with your ECAD designers on an ECAD design. You can export CircuitWorks™ data to ProStep EDMD files and you can import ProStep EDMD files into CircuitWorks. CircuitWorks communicates proposed changes to the ECAD files to the ECAD designer and updated changes to ProStep EDMD files are communicated back to CircuitWorks when you import the ECAD data.

To open a ProStep EDMD file, click **CircuitWorks** > **Open ECAD File**, navigate to the ProStep EDMD file and click **Open**. To set the ProStep EDMD options, in the CircuitWorks Options dialog box, click the ProStep EDMD  tab.



See *SolidWorks Help: ProStep EDMD Files*.

Configurations

This chapter includes the following topics:

- **Modifying Component Dimensions**
- **Equations in Configurations**

Modifying Component Dimensions

You can edit the dimensions of a multiple-configuration component in the assembly or drawing environment, regardless of which configuration is active in the component document.

See **Modifying Component Dimensions** on page 40.

Equations in Configurations

You can configure equations more easily. The new behavior for configuring equations is similar to the existing behavior for dimensions.

See **Equations** on page 78.

SolidWorks Costing

Available in SolidWorks Professional and SolidWorks Premium.

This chapter includes the following topics:

- [SolidWorks Costing Overview](#)

SolidWorks Costing Overview

The SolidWorks Costing tool helps you calculate how much it costs to manufacture sheet metal and machined parts by automating the cost estimation and quotation process.

The tool helps designers make design decisions based on the cost to manufacture, and helps manufacturers create quotes for customers. Whenever you change a design, you can see the new, updated cost immediately, along with a detailed cost breakdown. Additionally, you can generate automatic cost reports.


The Costing tool is driven by manufacturing and material information in templates that determine the manufacturing cost. In the templates you can specify the material used to create the part, the manufacturing processes (such as laser cutting, bending, or milling), and the associated costs of these materials and manufacturing operations. The templates also let you create custom operations such as packaging, ERP entry, painting, or cleaning.

Costing serves different audiences:


- **Designers:** Costing provides “should cost” estimates of how much parts should cost to manufacture. Costing can compare models so you can make decisions based on cost earlier in the design process. You can try “what if” scenarios such as removing features, changing materials, and using different manufacturing processes to see how these affect the cost. The cost estimates are repeatable because the results are based on data in the templates, so you always use the same information to calculate costs.
- **Manufacturers:** Costing creates accurate quotes based on the materials, processes, and other associated costs that are required to manufacture parts. Costing speeds up the quote process compared to manual methods such as using spreadsheets, counting features, and estimating material removed. Costing helps eliminate errors and provides an accurate, repeatable quoting system that you can update whenever new material or labor costs need revision.

The Costing tool is divided into two functional capabilities:

Sheet Metal Costing

Use the **Sheet Metal Costing**  tool for parts that are manufactured from sheet metal. To use this tool, the part must contain sheet metal features such as flanges, bends, and forming tools. Features such as holes and cuts are recognized as manufacturing cut paths for operations such as laser, water jet, and plasma cutting.

Machining Costing

Use the **Machining Costing**  tool for parts that are machined from solid block and plate stock material. Parts machined from blocks use manufacturing techniques such as milling and drilling. Parts made from plate use manufacturing techniques such as laser, water jet, and plasma cutting, in addition to milling and drilling.




The costs calculated by the Costing tool are as accurate as the data in your templates. Although SolidWorks provides pre-populated templates, it is best to create custom templates based on your manufacturer's costs. It is recommended that you work with your in-house manufacturing departments and external manufacturing vendors to create custom templates that reflect actual costs.

See *SolidWorks Help: SolidWorks Costing Overview*.

Setting File Locations

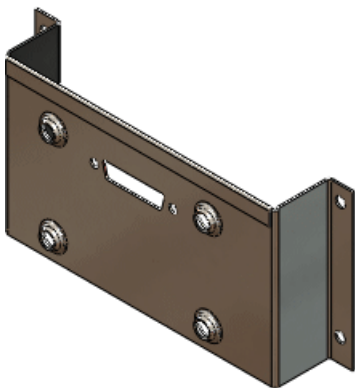
Before you use the Costing tool, you need to set the file locations to use the correct templates for this example.


1. Click **Options**  or **Tools > Options**.
2. On the System Options tab, click **File Locations**.
3. In **Show folders for**, select **Costing templates**.
4. Click **Add** and browse to `install_dir\samples\whatsnew\costing`.
5. Click **OK**.

Evaluating the Cost of a Sheet Metal Part

In this procedure, you run Sheet Metal Costing to determine the cost of manufacturing a sheet metal part. This procedure is helpful for manufacturers.

1. Open `install_dir\samples\whatsnew\costing\costing.sldprt`.
A sheet metal part appears.



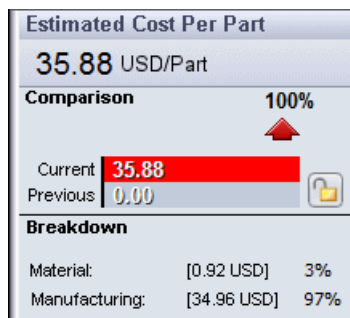
2. Click **Sheet Metal Costing**  (Tools toolbar) or **Tools > SolidWorks Costing > Sheet Metal Costing**.

3. In the Sheet Metal Costing Task Pane, under **Costing Template**, select **sample_shm_metric_waterjet**.

The Costing template contains the procedures you or your manufacturing supplier would use for manufacturing the part. Information such as material cost and sizes, cost of manufacturing operations, and manufacturing setup costs can be included and customized in the template. This Costing template is set up for sheet metal water jet cutting.

The Costing tool estimates the manufacturing cost using the material information in the part.

The Sheet Metal Costing Task Pane displays the **Estimated Cost Per Part** as \$35.88:

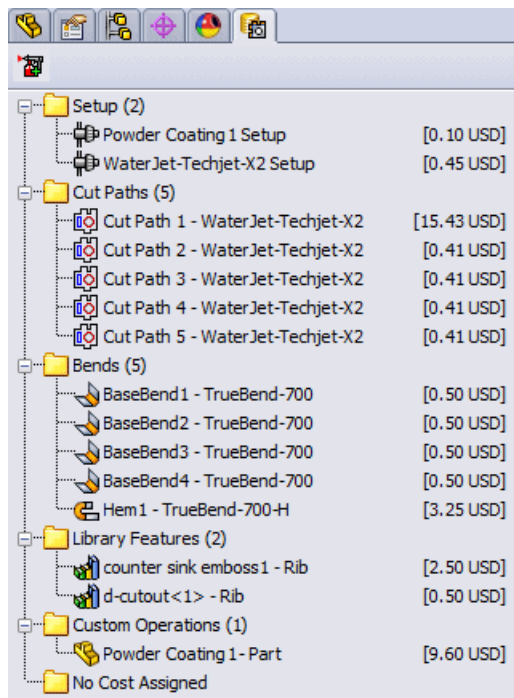


Sheet Metal Costing Task Pane Item	Description	Value
Comparison	Based on the data in the template and the part in the graphics area, the Costing tool calculated the cost of the part as \$35.88.	35.88 USD
Breakdown	A breakdown of the cost calculation indicates that the material accounts for approximately 3% and manufacturing (labor, machine time, and overhead) accounts for approximately 97% of the total cost.	<ul style="list-style-type: none"> Material: 0.92 USD Manufacturing: 34.96 USD

4. In the Sheet Metal Costing Task Pane, under **Material**, notice:

Sheet Metal Costing Task Pane Item	Description	Value
Class	Class of material based on the material that was set for the part in the SolidWorks software.	Steel
Name	Based on the material that was set for the part in the SolidWorks software.	Plain Carbon Steel
Thickness from template	Thickness value in the template that matches or is closest to the part thickness. If no thickness values in the template match the part thickness, the closest thickness value in the template is selected, and a warning icon ⚠ indicates that the thicknesses differ.	1.2100 mm
Material cost	The cost defined for the material at the selected thickness in the template.	3.30

5. The CostingManager in the FeatureManager design tree displays manufacturing features like bends and cut paths that were recognized in the part. In the CostingManager, notice:



Manufacturing Process	Description	Number of Instances	Price Calculation
Setup	<p>Costs associated with manufacturing setups such as setting up machines to run a batch (lot) of parts.</p> <p>Expand the Setup folder and hover over a setup cost to see how the cost is calculated.</p>	2	0.55 USD
Cut Paths	<p>There are five cut paths that are found for water jet cutting.</p> <p>Expand the Cut Paths folder and hover over a cut path to see how the cost is calculated. Select a cut path to highlight the geometry in the model.</p>	5	17.07 USD
Bends	<p>Five bends are recognized, including a hem. Bends are calculated as cost per bend.</p>	5	5.25 USD
Library Features	<p>Two library features are recognized.</p>	2	2.50 USD
Custom Operations	<p>Powder coating is a custom operation applied to this part. It was set up in the template to apply to all parts.</p>	1	9.60 USD
No Cost Assigned	<p>Anything that the Costing tool does not recognize or is not defined in the template. In this part, everything is recognized.</p>	0	

- Keep this part and the Sheet Metal Costing Task Pane open for the next procedure.

Changing Costing Inputs

You can perform “what if” studies to see how changes, such as lot size, might affect cost, without having to actually change the model. When you change Costing inputs, you can see how the changes affect the cost immediately.

- In the Sheet Metal Costing Task Pane, under **Quantity**, set **Lot size** to 50.

2. Press **Enter**.

Because **Total number of parts** is 100, you would need to run 2 lots to produce all of the parts. The **Estimated Cost Per Part** increases because you would incur 2 setup charges, one for each lot.

Switching the Costing Template

You can determine how manufacturing operations affect the cost of your design. For example, you can compare costs of getting the part manufactured by different vendors.

In this example, you check the costs of a vendor that uses laser cutting instead of water jet cutting.


In the Sheet Metal Costing Task Pane, under **Costing Template**, select **sample_shm_metric_laser**.


You change the template to get a cost estimate from a laser cutting manufacturer.

The **Estimated Cost Per Part** decreases, indicating it is cheaper to get the part made at the laser cutting vendor.

Setting a Baseline

You can lock in the value of a design so you can compare future changes to the cost. If you change the design later on, the cost will be compared to the locked cost.

In the Sheet Metal Costing Task Pane, under **Estimated Cost Per Part**, click **Set Baseline**  to lock the **Current** cost.

The **Current** cost is locked as **Baseline** and the button changes to . While the baseline price is set, the part is rotated, flattened, and refolded.

Changing Materials

Now you change the material and custom operations to see how these changes affect the cost with the new vendor.

1. In the Sheet Metal Costing Task Pane, under **Material**, change **Plain Carbon Steel** to **AISI 316 Stainless Steel Sheet (SS)**.

The **Estimated Cost Per Part** increases because stainless steel is more expensive than plain carbon steel. However, because you are using stainless steel, powder coating is no longer necessary and you can remove the powder coating custom operation.



2. In the CostingManager:
 - a) Expand **Custom Operations** .
 - b) Right-click **Powder Coating1 - Part**  and click **Remove Custom Operation**. The **Estimated Cost Per Part** decreases.

Modifying the Costing Template

You can modify the Costing template any time there are changes in cost structures. For example, if the price of steel increases, you can change the cost in the template.

1. In the Sheet Metal Costing Task Pane, under **Costing Template**, click **Launch Template Editor**.
2. In the Costing Template Editor, on the left pane, click **Thickness**.
The prices for each material are defined. You can edit the cost values permanently here, or you can edit them temporarily in the Sheet Metal Costing Task Pane.

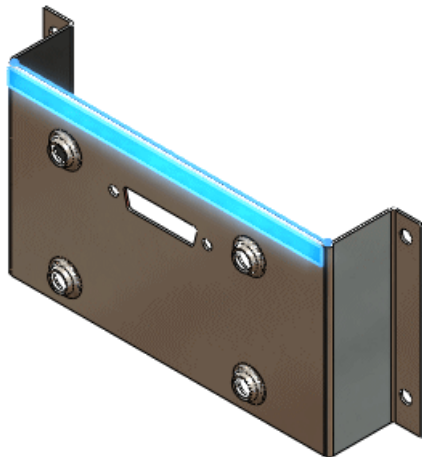
Filters				
	All	All	All	
	Class	Material Name	Thickness (mm)	Cost (USD/kg)
1	Aluminium Alloys	6061-T6 (SS)	1.0600	12.0000
2	Aluminium Alloys	6061-T6 (SS)	1.2100	12.0000
3	Aluminium Alloys	6061-T6 (SS)	1.3700	12.0000
4	Steel	AISI 316 Stainless Steel Sheet (SS)	1.0600	11.0000
5	Steel	AISI 316 Stainless Steel Sheet (SS)	1.2100	11.0000
6	Steel	AISI 316 Stainless Steel Sheet (SS)	1.3700	11.0000
7	Steel	Plain Carbon Steel	1.0600	3.3000
8	Steel	Plain Carbon Steel	1.2100	3.3000
9	Steel	Plain Carbon Steel	1.3700	3.3000
10	Select Class			

3. Close the Costing Template Editor without making any changes.
4. In the Sheet Metal Costing Task Pane, under **Material**, for **Material cost**, type 5.00.
5. Press **Enter**.
Material cost is highlighted in yellow to indicate that it has been overridden from the value in the template, and the **Estimated Cost Per Part** decreases.
6. Under **Estimated Cost Per Part**, click **Set Baseline**  to unlock the previously set baseline value.
7. Click **Set Baseline**  again to lock the **Current** price as the new baseline against which you compare future design changes.

Changing Model Geometry

You can change the model geometry to see how it affects manufacturing costs.

1. In the graphics area, right-click the hem and click **Suppress** .

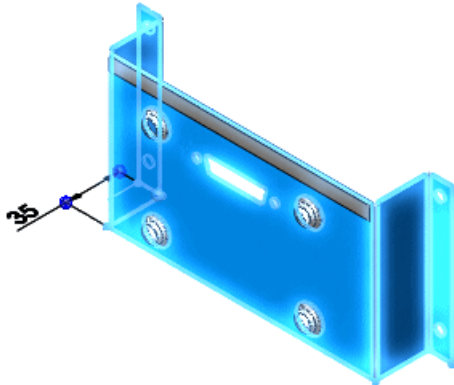




2. In the Sheet Metal Costing Task Pane, under **Estimated Cost Per Part**, click where it says **Click To Update**.

You need to manually update the cost estimate because you changed the model geometry.

The **Estimated Cost Per Part** decreases because you simplified the part.


3. In the graphics area, double-click the part.
4. Double-click the flange depth, **35**.



5. In the Modify dialog box, type 100 and click .
6. Click **Rebuild**  (Standard toolbar) or **Edit > Rebuild**.
The **Estimated Cost Per Part** increases.

Creating Reports

You can create reports of Costing results. The detailed line items in the report make it easy to see the affect of design decisions on cost.

At the bottom of the Sheet Metal Costing Task Pane, click **Generate Report** .

The report opens in Microsoft Word.

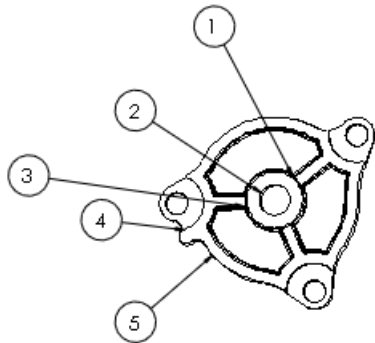
Drawings and Detailing

This chapter includes the following topics:

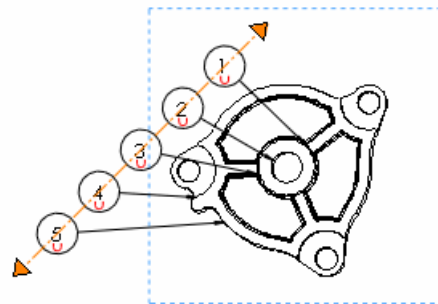
- **Align Balloons with Magnetic Lines**
- **Automatically Insert Center Marks**
- **Bend Tables**
- **Highlighting Changed Dimensions**
- **Customize Section View Label Fonts**
- **Display Hole Wizard Hole Sizes Using Hole Type**
- **Exploded Drawing Views**
- **Improved GB Standard Templates**
- **Initiate Bill of Materials Tool Without Preselecting a View**
- **Open Assemblies and Subassemblies from Drawing Views**
- **Ordering Balloons**
- **Punch Tables**
- **Reuse Letters from Deleted Views**
- **Set Section View Arrows**
- **Stacked Balloons**

Align Balloons with Magnetic Lines ✪

You can use magnetic lines to align balloons. You attach balloons to magnetic lines, choose to space the balloons equally or not, and move the lines freely at any angle. The balloons maintain their alignment when the magnetic lines are not visible. You can insert magnetic lines automatically when you use the **Auto Balloon** command. You can also insert balloons before or after inserting magnetic lines.




Without Magnetic Line



With Magnetic Line

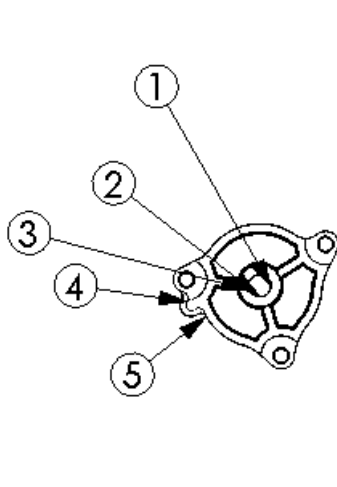
Inserting Magnetic Lines

To insert magnetic lines:

1. Open `install_dir\samples\whatsnew\drawings\cover1.slddrw`.
2. Click **Magnetic Line**  (Annotation toolbar) or **Insert > Annotations > Magnetic Line**.
3. Click to place the starting point of the magnetic line at the lower left of the drawing view.
4. Drag the other end point of the magnetic line straight up so that it is similar to the drawing below, and click to place the end point.




The line snaps to horizontal, vertical, and 45° angle positions, but can be located at any angle.

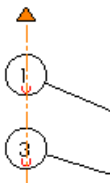


5. In the Magnetic Line PropertyManager, click .

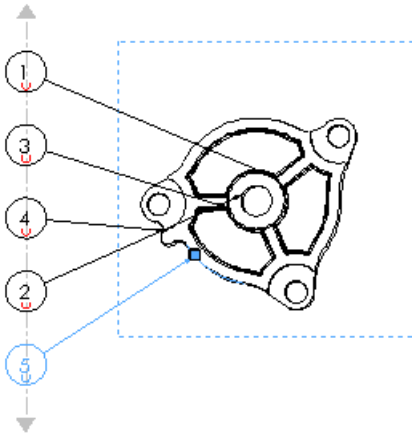
Attaching Balloons to Magnetic Lines

To attach balloons to magnetic lines, you can:

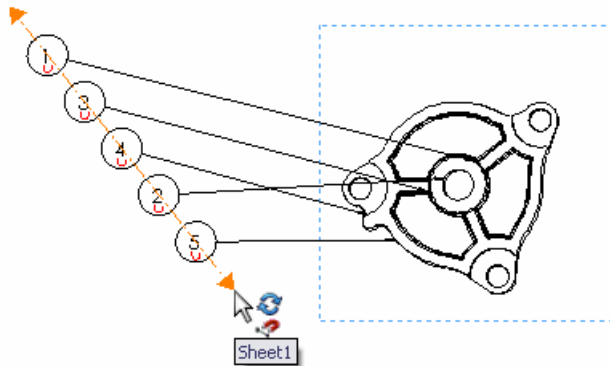
- Drag the balloon to the line. The cursor changes to a magnet symbol  when the balloon snaps to the line. Balloons that are snapped to a magnetic line display with a magnet symbol when the magnetic line is visible.



- Click one end of the magnetic line or drag the entire magnetic line from its middle to touch the balloons with the magnetic line. The balloons snap to the line.




1. Attach the balloons to the magnetic line using any combination of these methods.
2. Click ✓.
3. To move balloons attached to a magnetic line, click an end of the magnetic line or click anywhere on the line and drag it to the new location.



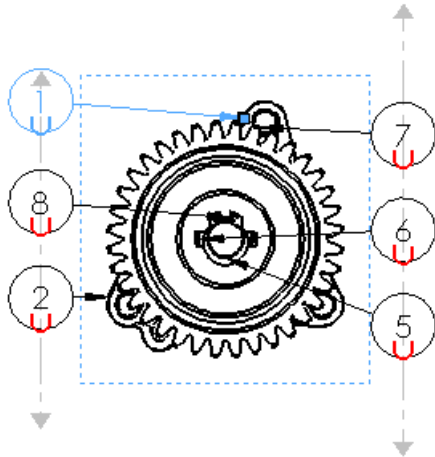
If magnetic lines are not visible, select any balloon to display the magnetic lines.

Inserting Magnetic Lines Automatically

You can insert magnetic lines automatically when using the **Auto Balloon** command.

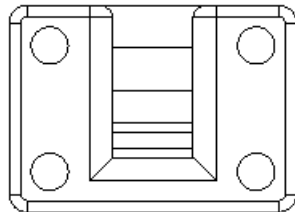
1. Open `install_dir\samples\whatsnew\drawings\oil_pump1.slddrw`.
2. Click **Auto Balloon**  (Annotation toolbar) or **Insert > Annotations > Auto Balloon**.
3. In the PropertyManager, under **Balloon Layout**, select **Insert magnetic line(s)**.

4. Set the options.
5. Click ✓.
6. Drag the balloons to the magnetic line.
7. Click ✓.

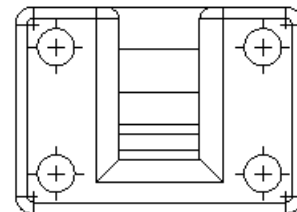


Automatically Insert Center Marks

You can automatically insert center marks for all holes, fillets, and slots in drawing views. Linear and circular patterns are recognized.




No Center Marks




Center Marks Inserted Automatically

Inserting Center Marks Automatically



To automatically insert center marks:

1. In a drawing, click **Center Mark**  (Annotation toolbar) or right-click and click **Annotations > Center Mark**.
2. In the PropertyManager, under **Auto Insert**:
 - a) Select one or more of the following:
 - **For all holes**

- **For all fillets**
 - **For all slots**
- b) Set the options.
 3. Select one or more drawing views.
 4. Click  to insert the center marks.

Bend Tables

There is a new bend table in SolidWorks 2012. You can insert the new bend table in the flat pattern view of sheet metal parts in drawings. Bend tables provide information about the bends in the sheet metal body including tags, directions, angles, and inner radii. You can insert bend tables or bend notes, but not both.


1. In a flat pattern view, click **Bend Table**  (Table toolbar) or **Insert > Tables > Bend Table**.
2. In the PropertyManager, set the options.
3. Click .
4. Move the cursor to a location in the drawing and click. When the bend table is displayed, the bend notes are not displayed.



- The bend table is not editable.
- You can hide columns and rearrange columns by dragging and dropping, or by clicking above a column and selecting the **Column type**.
- You can swap bends to change the bend order by dragging a bend number to another bend number in the flat pattern view.

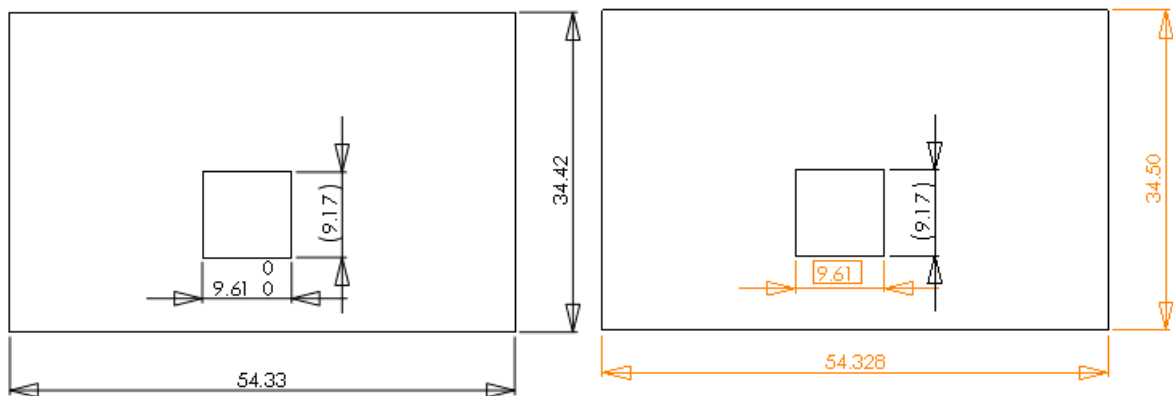
Setting Bend Table Properties

To set bend table properties:

1. In a drawing, click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the Document Properties tab, expand **Tables** and click **Bend**.
3. Set the options.
4. Click **OK**.

Highlighting Changed Dimensions

When opening a drawing, if dimensions have changed since the last time the drawing was saved, the changed dimensions are highlighted. To see changed dimensions, you must first save the part or assembly and the drawing in SolidWorks 2012. After you save and close the drawing with the changed dimensions, the changed dimensions highlighting is reset.




Before Changes

After Changes


Turning On Highlighting

To turn on highlighting for changed dimensions:

1. Click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the System Options tab, click **Colors**.
3. Select **Use specified color for changed drawing dimensions on open**.
4. Click **OK**.

Specifying Highlighting Color

To specify the color for highlighting changed dimensions:

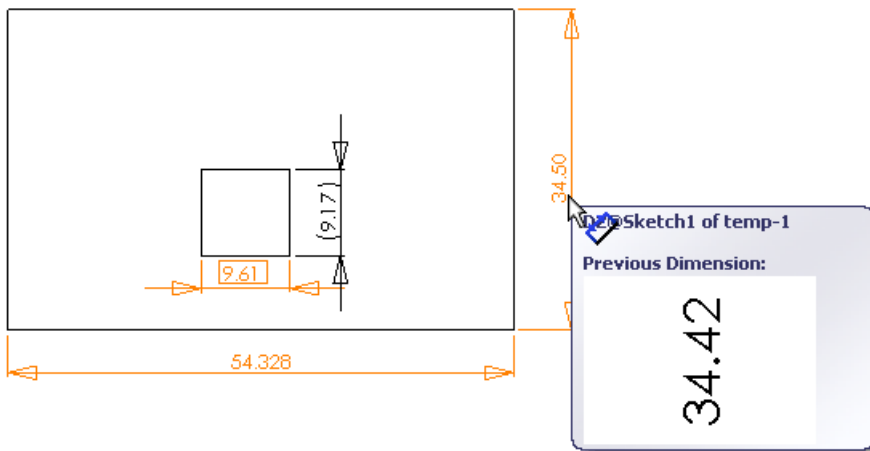
1. Click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the System Options tab, click **Colors**.
3. Under **Color scheme settings**, select **Drawings, Changed dimensions**.
4. Click **Edit**.
5. In the Color dialog box, select the color and click **OK**.
6. Click **OK**.

Changes that Are Highlighted

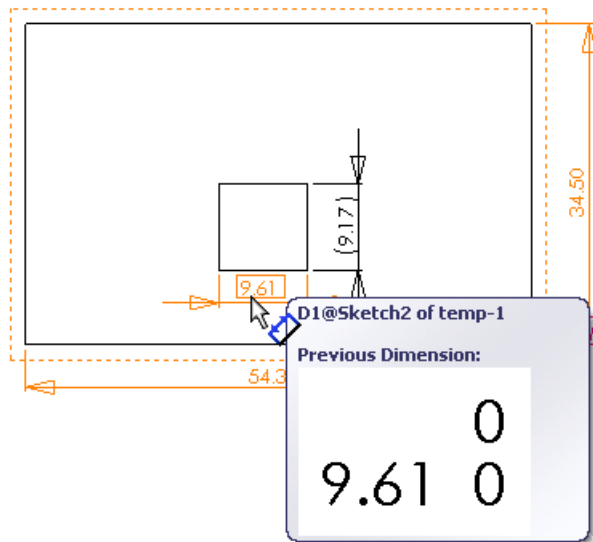
With highlighting turned on, these changes are highlighted in drawings:

- Model changes that cause base number values to change
- Imported dimensions where the tolerance type or value has changed in the model
- Imported dimensions where there is a change in **Add Parentheses** or **Inspection Dimension**
- Imported dimensions where there is a change in the **Dimension Text** (<DIM> by default)

To use this highlighting functionality, you must first save the part or assembly and associated drawings in SolidWorks 2012. When you reopen the drawing, the highlighted changes appear. If you hover over a changed dimension, a tooltip shows the previous dimension.



Changed Dimensions




Changed Tolerance


When you save the updated drawing, the highlighting functionality is reset.


Isolating Changed Dimensions

To isolate all changed dimensions:

Click **Isolate Changed Dimensions**  (Dimensions/Relations toolbar) or **View > Isolate Changed Dimensions**.

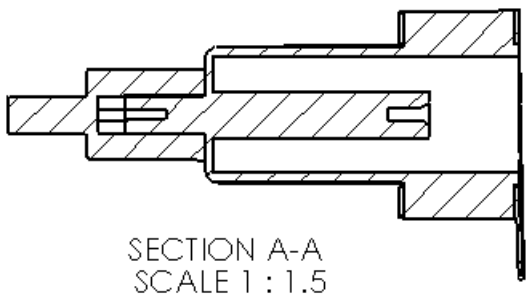
The changed dimensions highlight, all other dimensions turn gray, and the cursor

changes to . If you hover over a changed dimension, the tooltip shows the previous dimension.

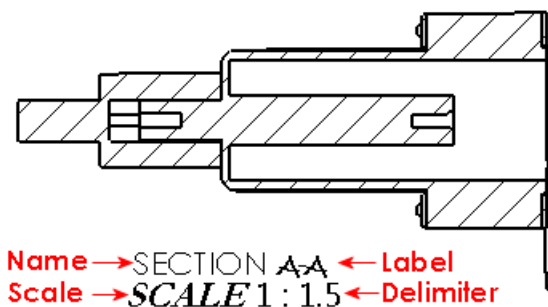
 You can use **Isolate Changed Dimensions** even if the **Use specified color for changed drawing dimensions on open** option is cleared. **Isolate Changed Dimensions** allows you to view changed dimensions *on demand* rather than automatically when you open the drawing, giving you the flexibility to see the changed dimensions only when you want to see them.

Customize Section View Label Fonts

In a drawing's document properties, you can set the font for each of the four areas of a section view label, including name, label, scale, and delimiter.




Previous Releases



SolidWorks 2012

Setting Section View Label Fonts

To set the section view label fonts:

1. In a drawing, click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the Document Properties tab, expand **View Labels** and click **Section**.
3. Under **Label options**, click **Font** for each area, select the font and parameters in the Choose Font dialog box, then click **OK**.
4. Click **OK** to close the Document Properties - Section dialog box.

Use the Options dialog box to set the section view label fonts so that the fonts are used in newly created section views. You can override the font of any specific part of the label by double-clicking the label section and selecting a font and parameters from the Formatting toolbar.

Display Hole Wizard Hole Sizes Using Hole Type

You can display ANSI inch hole sizes in hole tables using letters or numbers (for example, A or #40). This option displays holes created with the Hole Wizard tool with the same sizes as the **Hole Specifications** in the Hole Wizard tool.

TAG	X LOC	Y LOC	SIZE
A1	7.52	51.72	#45 ∇ 10
A2	19.84	51.72	#45 ∇ 10
B1	40.41	51.84	A ∇ 2.750
C1	47.02	12.39	W THRU ALL
D1	55.77	52.19	3/32 ∇ 5



Using hole type

TAG	X LOC	Y LOC	SIZE
A1	7.52	51.72	\varnothing 2.083 ∇ 10
A2	19.84	51.72	\varnothing 2.083 ∇ 10
B1	40.41	51.84	\varnothing 5.944 ∇ 2.750
C1	47.02	12.39	\varnothing 9.804 THRU ALL
D1	55.77	52.19	\varnothing 2.381 ∇ 5

Not using hole type


Displaying Sizes with Letters and Numbers

To display the ANSI inch sizes as they appear in the Hole Wizard tool:

1. In a drawing, click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the Document Properties tab, expand **Tables** and click **Hole**.
3. Under **Scheme**, select **Show ANSI inch letter and number drill sizes**.
4. Click .

Switching the Display

You can switch the display of sizes in a hole table.

1. Select the hole table.
2. In the Hole Table PropertyManager, under **Scheme**, select or clear **Show ANSI inch letter and number drill sizes**.
3. Click .

Exploded Drawing Views

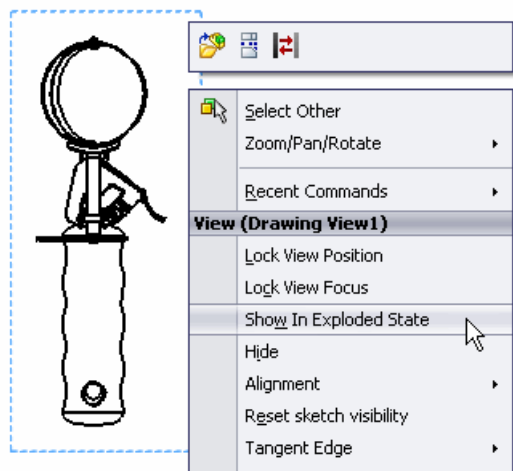
For assemblies and multibody parts that contain an exploded view, the View Palette now automatically contains an **Isometric Exploded** view.



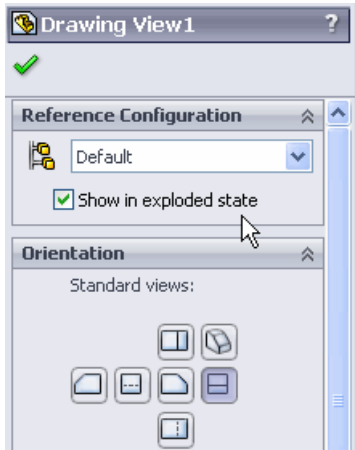
View Palette with Isometric Exploded View

To conveniently show any drawing view in an exploded state:

- In a drawing view, right-click and select **Show in Exploded State**.



- Alternatively, in the Drawing View PropertyManager, under **Reference Configuration**, select **Show in exploded state**.



Exploded Drawing View



Improved GB Standard Templates

The GB standard templates have been updated. New templates are available for parts, assemblies, and drawings.


Initiate Bill of Materials Tool Without Preselecting a View

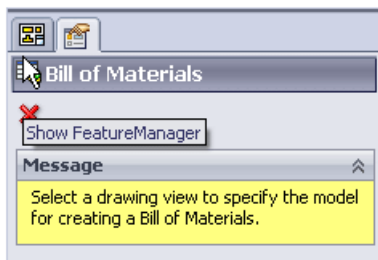
You can insert a bill of materials (BOM) on a different sheet than the sheet where the BOM is defined.

To switch drawing sheets while the BOM is attached to the pointer:

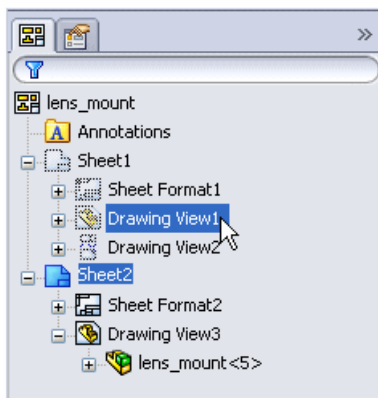
1. In a drawing with multiple sheets, click **Bill of Materials**  (Table toolbar) or **Insert > Tables > Bill of Materials**.
2. Click in a drawing view to select it.
3. In the Bill of Materials PropertyManager, set the options.
4. Click .
5. With the pointer attached to the BOM, at the bottom of the graphics area, click a sheet tab.
6. Move the pointer to locate the bill of materials and click.


To create a bill of materials from a drawing view on an inactive drawing sheet:

1. In a drawing with multiple sheets, click **Bill of Materials**  (Table toolbar) or **Insert > Tables > Bill of Materials**.
2. Select the FeatureManager design tree tab to activate it.



3. Select an inactive view from the FeatureManager design tree.



4. In the Bill of Materials PropertyManager, set the options.
5. Click .
6. With the pointer attached to the BOM, at the bottom of the graphics area, click a sheet tab.
7. Move the pointer to locate the bill of materials and click.

Open Assemblies and Subassemblies from Drawing Views

In addition to opening parts from the context toolbar in drawing views, you can open top-level assemblies or subassemblies directly from the context toolbar, if the parts are

contained in subassemblies. If the parts are only contained in assemblies, you can open the parts or assemblies.

Activating the Option







To activate the option to open parts, assemblies, and subassemblies from the context toolbar:

1. Click **Tools > Customize**.
2. On the Toolbars tab, under **Context toolbar settings**, select **Show in shortcut menu**.
3. Click **OK**.

Opening the Component From the Context Toolbar

To open parts, assemblies, or subassemblies from the context toolbar:

In a drawing view, click on a component.

Component Type	Icons in the Context Toolbar
Part	Open Part 
Assembly	Open Assembly  and Open Part 
Subassembly	Open Assembly  , Open Subassembly  , and Open Part 

Opening the Component From the Shortcut Menu

To open parts, assemblies, or subassemblies from the shortcut menu:

In a drawing view, right-click on a component.

Component Type	Action
Part	Under Component , select Open Part .
Assembly	Under Component , select Open Part , or under View , select Open Assembly .
Subassembly	Under Component , select Open Part , or under View , select Open Assembly or Open Subassembly .

Ordering Balloons

In addition to creating balloons, stacked balloons, and auto balloons following the assembly order, you can create balloons and number them sequentially. For balloons, auto balloons,

or stacked balloons, you can also control the direction of the ordering (for example, left to right or right to left for a linear arrangement), as well as which balloon is designated as the first balloon.

To order balloons sequentially:



- A bill of materials must exist in the drawing. Ordering balloons sequentially reorders the bill of materials.
- You must link the balloons to the bill of materials. When you create a new bill of materials, the balloons are automatically linked to it. In cases where they are not linked, you can link them by right-clicking in the drawing view and clicking **Properties**. In the Drawing View Properties dialog box, on the **View Properties** tab, under **Balloons**, select **Link balloon text to specified table**, and click **OK**.
- You must add the balloons to the drawing. Existing balloons cannot be reordered.

In this example, you:

- Insert a bill of materials.
- Use Auto Balloons to insert balloons following the assembly order.
- Reorder the balloons sequentially.
- Designate a balloon as the first balloon.




Inserting a Bill of Materials

To insert a bill of materials:

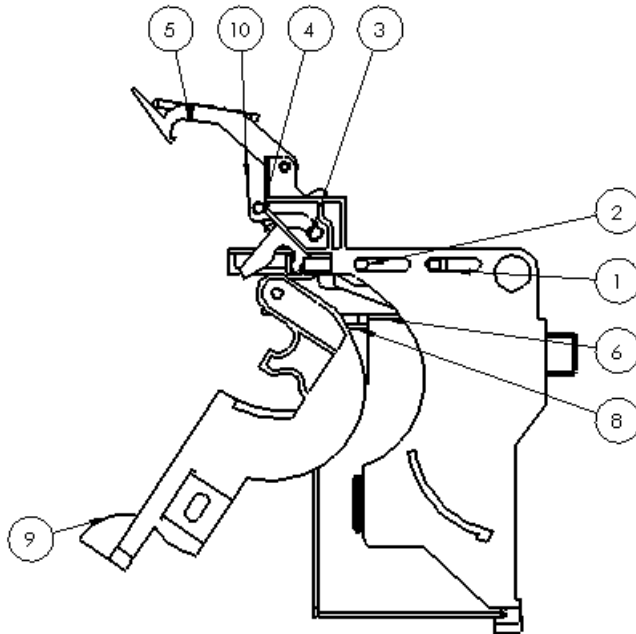
1. Open `install_dir\samples\whatsnew\drawings\Ordering_balloons.slddrw`.
2. In the drawing view, click **Bill of Materials**  (Table toolbar) or **Insert > Tables > Bill of Materials**.
3. In the PropertyManager, under **BOM Type**, select **Parts only**.
4. Click .
5. Move the pointer to the lower left of the drawing and click to place the bill of materials.

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	FB10172001.01	Ink Folder Carriage	1
2	IF10172001.04	Ink Cartridge Extractor	1
3	IF10172001.01	Ink Cartridge Latch	1
4	IF10172001.02	Ink Cartridge Link Support	1
5	IF10172001.05	Ink Cartridge Lock Down	1
6	IT10172001.01	Ink Floder	1
7	IT10172001.02	Ink Cartridge Motherboard	1
8	inkcartridge	Ink Cartridge	1
9	IT10172001.03	Ink Cartridge Lid	1
10	link101 finished	Ink Cartridge Link	1

Inserting Balloons Following Assembly Order

1. In a drawing view of the assembly, click **AutoBalloon**  (Annotation toolbar) or **Insert > Annotations > Auto Balloon**.
2. In the PropertyManager, under **Item Numbers**, select **Follow assembly order** .
3. Click .




The balloons appear as shown:



The numbered sequence of the balloons follows the order in the assembly FeatureManager.

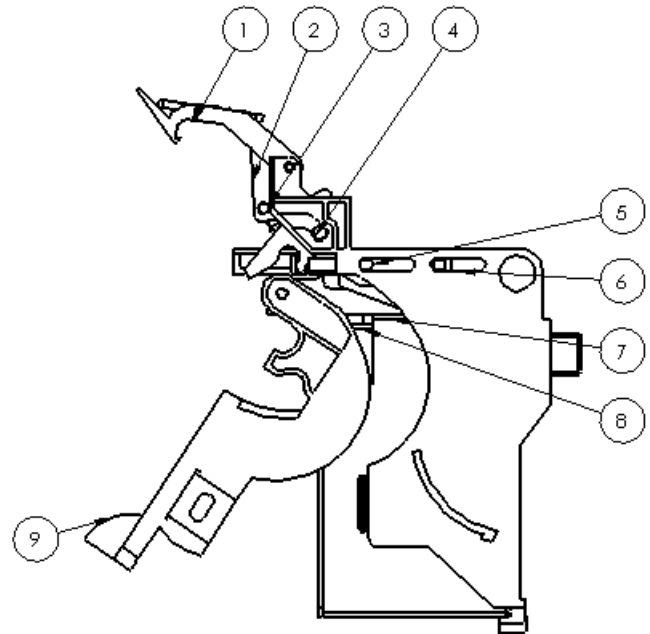
Numbering the Balloons Sequentially

To insert the balloons and renumber them sequentially starting with 1:

1. Open `install_dir\samples\whatsnew\drawings\Ordering_balloons.slddrw`.
2. If necessary, insert the bill of materials. See [Inserting a Bill of Materials](#) on page 68.
3. In the drawing view of the assembly, click **AutoBalloon**  (Annotation toolbar) or **Insert > Annotations > Auto Balloon**.
4. In the PropertyManager, under **Item Numbers**, select **Order sequentially** .
5. Click .


The balloons are reordered so that the top left balloon starts with 1 and continues sequentially, in a clockwise direction, through 9. The bill of materials updates to match the reordered sequence.

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	IF10172001.05	Ink Cartridge Lock Down	1
2	link101 finished	Ink Cartridge Link	1
3	IF10172001.02	Ink Cartridge Link Support	1
4	IF10172001.01	Ink Cartridge Latch	1
5	IF10172001.04	Ink Cartridge Extractor	1
6	FB10172001.01	Ink Folder Carriage	1
7	IT10172001.01	Ink Floder	1
8	inkcartridge	Ink Cartridge	1
9	IT10172001.03	Ink Cartridge Lid	1
10	IT10172001.02	Ink Cartridge Motherboard	1

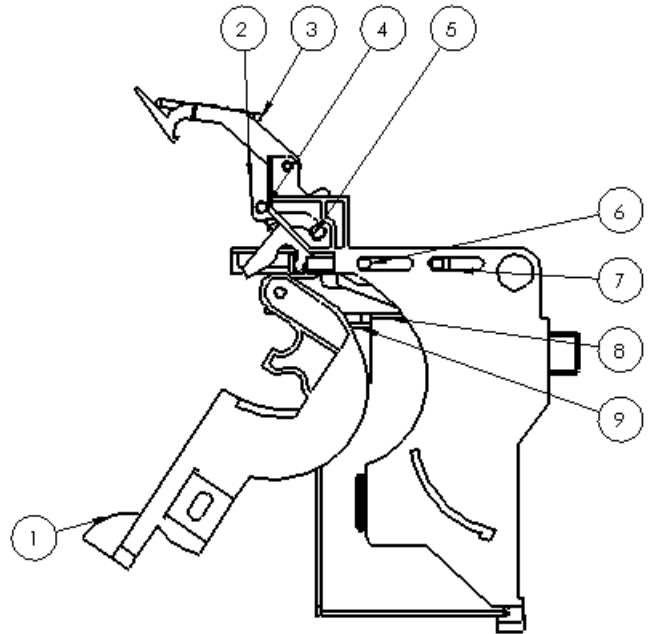


Setting the First Balloon

The balloons are numbered sequentially, starting with 1 at the top left of the drawing. To start the numbering at the bottom left of the drawing with 1 and continue clockwise:

1. Open `install_dir\samples\whatsnew\drawings\Ordering_balloons.slddrw`.
2. If necessary, insert the bill of materials. See [Inserting a Bill of Materials](#) on page 68.
3. In the drawing view of the assembly, click **Insert > Annotations > Auto Balloon**.
4. In the PropertyManager, under **Item Numbers**, select **Order sequentially**.
5. Under **Item Numbers**, click **First item**.
6. Click the bottom left balloon to use as the first item.
7. Click .

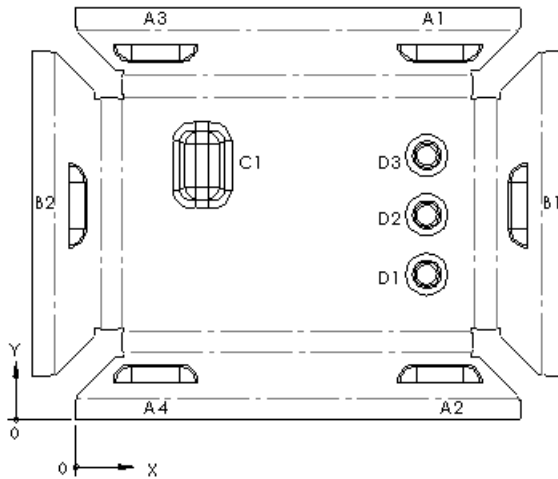
ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	IT10172001.03	Ink Cartridge Lid	1
2	link101 finished	Ink Cartridge Link	1
3	IF10172001.05	Ink Cartridge Lock Down	1
4	IF10172001.02	Ink Cartridge Link Support	1
5	IF10172001.01	Ink Cartridge Latch	1
6	IF10172001.04	Ink Cartridge Extractor	1
7	FB10172001.01	Ink Folder Carriage	1
8	IT10172001.01	Ink Floder	1
9	ink cartridge	Ink Cartridge	1
10	IT10172001.02	Ink Cartridge Motherboard	1



The item numbers in the bill of materials update with the new balloon ordering.

Punch Tables

You can insert punch tables in drawings. Punch tables are available with the flat pattern view of sheet metal parts and contain the form features and library features used in sheet metal bodies.



TAG	X LOCATION	Y LOCATION	PUNCH ID	QUANTITY	ANGLE
A1	153.46	146.44	LU-50-x10	1	90°
A2	153.46	26.66	LU-50-x10	1	270°
A3	33.83	146.44	LU-50-x10	1	90°
A4	33.83	26.66	LU-50-x10	1	270°
B1	178.81	89.85	LU-50x10	1	0°
B2	8.48	89.85	LU-50x10	1	180°
C1	53.51	106.73	RE-20x30	1	0°
D1	147.94	60.89	RO-20	1	0°
D2	147.94	85.89	RO-20	1	0°
D3	147.94	110.89	RO-20	1	0°

A punch table includes:

Tag	Annotation added to each punch feature in the flat pattern
X Location	Distance from X-axis to the point of insertion of the tool
Y Location	Distance from Y-axis to the point of insertion of the tool
Punch ID	Property of the forming tool/library feature used in the component
Quantity	Number of times form/library feature is used in the flat pattern
Angle	Angle between X-axis and the tool

To learn how to create Punch IDs, see [Creating Punch IDs](#) on page 142.


For information on inserting a Punch Table, see [Inserting a Punch Table](#) on page 142.

Reuse Letters from Deleted Views

If you delete a derived view (section, detail, or auxiliary) from a drawing, you can reuse the letters from the deleted view in the drawing without manually relettering the views. This option is off by default.



In previous releases, if you deleted a derived view, such as a section view, and then created a new section view, the lettering would use the next sequential letter. For example, if you deleted **Section A-A** from a drawing view and then created a new section view, the new label would be **Section B-B**. You can now reuse the letters from deleted views automatically, so the newly created section would be **Section A-A**.

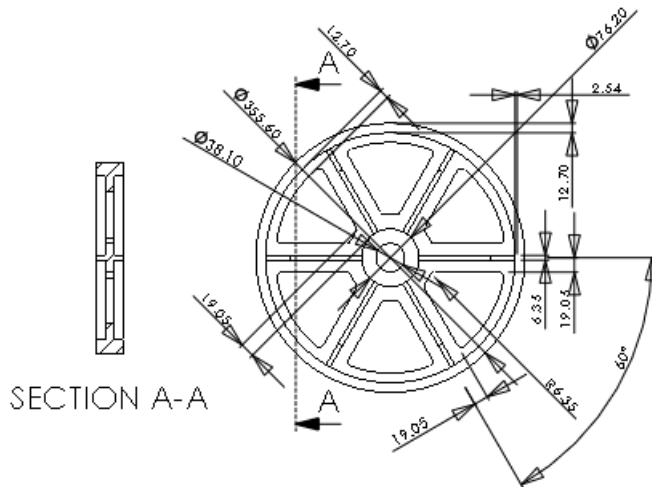
Reusing Letters from Deleted Views

1. In a drawing, click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the System Options tab, click **Drawings**.
3. Select **Reuse view letters from deleted auxiliary, detail, and section views**.
4. Click **OK**.

Set Section View Arrows


You can set the arrow type for section views independently of the arrow type for dimensions. Previously, section views used the same arrow type as dimensions.

In this example, the dimensions use  arrows and the section view uses  arrows.



Setting Arrow Types

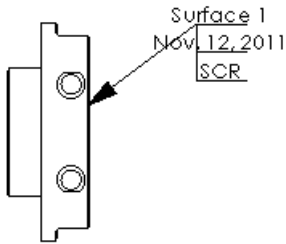
To set the arrow type for section views:

1. In a drawing, click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the Document Properties tab, expand **View Labels** and click **Section**.
3. Under **Section/view size**, select the style.
4. Click **OK**.

Stacked Balloons

The stacked balloon functionality for horizontal lines is improved. You have more control over the appearance of stacked balloons including automatic and manual extension of horizontal lines to accommodate text, and horizontal text justification. Also, when you drag a stacked balloon annotation, the position of the stack flips to the other side when necessary.

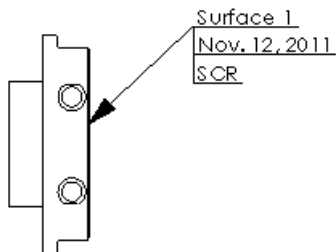
In previous releases, if you created stacked balloons with horizontal lines, they could appear as shown:



You can now:





- Manually control each line length by dragging.
- Align the notes.
- Move the stack and have the items change location as necessary so that the leader does not interfere with the stack.

After creating the stacked balloons, you can now adjust the notes:

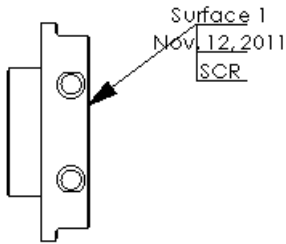


Creating Stacked Balloons

To create stacked balloons:

1. In a drawing view, click **Stacked Balloon**  (Annotation toolbar) or **Insert > Annotations > Stacked Balloon**.
2. In the PropertyManager, under **Balloon Settings**, select:
 - **Underline** for **Style**
 - **User Defined** for **Size**
 - **Stack Down**  or **Stack Up** 
3. In the drawing view, click to place the leader.
4. Move the pointer and click to place the first balloon.
5. Continue to place additional balloons by clicking in the drawing view.
6. In the Stacked Balloon PropertyManager, click .

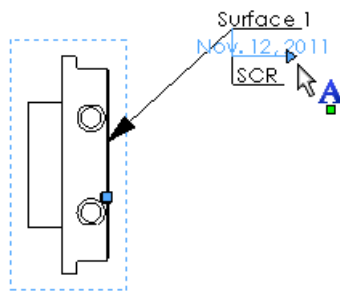
The balloons appear as:



The stacked balloons look similar to previous releases, except you now have more control over the line length and alignment when you edit the balloons.

Changing Line Lengths

To change a line length, click a balloon and drag the arrow.



Aligning Stacked Balloons

To align stacked balloons:

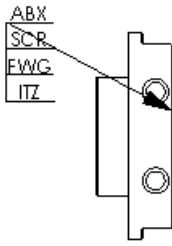
1. Select the balloons in the stack to align.
2. In the Stacked Balloon PropertyManager, click **More Properties**.
3. In the Note PropertyManager, under **Text Format**, click one of the following:

- **Left Align**
- **Center Align**
- **Right Align**

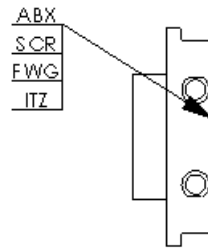
4. Click

Dragging Stacked Balloons

If you drag a stacked balloon annotation from one side of a drawing view to the other, the stack flips so that the leader does not interfere with the stack.



Previous releases



SolidWorks 2012

eDrawings Professional is available in SolidWorks Professional and SolidWorks Premium.

This chapter includes the following topics:


- **Bills of Materials**
- **File References**

Bills of Materials

When you open a file in SolidWorks eDrawings[®] that has a hidden bill of materials (BOM), the BOM remains hidden.

File References

When you open a SolidWorks document, eDrawings checks the document's file references to see if the references have been updated. This helps you determine if the document is out of date.

If eDrawings cannot determine whether file references are up to date, or if it confirms that the file references are not up to date, a watermark is displayed. You can click the warning icon  in the bottom of the eDrawings window and click any of the following:

- **Check if File is Out of Date**
- **Always Show Warning Watermark**
- **Show Warning Watermark for this document only**

To use the file reference checking for standalone eDrawings installations, you must download and install the SolidWorks Document Manager when prompted.



The Mac OS[®] version of eDrawings cannot check file references.

SolidWorks provides a completely redesigned dialog box for creating and editing equations, global variables and dimensions. This release contains a host of new options that improve the workflow and efficiency for building and managing equations. Now you can perform all tasks related to equations in one dialog box without needing to open additional windows. You can also create equations and global variables directly in the Modify dialog box for dimensions.

Enhancements to functionality include:

- automatic determination of the proper solve order of equations, to avoid circular references and reduce equation troubleshooting.
- automatic solving of cases that previously required multiple rebuilds.
- an automatic rebuild option, to display graphically the impact of each change to equations.
- the ability to create permanent links to external text files, to make these files easily accessible.
- support for instance numbers, to provide unambiguous references to components in an assembly.
- a new measure option to speed up the process of creating reference dimensions.
- and new syntax using `if` statements and the ability to use the `=`, `<=`, and `=>` conditional operators.

Other enhancements include type-ahead capabilities and drop-down lists to speed up entering equations, and automatic syntax checking to increase accuracy.


This chapter includes the following topics:

- **Accessing the Equations Dialog Box**
- **Three Views of Equations**
- **Interface Enhancements**
- **Functionality Enhancements**
- **Create and Edit Equations in the Equations Dialog Box**
- **Export, Import, and Link Equations to a File**
- **Create Equations in the Modify Dialog Box**
- **Equations and Configurations**
- **Use Equations**

Accessing the Equations Dialog Box

Display the Equations, Global Variables and Dimensions dialog box by doing one of the following:




- Click **Equations**  (Tools toolbar).
- Click **Tools > Equations**.

- Right-click the **Equations**  folder in the **FeatureManager** design tree, and select **Manage Equations**.

Three Views of Equations

The Equations, Global Variables and Dimensions dialog box offers three views. Each view shows a different combination and sequence of equations, global variables and dimensions to help you perform tasks such as finding a specific equation, viewing all dimensions used in a part, and changing the order in which equations are solved.

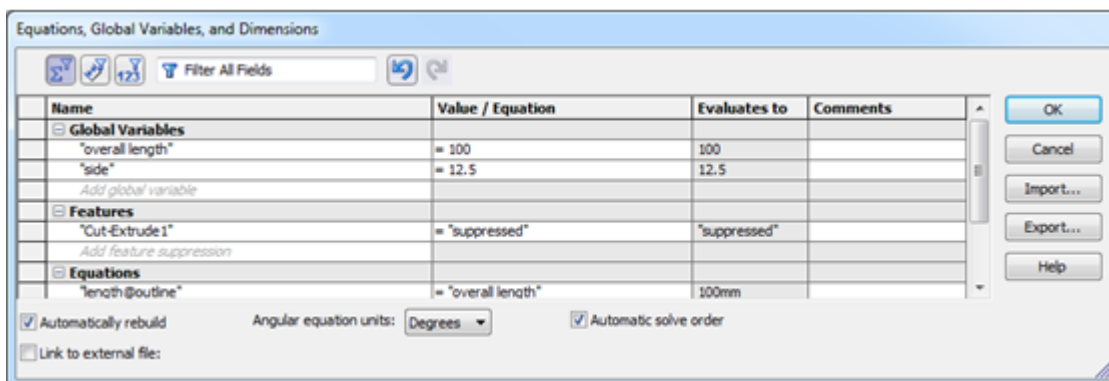
To select a view:

- Click the Equations View  to see all global variables and equations for dimensions and features.
- Click the Dimensions View  to see all global variables, equations, and dimensions used in the active part or assembly, whether they are associated with an equation or not.
- Click the Ordered View  to see global variables and equations in the order they are solved, and to see suppressed equations.

Equations View

The Equations View displays all global variables, equations to suppress features, and other equations for the part or assembly, grouped in those three categories.

In this view you can add new global variables and equations. You can also edit, delete and suppress existing ones, and add comments.



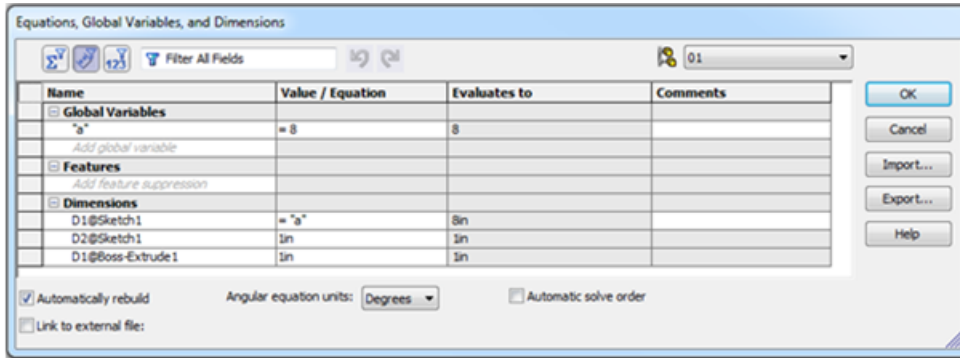
Dimensions View

The Dimensions View displays all the dimensions used in the active part or assembly, including those with a set value and those determined by equations. This view makes it easy to rename or change the values of several dimensions.

Dimensions with a set value are visible only in this view, not in the Equations View or the Ordered View.

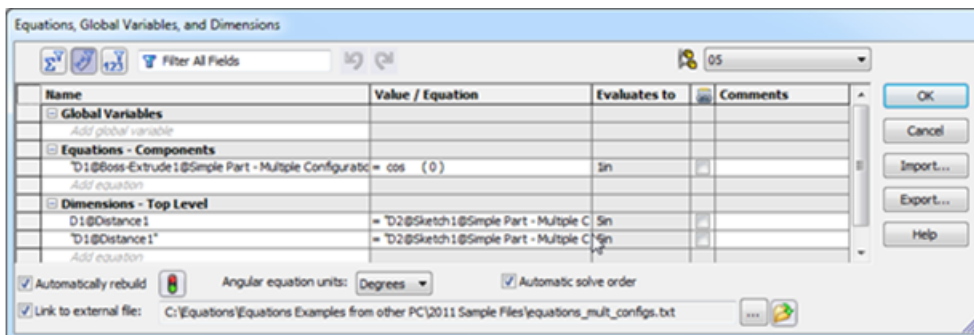
There are two versions of the Dimensions View, one for parts, and one for assemblies.

In the Dimensions View for a part, you can:



- Add new global variables, and rename, edit, delete, and suppress existing ones.
- Add equations to suppress features.
- Rename and change dimension values or add equations to dimensions.
- Add comments to global variables and features, and to dimensions that have equations.

In the Dimensions View for an assembly, you can:



- Add new global variables, and rename, edit, delete, and suppress existing ones.
- Add equations for components, and rename, edit, delete, and suppress existing ones.
- Rename and edit top-level dimensions of the assembly.
- Add comments to global variables and equations for components, and to top-level dimensions that have equations.

When you suppress an equation assigned to a dimension, the dimension uses the last evaluated value. When you unsuppress the equation, the dimension uses the equation instead of the value.

- ★ In the **Name** column, names of global variables, features, and equations are enclosed in quotation marks, while the names of dimensions are not. In the **Value/Equation** column, dimensions determined by equations start with = (equal sign), while dimensions with a set value do not start with =.

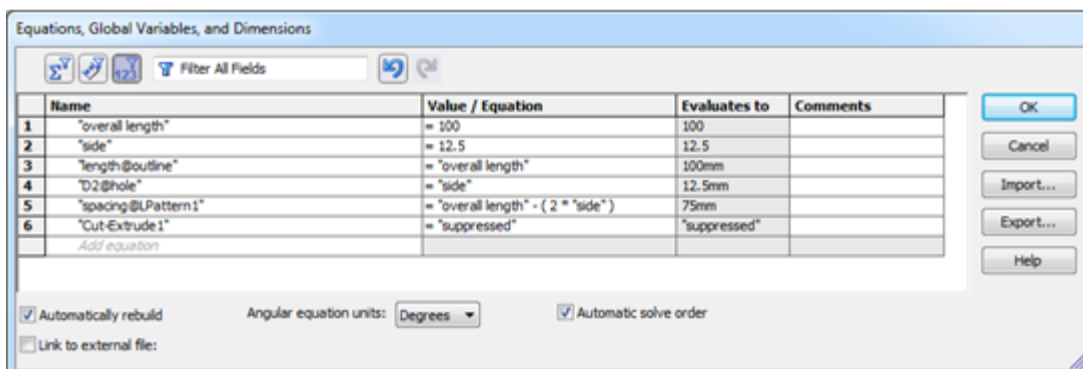
Ordered View

The Ordered View displays equations and global variables in the order they are solved. It is also the only view where you can view suppressed equations and unsuppress them.

In this view you can:

- Add new equations and global variables, and rename, edit, and delete existing ones.
- Unsuppress and suppress equations and global variables.

- Change the order of the equations and global variables (if the **Automatic solve order** option is clear).



Interface Enhancements

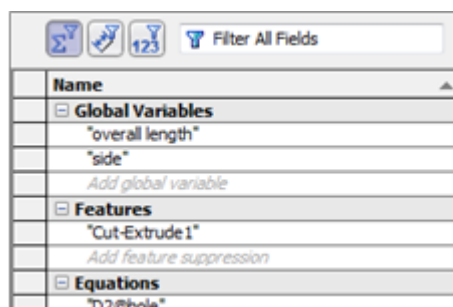
The following enhancements are available from all views in the Equations, Global Variables and Dimensions dialog box.

Sort and Filter Equations

New sorting and filtering capabilities make it much easier to find equations and global variables and see how they are related. Sorting and filtering work in all views of the Equations dialog box.

Sorting

1. Select any view in the Equations dialog box.
2. Click any column heading. A small arrow appears on the right side of the cell.



3. Click once to sort in ascending order.
4. Click a second time to sort in descending order.
5. Click a third time to return the rows to the original order.

Creating a Filter

1. Select any view in the Equations dialog box.
2. Enter your filter criteria in the **Filter** field and press **Enter**.

The filter returns results that contain characters in any column. For example, entering `ov` might return the global variable `overall length`, an equation that includes

overall length as one of its terms, and another equation that has a comment Don't overlook the ring.

3. Continue to enter criteria until you find the desired global variable, feature, or equation.
4. Clear the filter by clicking **X** in the **Filter** field.

Selecting Multiple Rows

You can select multiple rows in the Equations dialog box to perform the same task on all of the equations, for example to suppress, unsuppress, or delete several equations at once.

To select multiple rows, position the cursor in the left-most column of a row. A small arrow appears in the cell.



Do one of the following:

- Select several rows by dragging the cursor up or down in the left-most column.
- **Ctrl** + click the left-most column in one or more rows.
- **Shift** + click the left-most column in the last of a series of rows.

Then right-click to display a menu and select one of the available tasks.

Undo and Redo

You can undo and redo steps by clicking Undo and Redo at the top of the Equations dialog box.

Undo  and **Redo**  are available in all views. You can undo and redo any edits that you perform since invoking the dialog box.



If you change configurations while in the Equations dialog box, **Undo** and **Redo** lists are reset. Prior steps cannot be undone or redone.

Type-Ahead Entry

When you enter an equation in the Equations dialog box and type in one or more characters, a type-ahead feature displays a drop-down list showing all global variables, mathematical functions, and file properties that start with those characters. You can select an entry from the drop-down list instead of typing the entire entry.

Syntax Checking with Color Coding

The Equations dialog box includes syntax checking to avoid errors and reduce the time you spend on troubleshooting mistakes.

When you enter text into the **Name** and **Value/Equation** columns, the text color varies depending on these conditions:

- Valid global variables, features and dimensions appear in blue.
- Incomplete and invalid text displays in red.
- Potentially invalid text, for example text that may include a circular reference, is displayed in yellow. A yellow warning icon is also displayed in the row.
- Valid operators, functions, and other inputs appear in black.

Positioning the cursor over the yellow or red text displays a warning message. Clicking the warning icon displays a What's Wrong window describing the issue or error.

Navigate Table Cells

In the Equations dialog box, you can navigate from cell to cell, across rows, and columns using the following keys:

- **Enter**
- **Shift + Enter**
- **Tab**
- **Shift + Tab**

Functionality Enhancements

These sections describe several of the functionality enhancements to the Equations, Global Variables and Dimensions dialog box.

Automatic Solve Order Option

You can use the **Automatic Solve Order** option to automatically sequence the equations in an order determined by the software to produce accurate results.

When you select the Automatic Solve Order option, SolidWorks identifies dependencies and orders equations so that independent equations are executed before dependent equations. That is, if equation B is defined as a function of equation A, then equation A always needs to be solved first.

For example, in the following two equations, SolidWorks evaluates "D2@Sketch1@part_inside.Part" first, so it can safely be used as a variable in the equation for "D1@Sketch1@part_outside.Part".

```
"D1@Sketch1@part_outside.Part" = "D2@Sketch1@part_inside.Part" + 2
```

```
"D2@Sketch1@part_inside.Part" = "Side" - 6
```

You can view the solve order in the Ordered View  of the Equations dialog box.



You must clear this option to manually change the order of the equations

Automatic Rebuild Option

You can use the **Automatically rebuild** option to automatically rebuild the model each time you change an equation.

This option makes it easier to visualize the impact of each change you make to equations.

Linking to a Text File

In the Equations dialog box, you can link to an external text file that contains equations and global variables. You can use this option in place of the import and export functions

when you want to set up an ongoing link. However, you cannot use it for a one-time import or export.

Suppressing Equations

You can suppress equations and global variables from all views in the Equations dialog box. You can only view and unsuppress them in the Ordered View.

You can suppress equations and global variables to help you troubleshoot equations.

You can also suppress equations so that different equations can be applied to different configurations of a part or assembly.

You can edit suppressed equations and global variables in the Ordered View, but the changes do not take affect in the model until the equation or variable is unsuppressed.

Components with Multiple Instances

In this release the Equations dialog displays components that are referred by unique instance numbers. You can apply equations to specific instances. This ensures there is no ambiguity about the instance referenced in each equation.

For example, the following equations refer to the same part `BigAndSmall`, which has instances `<1>` and `<2>`. In the Equations dialog, you can select which instance to apply to an equation.

```
"HoleWidth@Sketch1" = "Width@Sketch1@BigAndSmall<2>.Part"
```

```
"HoleDepth@Cut-Extrude1" = "Depth@Boss-Extrude1@BigAndSmall<1>.Part" / 2.0
```

```
"HoleWidthOffset@Sketch1" = ("Width@Sketch1@BigAndSmall<1>.Part" / 2) -  
("Width@Sketch1@BigAndSmall<2>.Part" / 2)
```



The instance number is always enclosed in `< >` angle brackets.

In previous releases, the software did not display the instance number. Instead, it selected the latest instance and action that was available in memory.

The Measure Option

You can use the **Measure...** option to create a driven dimension or reference dimension that is based on a measurement of the part or assembly.



The value of this dimension might change if the model changes. For example, if an equation includes a measurement of the height of a part, and that height is doubled, then the equation evaluates to twice the previous value.

The **Measure** option is available from all three views of the Equations dialog box, and also from the Modify dialog box.




You can select the **Measure...** option from the flyout menu when you are in the **Value/Equation** column creating or editing global variables, features, equations, and dimensions. It appears in the flyout menu whenever a measurement is valid.

Create and Edit Equations in the Equations Dialog Box

You can use the Equations dialog box to create, edit, suppress, unsuppress and delete equations. Global variables and equations are available in one place, and flyout menus help you create global variables and equations more quickly and accurately.



Adding an Equation

To add an equation in the Equations View:

1. Do one of the following:
 - Click **Equations**  (Tools toolbar).
 - Click **Tools > Equations**.
 - Right-click the **Equations**  folder in the FeatureManager design tree, and select **Manage Equations**.
2. Select the Equations View .
3. In the **Equations** section, click an empty cell in the **Name** column.
4. Click a dimension in the graphics area.

The SolidWorks software does the following:

 - Propagates the dimension name to an empty cell in the **Name** column and encloses it in quotation marks.
 - Moves the cursor to the **Value/Equations** column and inserts = (equal sign).
 - Displays a flyout menu with options for starting the equation.
5. After = (equal sign), add a term to the equation by doing one of the following:
 - Type a number or a conditional statement.
 - Select a **Global Variable, Function** or **File Property** from the flyout menu.
 - Select **Measure ...** from the flyout menu and use the Measure Tool to create the term.




A  appears in the cell to indicate that the syntax is valid.
6. Type + (plus), - (minus) or another mathematical symbol.
7. Add another term to the equation.
8. When the equation is complete, click .

The solution to the equation appears in the **Evaluates to** column and the cursor moves to the next cell in the **Comments** column.
9. Add comments to document your design intent.
10. Click **OK** to close the dialog box.



Editing an Equation

When editing, keep the following in mind:

- Dimension names must be enclosed in quotes.




- Equations are solved left to right (that is, the dimension on the left is driven by the value on the right).
 - Equations are solved in the order in which they appear in the Ordered view. You can change the order, if necessary.
1. Do one of the following:
 - Click **Equations**  (Tools toolbar).
 - Click **Tools > Equations**.
 - Right-click the **Equations**  folder in the FeatureManager design tree, and select **Manage Equations**.
 2. Select the Equations View .
 3. Select an equation, global variable or dimension.
 4. Edit the entry.

Use the type-ahead entry, flyout menu of **Equations, Global Variables, Functions** and **File Properties**, and syntax checking to edit the equation.

 If you make a mistake, click **Undo**  to clear each consecutive edit. You can clear one edit each time you click **Undo**.
 5. Click **OK** to close the dialog box.


Suppressing an Equation

You can suppress equations and global variables to help you troubleshoot equations. You can also suppress equations so that different equations can be applied to different configurations of a part or assembly.

1. Do one of the following:
 - Click **Equations**  (Tools toolbar).
 - Click **Tools > Equations**.
 - Right-click the **Equations**  folder in the FeatureManager design tree, and select **Manage Equations**.
2. Select the Equations View .
3. Select the row containing the global variable or feature or equation you want to suppress and right-click.

A flyout menu displays the choice to suppress the item.
4. From the flyout menu, select **Suppress**. If you are working on a part or assembly with multiple configurations, you can suppress:
 - **This Configuration**
 - **All Configurations**
 - **Specify Configurations**



To unsuppress, select the Ordered View , select the row and right-click. From the flyout menu, select **Unsuppress**.

Deleting an Equation




To delete a global variable or an equation from any view:

1. Select one or more rows, containing the global variables or equations you want to delete and right-click.
2. From the flyout menu, select **Delete** or the **Delete** key.
3. Click **OK** to close the Equations dialog box.


Deleting a global variable or equation may cause other equations that include the global variable or equation to become invalid. Syntax checking identifies this situation immediately by displaying a warning symbol and displaying the deleted term in red anywhere it appears in the **Value/Equation** field. This can dramatically reduce the time needed to detect and troubleshoot orphaned equations.

Using the Measure Option

To use the **Measure** option when creating or editing an equation:

1. Do one of the following:
 - Click **Equations**  (Tools toolbar).
 - Click **Tools > Equations**.
 - Right-click the **Equations**  folder in the FeatureManager design tree, and select **Manage Equations**.
2. Select the Equations View .
3. In the **Equations** section, click an empty cell in the **Name** column.
4. Click a dimension in the graphics area.

The SolidWorks software does the following:



 - Propagates the dimension name to the empty cell in the **Name** column and encloses it in quotation marks.
 - Moves the cursor to the **Value/Equations** column and inserts = (equal sign).
 - Displays a flyout menu with options for starting the equation.
5. In the flyout menu, select **Measure....**
6. In the graphics area, select the references to create a reference dimension.
7. In the Dimension PropertyManager, make any changes to the parameters of the dimension and click .

The new dimension appears as part of the equation.

Changing Dimensions by Configuration

If a part has several configurations, you can decide whether changes to a dimension will affect one configuration, all configurations, or selected configurations.

To specify configurations affected by a change in dimensions:

1. Select the Dimensions View  in the Equations dialog box.
2. Edit the **Value/Equations** field for the dimension.
If the dimension applies to more than one configuration, a configuration icon  appears in the column.
3. From the flyout menu, select from the following choices:
 - **This Configuration**
 - **All Configurations**
 - **Specify Configurations**
4. Click **OK** to close the dialog box.

Export, Import, and Link Equations to a File

Enhancements in this release give you much more control over importing, exporting, and linking equations.

You can export all equations or selected equations to a text file, and import all equations or selected equations from a text file. The text file can be shared with other parts and assemblies that use the same global variables and equations.

You can create permanent links with text files using the **Link to external file**, so that changes to the text file are propagated to the model.

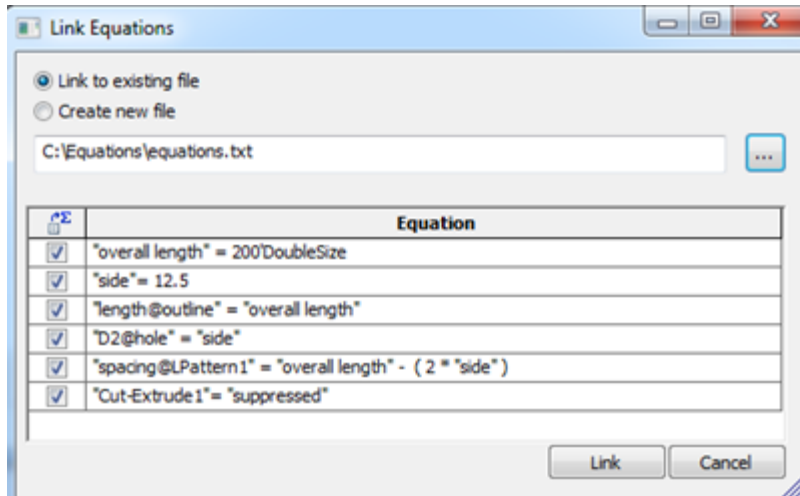
The **Link to external file** always creates a link with the external file. The **Export** and **Import** processes include options for one-time exports and imports without a link.

Linking to External Files

To export equations using **Link to external file** in the Equations dialog box:

1. In any of the views, select the **Link to external file** check box.
2. In the Link Equations dialog box:
 - a) Select **Create new file**.
 - b) Specify a name and location for the text file.
 - c) In the **Equations** column, clear the check mark for any equation you do not want to export.
 - d) Click **Link**.

The equations are saved in a text file. The text file is available to import into other parts and assemblies.



Then in the Equations dialog box:

- A **Link** column appears and indicates which equations are linked to the file.
- The **Link to external file** check box is selected and the path to the external file appears.

Subsequent changes to the text file will be propagated to the model.



Equations and global variables that are linked to an external file cannot be modified in the Equations dialog box. They must be edited in the external file. To remove the link and modify an individual equation in the dialog box, clear the check box for that equation in the **Link** column.

You can always access the external file by clicking **Open linked file** .

Exporting Equations

To export equations to a text file:

1. Define equations in the Equations dialog box.
2. Click **Export**.
3. In the Save As dialog box:
 - a) Specify a name and location for the text file.
 - b) Enter a description of the file.
 - c) Click **Save**.
4. In the Export Equations dialog box:
 - a) Select or clear **Link to file**.
 - b) In the **Equations** column, clear the check mark for any equations you do not want to export.
 - c) In the **Link** column, clear the check mark for any equations you do not want to link to the external file.
 - d) Click **Export**.

The equations are saved in a text file. The text file is available to import into other parts and assemblies.

If **Link to File** was selected, changes you make in the text file are propagated to the model. Then in the Equations dialog box:

- A **Link** column appears and indicates which equations are linked to the file.
- The **Link to external file** check box is selected and the path to the external file displays.



Equations and global variables that are linked to an external file cannot be modified in the Equations dialog box. They must be edited in the external file. To break the link and modify an equation in the dialog box, uncheck the box for that equation in the **Link** column.

Importing Equations

You can import equations from a text file into a part or assembly.

1. In the part or assembly, click **Tools > Equations**.
2. In the Equations dialog box, click **Import**.
3. In the **Open** dialog box:
 - a) Select the text file.
 - b) Click **Open**.
4. In the Import Equations dialog box:
 - a) Select or clear **Link to File**.
 - b) In the **Equations** column, clear the check mark for any equation you do not want to import.
 - c) In the **Link** column, clear the check mark for any equation you do not want to link to the external file.

When **Link to file** is selected, changes you make in the text file update the equations and variables in the model.
 - d) Click **Import**.

The equations from the text file are imported into the model.

If **Link to File** was selected, changes you make in the text file are propagated to the model. Then in the Equations dialog box:

- A **Link** column appears and indicates which equations are linked to the file.
- The **Link to external file** check box is selected and the path to the external file displays.



Equations and global variables that are linked to an external file cannot be modified in the Equations dialog box. They must be edited in the external file. To remove the link and modify an equation in the dialog box, clear the box for that equation in the **Link** column.

Creating Equations in External Files

You can create equations in a text file and then import them into one or more parts and assemblies.

1. Open a text file in an application such as Notepad.

2. Type equations in the same format as used in the Equations dialog box.
For example, you might define a global variable, `base`, and define two related dimensions.

```
"base" = 20
"D1@Boss-Extrude1" = "base" + 10
"D2@Sketch1" = ("base"*3) + 5
```



3. Save the text file.

You can import the text file into one or more parts and assemblies with the Equations dialog box. You can choose to link models to the text file, so that changes you make in the text file update the equations and variables in the models.

Changing Equations Through a Linked File

You can change equations in a linked text file and see the changes reflected in a part.

To change equations using a linked text file:

1. At the bottom of the Equations dialog box, click **Open linked file**  from any view.
The text file opens.
2. Change one or more of the equations.
3. Save the text file.
4. In the Equations dialog box, you can link and unlink individual equations by selecting and clearing the check boxes in the **Links** column.
5. Click **Rebuild**  to see the changed equations.

Create Equations in the Modify Dialog Box

When working on a model, you can enter equations and global variables directly in the Modify dialog box for dimensions. You do not have to open the Equations, Global Variables, and Dimensions dialog box to create an equation for a component.





The equations and global variables that you create in the Modify dialog box are reflected in the Equations dialog box.

Creating a Global Variable

To create a global variable in the Modify dialog box for a dimension:

1. In the model, double-click a dimension.
The Modify dialog box for dimensions displays, showing the name of the dimension and the current value.
2. In the second line of the Modify dialog box, enter `=` (equal sign). Enter the name of a new global variable and enclose it in quotation marks.

The text displays in yellow and the **Create Global Variable** button  appears next to the name.

3. Click the **Create Global Variable**  button, or hit **Enter**, or click  and **Yes**.
The global variable button  appears to the left side of the equation. Clicking the global variable button causes the entry to toggle between the global variable name and the evaluated value.
4. Click  to confirm.


Changes to the value of the global variable update all equations and variables in the model that reference that global variable.

The next time you open the Equations dialog box it will include:

- The new global variable.
- The equation making the dimension equal to the global variable.

Creating an Equation

To create an equation in the Modify dialog box for a dimension:

1. In the model, double-click a dimension.
The Modify dialog box for dimensions displays, showing the name of the dimension and the current value.
2. In the second line of the Modify dialog box, enter = (equal sign). Then enter an equation in the same way as in the Equations dialog box.
3. Click .

An Equations icon  appears next to the equation.

The next time you open the Equations dialog box, the new equation is listed.



You cannot change the evaluated value of an equation in the Modify dialog box.

Equations and Configurations

For parts and assemblies that have multiple configurations, you can view the global variables, dimensions, and equations that apply to a specific configuration by using the configuration drop-down list in the upper right corner of any view.

You can create global variables and equations that will be applied to some configurations but not others, for parts and assemblies that have multiple configurations. You can do this by creating the equation, then suppressing it for some configurations.


Applying Equations to Selected Configurations

To create a global variable or equation that will be applied to some configurations but not all:

1. Select the Equations View .
2. Create the global variable or equation and right-click.


- From the flyout menu, select **Suppress Specify Configurations** and indicate which configurations should not use this global variable or equation.



Select the Ordered View , if you want to see all global variables and equations that have been created for a part or assembly, including those that have been suppressed for some configurations.

Unsuppressing Equations to Selected Configurations

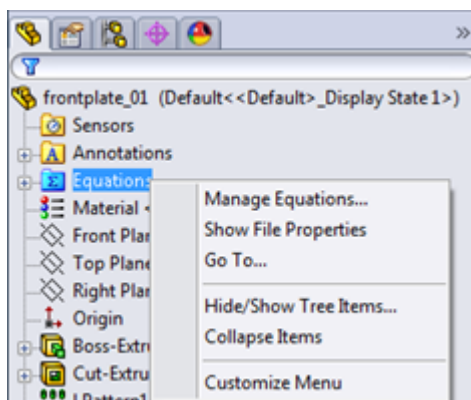
To unsuppress a global variable or an equation for one configuration:

- Select the Ordered View .
- Select a configuration from the drop-down list in the upper right corner of the dialog box.
- Select the row containing the global variable or equation and right-click.
- From the flyout menu, select **Unsuppress This Configuration**.


Use Equations

In this example, you modify the size of a model using the Equations, Global Variables and Dimensions and Modify dialog boxes.

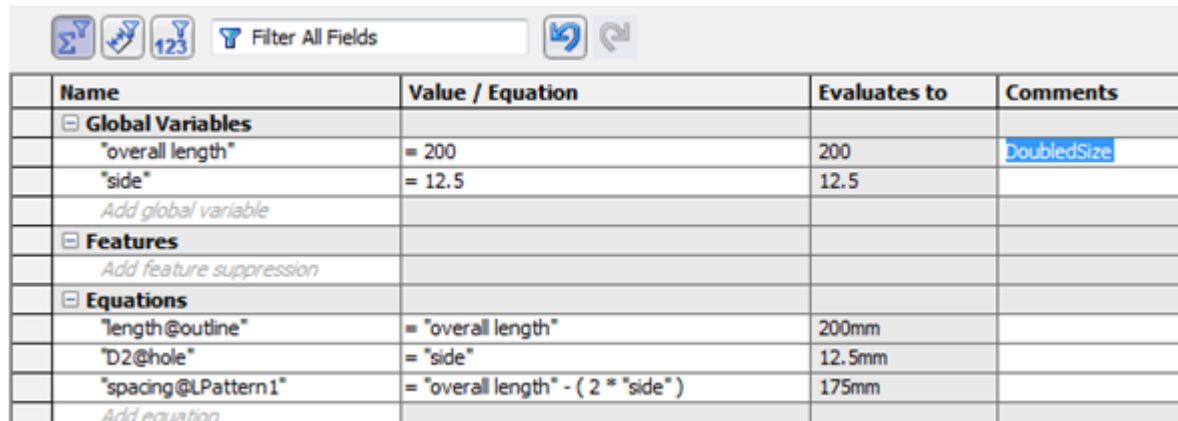
- Open `install_dir\samples\whatsnew\Equations\front_01.sldprt`.
- In the FeatureManager[®] design tree, right-click the **Equations** folder and click **Manage Equations**.




The Equations dialog box opens.

- Select the **Equations**  View.
- Click the first row in **Global Variables** for "overall length" and change the entry in the **Value/Equation** column from 100 to 200.
- Press **Enter** or **Tab** to move to the next cell in the **Comments** row.
The number updates in the **Evaluates to** column and the model automatically rebuilds. All the dimensions that use the global variable, "overall length", also double in number.


6. Type a comment such as Doubled Size.



Name	Value / Equation	Evaluates to	Comments
Global Variables			
"overall length"	= 200	200	DoubledSize
"side"	= 12.5	12.5	
<i>Add global variable</i>			
Features			
<i>Add feature suppression</i>			
Equations			
"length@outline"	= "overall length"	200mm	
"D2@hole"	= "side"	12.5mm	
"spacing@LPattern1"	= "overall length" - (2 * "side")	175mm	
<i>Add equation</i>			

7. Click **Undo**  until the "overall length" restores to its original size of 100mm.

Because **Automatically Rebuild** is selected, the model resizes to the original dimensions. All the dimensions that use the global variable "overall length" restore to the original dimensions. The **Comments** cell is also cleared.

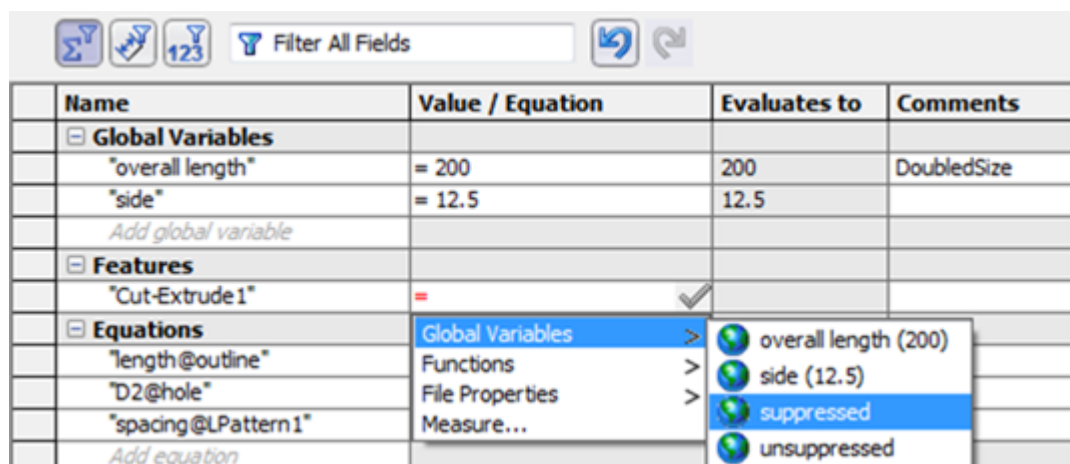
8. Click **Redo**  to restore the model to its new dimensions.

9. Suppress the **Cut-Extrude1** feature:

- In the **Features** section, click an empty cell.
- In the FeatureManager[®] design tree, click **Cut-Extrude1**.


The **Features** name displays. The SolidWorks software populates the next cell with = (equals sign) and displays a drop-down list with options for **Global Variables**, **Functions**, **File Properties**, and **Measure**.

- Expand **Global Variables**, select **suppressed**, and click .



Name	Value / Equation	Evaluates to	Comments
Global Variables			
"overall length"	= 200	200	DoubledSize
"side"	= 12.5	12.5	
<i>Add global variable</i>			
Features			
"Cut-Extrude1"	=		
Equations			
"length@outline"			
"D2@hole"			
"spacing@LPattern1"			
<i>Add equation</i>			

Suppressed is added to the **Evaluates to** column.

 You can also define suppression by using the syntax of the Visual Basic `if` function.

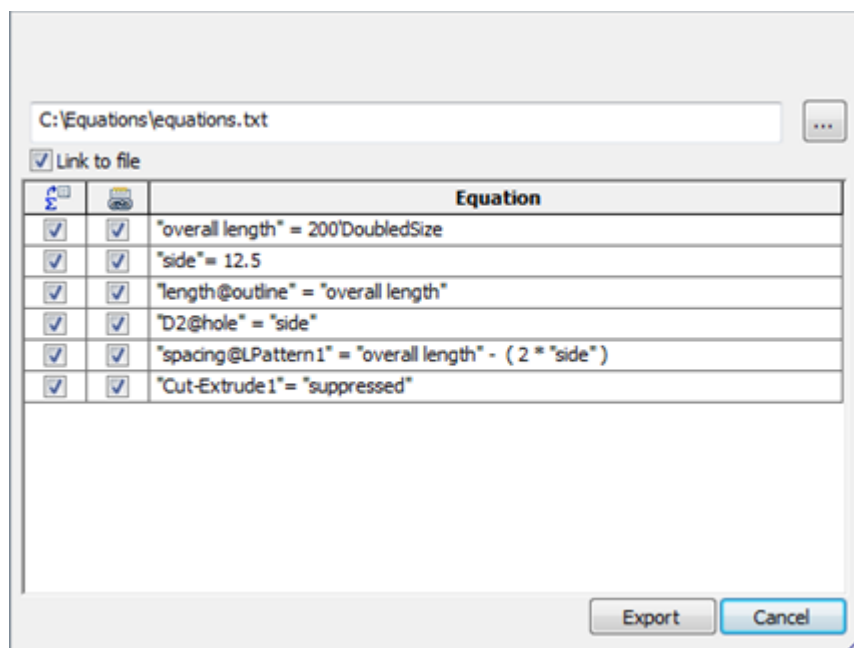
Exporting and Linking Equations


These steps demonstrate the improved workflow for exporting and linking to text files.

1. In the Equations dialog box, click **Export**.
2. In the Save As dialog box:
 - a) Specify a name and location for the text file.
 - b) Click **Save**.

The equations are saved in a text file. The Export Equations dialog box appears.


The Export Equations dialog box includes two columns. The first column indicates the equations that will be exported to a text file. The second column indicates the equations that will be linked between the model and the text file, so that changes are replicated in both. By default, all equations are exported and linked.



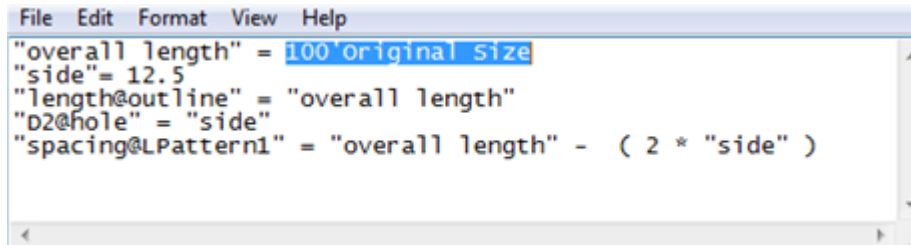
 You can link and unlink individual equations by selecting and clearing the check boxes in the **Links** column. The same capabilities are available in the Equations dialog box when you link to an external file.

3. In the Export Equations dialog box, click **Export**.
In the Equations dialog box, the **Link to external file** option is selected. The file path also appears.



4. In the Equations dialog box, click **Open linked file** .
The file opens in a text editor.


- Change "overall length" from 200 to 100 and update the comments from DoubledSize to Original Size. Then save the file.



```
File Edit Format View Help
"overall length" = 100 original size
"side" = 12.5
"length@outline" = "overall length"
"D2@hole" = "side"
"spacing@LPattern1" = "overall length" - ( 2 * "side" )
```

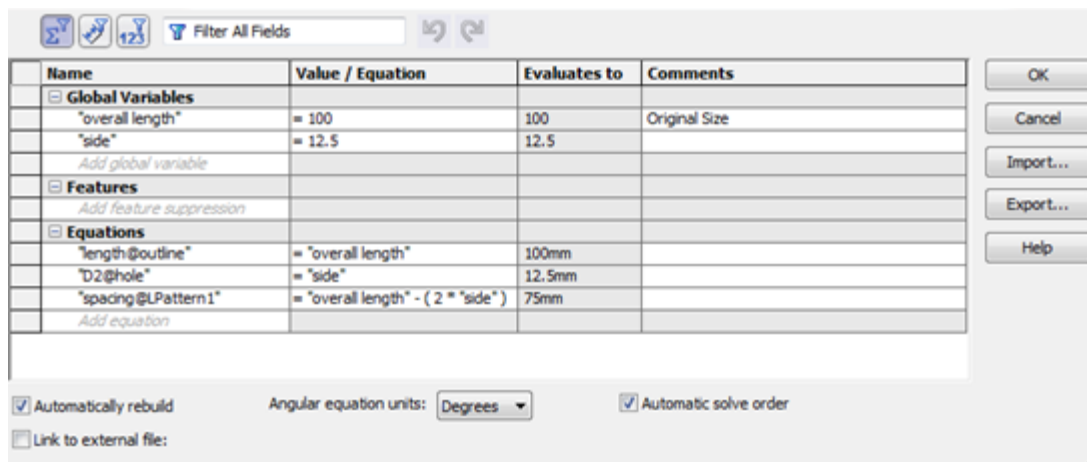
- In the Equations dialog box, click **Rebuild** .

All the dimensions that use the global variable, "overall length" reduce to the original dimension. The Equations dialog box displays an additional column for all the entries linked to the external file.

 Equations and global variables that are externally linked can no longer be changed in this dialog box. You must use the external file to modify them.

- In the Equations dialog box, clear **Link to external file**.

Now all equations and global variables can be changed in the dialog box. The path to the external file and the **Link** column are also removed.

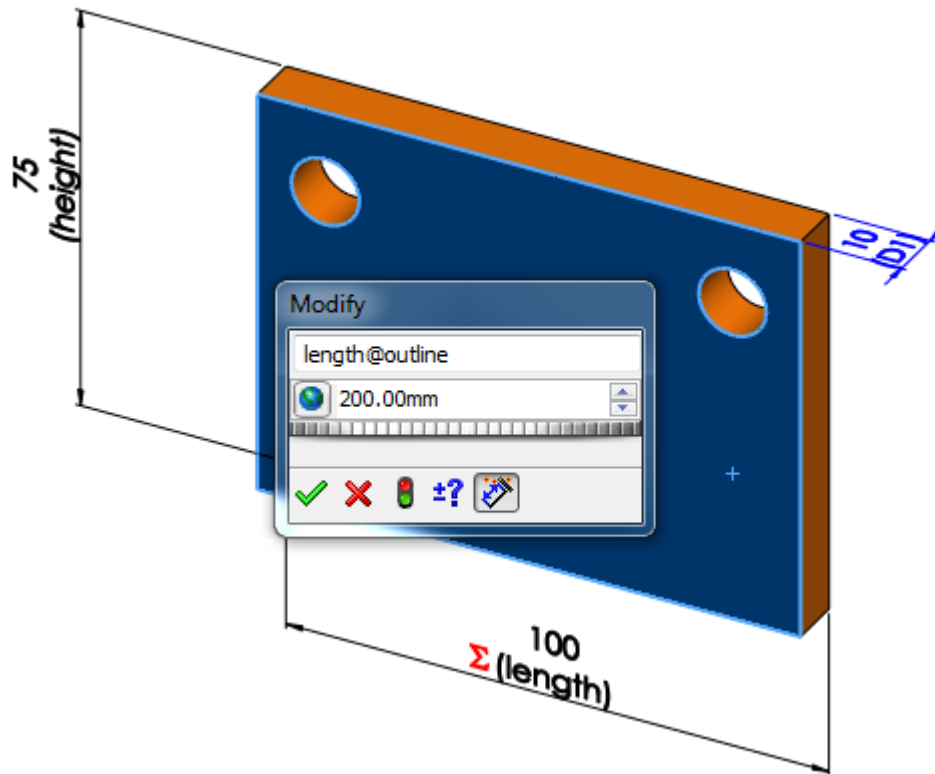




Changing Equation Values with the Modify Dialog Box



When you change a value in the Modify dialog box, the value is automatically updated in the model and the Equations, Global Variables and Dimensions dialog box.

- In the FeatureManager design tree, click **Boss-Extrude1**.
The model opens in sketch mode.
- In the sketch, double-click the `length@outline` dimension.

- In the Modify dialog box, change the dimension from 100 to 200.



 When you insert or change a global variable in the Modify dialog box, it displays the Create Global Variable  in the input field. Both the dimension "length@outline" and the global variable, "overall length", represent the same value.

- Click **Rebuild**  and then .
- In the FeatureManager[®] design tree, right-click the **Equations** folder and click **Manage Equations**.
The Equations dialog box displays the new value for "length@outline" dimension and the global variable, "overall length".

SolidWorks Enterprise PDM

Available as a separately purchased product that can be used in SolidWorks Standard, SolidWorks Professional, or SolidWorks Premium.

This chapter includes the following topics:

- **Administration Tool**
- **File Explorer and SolidWorks Add-in**
- **Product Support**
- **System Performance**
- **Web Client Access Extended**

Administration Tool

Customized Enterprise PDM Menus

You can customize how menus and submenus are displayed in the Windows Explorer user interface.

You can change the available commands and command order for these menus on the Enterprise PDM Menu bar:

- **Actions**
- **Modify**
- **Display**
- **Tools**

You can customize shortcut menus that display in the File View pane when:

- Files are selected
- Folders are selected
- Nothing is selected

Customizing Menus

Use the Menus page of the Settings dialog box to customize menus displayed to users in the Windows Explorer File View.

To customize a user's menus:


1. In the Administration tool, expand **Users** and double-click the user whose menus you want to customize.
2. In the Properties dialog box, click **Settings**.
3. In the left pane of the user's Settings dialog box, click **Menus**.


4. In the **Destination Menu** drop-down list, select the File View menu to modify:



Background Right click	The shortcut menu displayed when the user right-clicks with nothing selected.
File Right click	The shortcut menu displayed when the user right-clicks with a file selected.
Folder Right click	The shortcut menu displayed when the user right-clicks with a folder selected.
Bar\Actions	The Actions menu on the Enterprise PDM menu bar.
Bar\Modify	The Modify menu on the Enterprise PDM menu bar.
Bar\Display	The Display menu on the Enterprise PDM menu bar.
Bar\Tools	The Tools menu on the Enterprise PDM menu bar.

The commands for the selected menu are listed.

5. Do any of the following:

- To remove a command, select it and click .
- To add a separator to the menu, select the command you want below the separator and click .
- To add a command to the menu, under **Available commands**, use the Flat list or Categorized tabs to display the available commands.

Drag the command to the location you want or click  to add the selected command to the bottom of the menu.

- To change the location of a command or divider in the menu, select it and click  or .

6. To modify another menu, repeat steps 4 and 5.



You can add the same command to multiple menus.

7. Do one of the following:

- Click **Apply** to save your changes and close the Settings dialog box.
- Click **Cancel** to close the Settings dialog box without saving your changes.

When prompted to confirm that you do not want to save changes, click **Yes**.

Copying Menu Settings

You can create customized menus for one user and copy them to other users.

To copy customized menus from another user:

1. On the Menus page of the Settings dialog box, click **Copy**.

2. In the Copy Menu Settings dialog box, under **Menus to copy**, do one of the following:
 - Select **<Copy All Menus>**.
 - Select a specific menu to copy.
3. Under **Source user**, select the user whose settings you want to copy.
4. Click **OK**.
5. Click **Apply**.

Returning to Default Menu Settings

You can return a user's menus to their default settings. Resetting returns all menus to their default settings.

1. On the Menus page of the Settings dialog box, click **Reset**.
2. When prompted to confirm that you want to reset all menus, click **OK**.

Displaying Full User Names in the User Interface

If login names are cryptic, you can display users' full names in the user interface to make it easier to identify who is using a file.

In Enterprise PDM, user names are displayed to indicate who has checked out or created files, sent or received mail messages, or initiated actions performed on files.

If your company creates users with truncated user names, for example, by importing users from Windows Active Directory, it may be difficult to identify individuals just by their login names.

You can modify your user settings in the Administration tool to specify that the full names of users are displayed throughout the user interface.

Compare these examples of the **Checked Out By** column in File View in Windows Explorer:

Login names displayed:

Name	Checked Out By
Bevel Gears.SLDASM	Dev1
Base.SLDPRT	Dev2
bevel3.SLDPRT	Dev2
Bevel Gears.avi	Mgt2
Bevel_Gearing.pdf	Sales1
Optimized_Straight_Bevel_Gear.pdf	Sales1

Full names displayed:

Name	Checked Out By
Bevel Gears.SLDASM	Paul Burke
Base.SLDPRT	Donna Parsons
bevel3.SLDPRT	Donna Parsons
Bevel Gears.avi	Irina Tresvik
Bevel_Gearing.pdf	Amit Sagi
Optimized_Straight_Bevel_Gear.pdf	Amit Sagi

To display full user names:

1. In the left pane of the Administration tool, expand **Users**.
2. Right-click your user name and click **Settings**.
3. In the left pane of the Settings dialog box, click **Explorer**.
4. Under **Miscellaneous**, select **Show full user names**.
5. Click **Apply**.

Replication Failover

Replication has been made more fault-tolerant to handle WAN scenarios where not all servers are available at all times.

If the first attempt to retrieve a file fails, the Archive server uses a lookup table to select other servers from which to replicate the file. This table prioritizes servers based on replication settings you provide using the Administration tool. If each attempt to get the file fails, the Archive server retries the operation.

By default, the replication server retries twice before it times out. You can change the number of retries by creating a new DWORD key on the archive server called `RetriesCount` at the following registry location:

```
HK_LOCAL_MACHINE\SOFTWARE\SolidWorks\Applications\  
PDMWorks_Enterprise\ArchiveServer\Vaults\vault_name
```

The hexadecimal value you set for the `RetriesCount` keyword controls the number of retries.

File Explorer and SolidWorks Add-in

Browse to Option

The **Browse to** option on the shortcut menu of the Contains and Where Used tabs changes the focus of the tab to the referenced file to which you browse.

In previous versions, the **Browse To** option opened a new Windows Explorer window. In SolidWorks Enterprise PDM 2012, the **Browse to** option changes the focus to the referenced file without opening a new window.

The new behavior also applies when you navigate to vault files using the File Explorer Contains and Where Used tabs in the Open and Save As dialog boxes. This behavior occurs whether you access the dialog boxes from the SolidWorks software or from any standard Microsoft Windows application such as Notes and Microsoft Office.

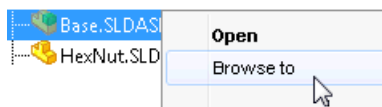
You can also open a new window using the new **Browse to in a New Window** option.

Using Browse to

Use **Browse to** when you want to change focus to a referenced file without opening a new dialog box.

This example describes how to change the focus of the Contains tab to a referenced file without opening a new dialog box.

1. In the File View pane, select a vault file and click the Contains tab to list the files it references.
2. Right-click a referenced file and click **Browse to**.



The focus of the Contains tab changes to the file you selected.

Using Browse to in a New Window

Use **Browse to in a New Window** when you want to open a new window with the focus on a selected file.

1. In the File View pane, select a vault file and click the Where Used tab to list the files that reference it.
2. Right-click a file, and click **Browse to in a New Window**.



A new window opens to the same folder and tab, but with the focus changed to the file you selected.

The original window remains open.

Exporting Indented Bills of Materials

You can preserve the indented structure of an Enterprise PDM computed bill of materials (BOM) or a saved BOM when you export it to a Microsoft Excel spreadsheet. When prompted, choose to add a Level column to the output file. The added column contains a numerical representation of the BOM's levels.

To preserve the indented structure of a BOM:

1. In Windows Explorer, select a vault assembly.
2. On the Bill of Materials tab, click **BOM Display - Indented** .
3. Click **Open As CSV** .
4. When prompted to create a Level column in the CSV file, click **Yes**.

A **Level** column, with numerical values representing the BOM structure, appears as the first column in the spreadsheet. You can use the column to sort the exported BOM.


Searching Within File-based Dialog Boxes

The embedded Enterprise PDM search lets you search for files and folders within Windows Explorer and all standard file-based dialog boxes.

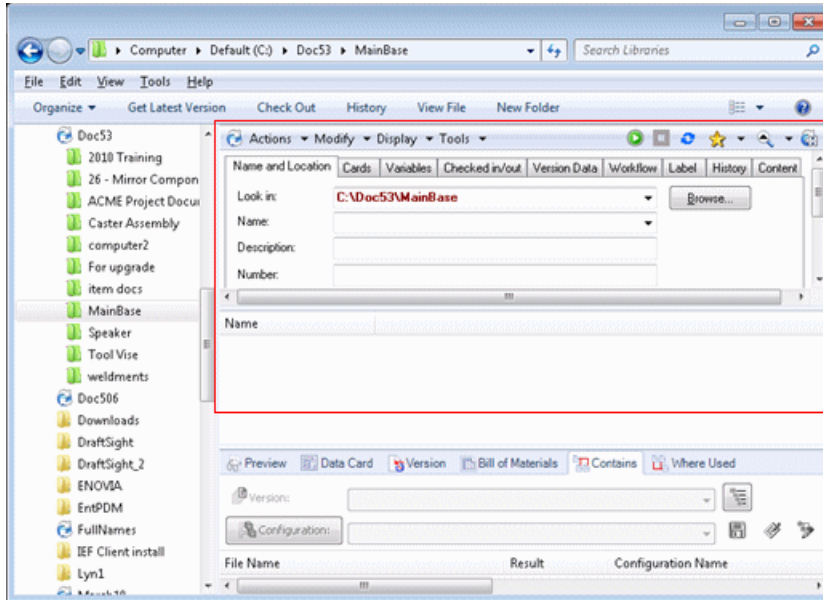
You can use this method to search from:

- SolidWorks file-based dialog boxes, such as Open, Save As, Insert, and Replace.
- The Open and Save As dialog boxes of applications such as Microsoft Word and Excel, and Adobe Reader.



To perform an embedded search:


1. Do one of the following:
 - Click **Open Search**  (Enterprise PDM menu bar).
By default, the **Complete Search** card is used.
 - If you have access to multiple file search cards, expand **Open Search** and select a search card to use.

The search card appears in the top right pane.



2. Do one of the following:


- Use the criteria on the card to define the search and click **Start Search** .
- If you have saved searches as favorites, expand **Favorite Searches**  and select the search to use.

3. To restore all fields on the search card to their default values, click **Clear Search** .

4. To close the embedded search, click **Close Search** .

The embedded search can only be used to search for files or folders. To search for items or users, use the Enterprise PDM Search tool.

To open the Enterprise PDM Search dialog box:

1. Expand **Open Search**  and click **Search Tool**.
2. Select the search card to use from the **Searches** list in the left pane.

Version Toggle on Where Used Tab

On the Where Used tab, you can use the **Version** button to toggle between viewing references for all versions of a file and selecting the file version for which references are displayed.

Additionally, a redesigned **Configurations** button on the Where Used and Contains tabs hides all configurations or lets you select the configuration to use.

Both the **Version** and **Configurations** toggle states persist:


- When you select a different file in File View
- When you change folders
- Between Enterprise PDM vault sessions

Displaying Versions

The **Version** button is active on the Where Used tab.

On the Contains tab, **Version** serves as a label for the version dropdown list. You can select specific versions, but you cannot select **<All Versions>**.

With a vault file selected in File View, on the Where Used tab, do one of the following:

- To display files that reference all versions of the selected file, to the left of the top drop-down list, click  Version: .

The **Version:** button is highlighted, and the drop-down list displays **<All Versions>**.



When all versions are displayed, you cannot select a specific configuration to display.

- To display files that reference a specific version of the selected file, click  Version: .

The **Version** button is no longer highlighted, and you can choose a specific version from the drop-down list.



The highlighted **Configuration** button indicates that you can select a configuration from the drop-down list.

Displaying Configurations

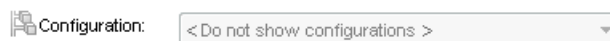
On both the Where Used and Contains tabs, the **Configuration** button lets you enable or disable the choice of configurations.

With a vault file selected in File View, use these methods to display configurations:

1. On the Contains tab:

- When the **Configuration** button is active, use the dropdown list to select the configuration for which to display referenced files.
- To deactivate the **Configurations** drop-down list, click **Configuration**.

The button becomes a label and the drop-down list indicates that you have chosen not to show configurations.



2. On the Where Used tab:

- a) If **Version** is active, click to deactivate it, and click **Configuration** to activate the ability to choose configurations.










The **Configuration** button is highlighted and the drop-down list becomes active.

- b) Use the drop-down list to select the configuration to display.






Dialog Box Toolbars

A toolbar has been added to the dialog boxes you use to manipulate groups of files. This makes it easier to access commands that were previously only available through a shortcut menu.

The toolbars contain new buttons and in some cases, modified buttons.

Button	Command	Description
	Select Files	Lets you use wildcards to select a group of files.
	Open File List	Opens a list of files in Microsoft Excel.
	Save File List	Lets you save the file list as a comma-separated .txt file.
	New flyout control:	
	Generate for configurations	Lets you link an item to a specific file configuration.
	Generate for files only	Lets you link an item to a file independent of configurations.
	New flyout control:	
	Referenced version	Uses the attached version of the references.
	Latest version	Uses the latest version of the references.

The following dialog boxes contain the toolbars:

Dialog Box	Access					
Get	Action > Get Latest Version Action > Get Version	X	X	X		
Check In	Action > Check In	X	X	X		
Check Out\ Undo Check Out	Action > Check Out	X	X	X		
Do Transition	Modify > Change State	X	X	X		
Increment Revision	Modify > Increment Revision	X	X	X		X
Link Files to Item	From Item Explorer dialog box: Right-click an item and select Link Files to Item . Select one or more files and click Open .	X	X	X	X	X

Product Support

SolidWorks Enterprise PDM 2012 offers expanded support for CAD systems, Microsoft[®] Office, Internet Explorer[®], and 64-bit processors.

The following support is available:

CAD Tools	Microsoft Products	64-bit Platforms
SolidWorks 2012	Microsoft Office 2010	Enterprise PDM Web Server
AutoCAD [®] 2012	Internet Explorer 9	AutoCAD
Autodesk Inventor [®] 2012		Solid Edge
Solid Edge [®] ST3		

System Performance

System performance has been improved for users working with large data sets and users working remotely with Copy Tree and template-created documents.

- Improved system response time with large data sets

Users will now see significantly quicker system response when working with large data sets.

In previous versions of Enterprise PDM, long delays or system stoppages could occur when users worked with large data sets with many parent child relationships. Affected functions included viewing the information on the Contains tab in Windows Explorer or in a CAD add-in when the Archive refresh rate is set at a low value, for example, five seconds.

- Creating new documents from remote replicated sites with high latency connections to the SQL Server

It now takes less time for users in remote offices to create new documents using templates and the Copy Tree tool.

Combining the queries that save card variables has made the creation of documents from templates as much as three times faster.

Improved query methods have optimized WAN traffic for the Copy Tree operation, making the new response time up to 10 times faster.

Web Client Access Extended

The SolidWorks Enterprise PDM Editor license now includes access to the Enterprise PDM Web Client. This provides greater flexibility for users that are away from the office but still need to access the system, and for external users, such as vendors or customers, that need occasional access.

In addition to providing access to CAD applications, you can use Editor licenses to access the Web client to search the vault and perform basic functions to enable remote work processes.

Licenses in the license pool are used in this order:

- Contributor licenses

- Editor licenses
- Processor license

Flow Simulation

Available as a separately purchased product that can be used with SolidWorks Standard, SolidWorks Professional, or SolidWorks Premium. Some Flow Simulation features are only available with the HVAC module add-on.

This chapter includes the following topics:

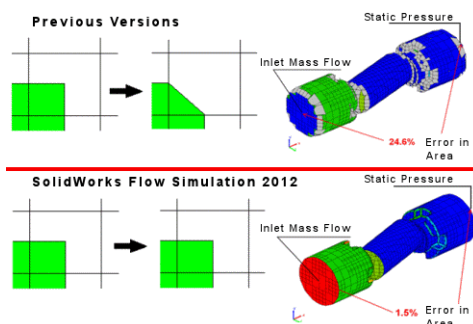
- **Physical Models and Technology**
- **Pre-Processor**
- **Post-Processor**

Physical Models and Technology

Advanced Geometry Resolution

Enhanced meshing technology improves the CAD geometry representation by the computational mesh. As a result, the accuracy of calculation is improved with fewer number of cells.

In previous releases, there were partial cells that simplified the geometry by cutting the sharp corners residing in the cell. The new technology uses additional information from the CAD (e.g. edges) to construct the non-simplified geometry, wherever possible.



DO Radiation Model

The DO radiation model is improved to give better accuracy in cases where geometrical optics (e.g. focusing on lens, shadowing) is essential for the simulation.

Three discretisation level parameters have been added to the DO model:

- Level 5 (2nd order)
- Level 6 (2nd order)
- Level 7 (2nd order)

For the directional (solar) radiation, the new algorithm is always enabled providing the sharper shadows behind the objects.

For diffusive (thermal) radiation, it is optionally enabled by selecting one of the 3 new discretisation levels.



The DO radiation model is available in the **HVAC** module only.

Draught Rate

A new comfort parameter called **Draught Rate (DR)** is available that represents a percentage of people feeling discomfort by draught (ISO 7730).



The new comfort parameter is available in the **HVAC** module only.

Hybrid Technology for Isothermal Cavitation

To solve hydraulic (isothermal) cavitation problems, a new hybrid method is introduced.

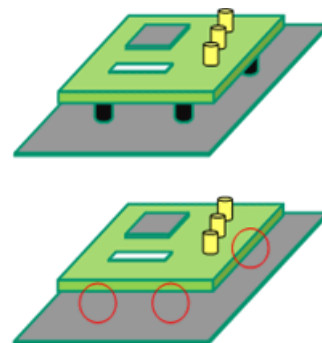
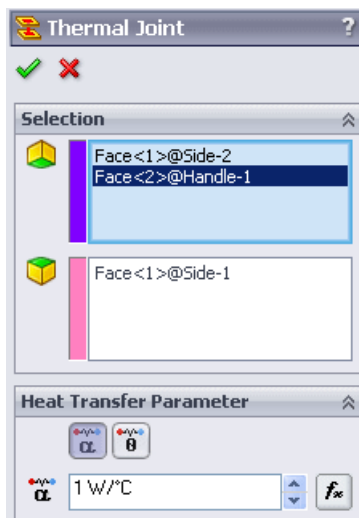
Hydraulic (isothermal) cavitation occurs in liquid flows when the pressure falls below a critical value. As a result a supersonic flow frequently occurs in two-phase cavitation zones. To solve such problems, the hybrid solver applies both incompressible (liquid area) and compressible solvers (two-phase area).

The hybrid method automatically turns to appropriate approach depending on flow regime. It only applies to isothermal cavitation.

Thermal Joint

The new **Thermal Joint** helps you simulate the heat transfer from one face to another by specifying thermal resistance.

You can transfer heat from one face (or assembly of faces) to another face (or assembly of faces) without modeling the conductor itself. Specify the Heat Transfer Parameter either as the Integral Heat Transfer Coefficient (W/K), or the Thermal resistance (K/W).



Tracer Study

With a new tracer feature, you can quickly simulate concentration source, surface condensation, or evaporation of an admixture fluid (for example, water vapor or toxic gas) in the carrier fluid (for example, air). The presence of the admixture has negligible influence on the carrier fluid flow.

Tracer is an admixture presented at low mass fraction in a carrier fluid with no influence (passive) on this carrier fluid. If concentration of a gas admixture is low you can consider it as a tracer (examples of tracers in air: water vapor, H₂, CO, NO, NO₂, Cl₂, HCl, NH₃, H₂S, SO₂).



The tracer feature is available in the **HVAC** module only.

Pre-Processor

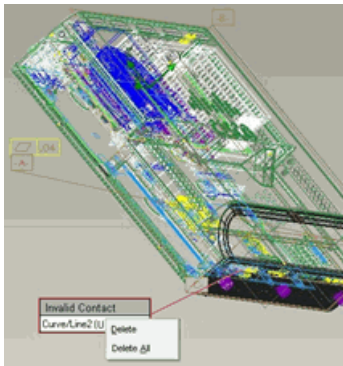
Automatic Rebuild Options

You can now disable the automatic rebuilds.

When you define a Fluid Flow Simulation and modify the design, you can disable the automatic rebuild in the option. This enhancement improves efficiency when you test different design scenarios.

Callout at Invalid Contact

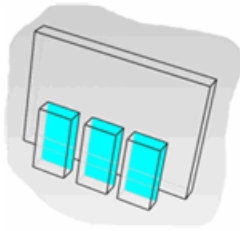
You can add callouts to invalid contact locations. The callouts identify improper contact surfaces so you can fix them. The callouts also save you time by detecting the reasons for geometry issues.



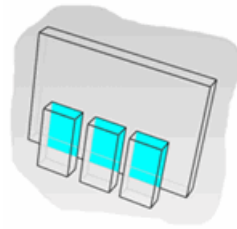
Contact Resistance

The **Apply to solid/solid only** option in the Contact Resistance dialog box allows you to specify contact resistance only at overlapping parts of solid surfaces in contact.

In the **Contact Resistance** PropertyManager, select **Apply to solid/solid only**.



Apply to solid/solid only: OFF



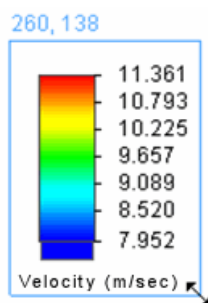
Apply to solid/solid only: ON

Post-Processor

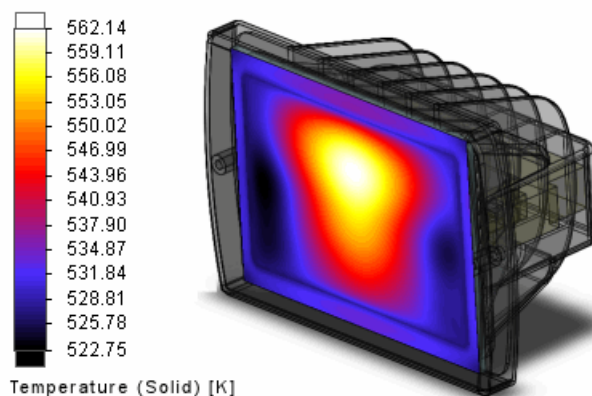
Color Bar

Several usability improvements are implemented for the **Color Bar**.

- You can dynamically adjust the palette size, position, and number of levels of the **Color Bar**.



- A new option **Auto scale** in the Font dialog box adjust automatically the font size for text in the **Color Bar**.
- You can define the default settings and appearance of the **Color Bar** under **General Options**.
- The **Color Bar** dialog box now contains a range of new palettes optimized for visualization of the temperature distribution.

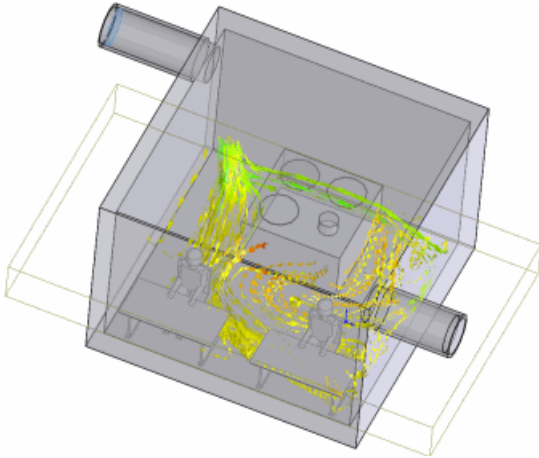


Context Animation

You can save the animation displayed with the **Context Animation** tool.

Crop Region for Flow Trajectories

You can define a 3D box that crops the flow trajectories area. Use this option for large models to focus on the region of interest.

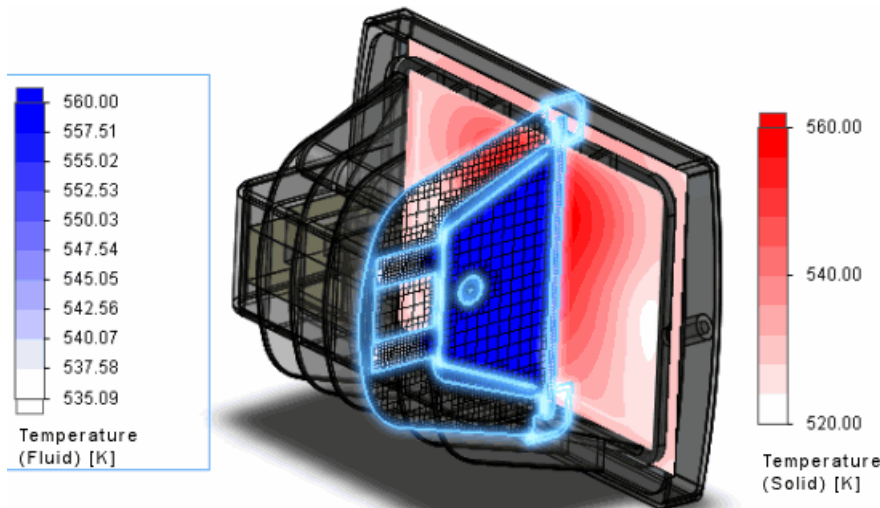


Exporting Results

You can simultaneously export multiple surface parameters to Microsoft Excel.

High-quality Highlight

A high-quality highlight is available for indicating the selected plots or trajectories. Use this option to distinguish contours of different parameters in one plot.



Parameter List

The **Parameter List** dialog box has an improved interface that allows you to select parameters more easily while viewing results. You can sort them by name, group or unit.

Reports

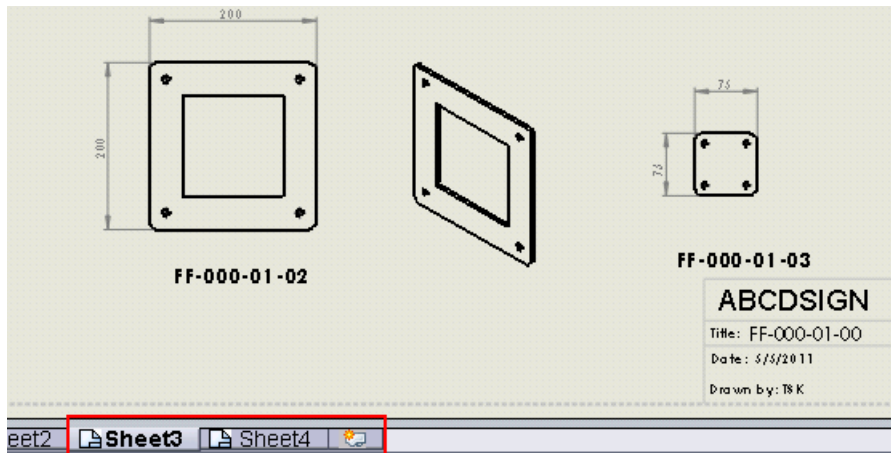
- You can use custom templates to generate reports when you analyze the results in the Batch-Results-Processing mode.
- You can generate analysis reports in HTML file format in addition to Microsoft Word file format.

This chapter includes the following topics:

- **Exporting Multisheet Drawings to the DXF or DWG Paper Space**
- **Importing Creo Elements/Pro (Pro/Engineer) Files**
- **Importing .IFC Files**
- **Importing Unigraphics Files**
- **STEP and Parasolid Assembly File Import**

Exporting Multisheet Drawings to the DXF or DWG Paper Space

You can export all sheets of a multisheet drawing to the file layout (paper) space and save the document as a single .dxf or .dwg file.



From a multisheet drawing:

1. Click **File > Save As**.
2. In the dialog box, for **Save as type**, select **DXF** or **DWG**.
3. Click **Options**.
4. In the Export Options dialog box, for **Multiple sheet drawing**, select **Export all drawing sheets to paper space**.
5. Set other options and click **OK**.
6. In the Save As dialog box, type the filename and click **Save**.
7. If the SolidWorks to DXF/DWG Mapping dialog box appears, specify mapping options and click **OK**.

Importing Creo Elements/Pro (Pro/Engineer) Files

You can import Pro/ENGINEER files up to version Creo Elements/Pro 5.0 (formerly Pro/ENGINEER Wildfire 5).

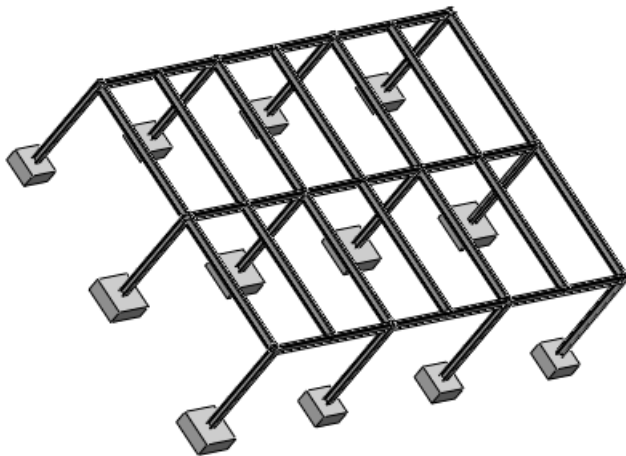
1. Click **File > Open**.
2. For **Files of Type**, select **ProE Part** or **ProE Assembly**.
3. Select the file and click **Open**.

Importing .IFC Files

You can import `.ifc` files into the SolidWorks software to include information from models originally created with structural software applications.



When you import `.ifc` files into SolidWorks, the file geometry is imported but the resulting SolidWorks model has essentially no features.



1. Click **File > Open**.
2. For **Files of Type**, select **IFC 2x3**.
3. To set general options, click **Options**, set the options, and click **OK**.
4. Select the file and click **Open**.

Importing Unigraphics Files

You can import Unigraphics files up to version 7.5.

1. Click **File > Open**.
2. For **Files of Type**, select **Unigraphics**.
3. To set general options, click **Options**, set the options, and click **OK**.
4. Select the file and click **Open**.

STEP and Parasolid Assembly File Import

STEP and Parasolid assembly file import performance has been improved. In addition, the SolidWorks software does not automatically save the resulting the SolidWorks file to your disk.

Motion Studies

Available in SolidWorks Premium.

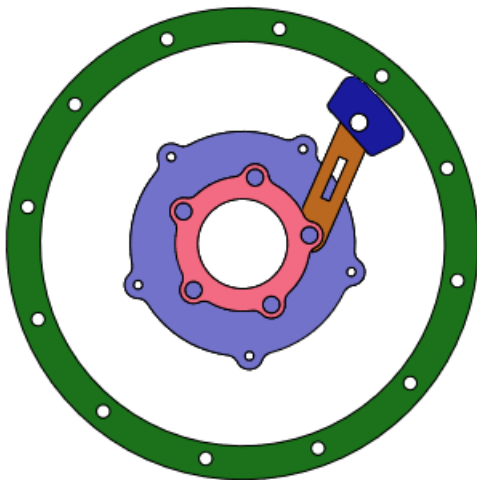
This chapter includes the following topics:

- **Motion Optimization**
- **Including Motion Results in Sensors**

Motion Optimization

You can create a SolidWorks Simulation design study to optimize motion studies features.

- From the Parameters dialog box you can create and link variables to features from Motion Analysis type studies.
- In a design study, you can specify constraints and goals that are linked to motion study results through sensors. You can perform an optimization or evaluate specific design scenarios.
- For each design iteration, you can view the motion animation within the design study. For any iteration of the design study, you can view detailed motion results in the motion study.










Including Motion Results in Sensors

You can define motion data sensors from motion results.



You can use motion data sensors to track a motion study result value. With a SolidWorks Simulation Professional license, you can use motion data sensors to optimize motion parameters in a SolidWorks Simulation design study.

To create a motion data sensor to track a motion study result:

- From a Motion Analysis type study, do one of the following:
 - Define the sensor by creating a new result first:
 1. Click **Results and Plots**  (MotionManager toolbar).
 2. Specify the result details.
 3. Select **Create new motion data sensor**.
 4. Specify the other sensor properties and click .
 - Define a new sensor to reference an existing result:
 1. From the FeatureManager design tree, right-click **Sensors**  and click **Add Sensor**.
 2. For **Sensor Type** , select **Motion Data**.
 3. For **Motion Study** , select a motion study.
 4. For **Motion Study Result** , select a result defined in the selected motion study.
 5. Specify the other sensor properties and click .

Parts and Features

This chapter includes the following topics:

- **Feature Freeze**
- **Exploded Views of Multibody Parts**
- **Hole Wizard - Placement of Holes**
- **Part Reviewer**
- **Templates for Split Parts**
- **Reference Planes**

Feature Freeze

You can freeze features to exclude them from rebuilds of the model.

The freeze bar controls the point at which a part's FeatureManager design tree rebuilds. Features above the freeze bar are frozen – you cannot edit them, and they are excluded from rebuilds of the model.

Freezing a portion of a model can be useful if you work with complex models with many features. Freezing the features helps to:

- Reduce rebuild time
- Prevent unintentional changes to the model



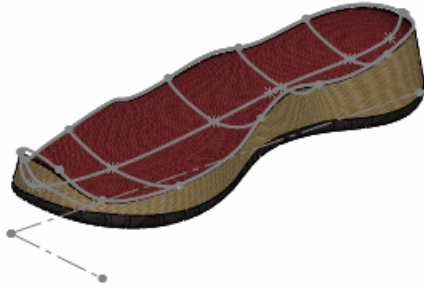
Feature Freeze prevents the geometry of frozen features from being rebuilt. However, you might still experience long rebuild times due to other processes that are not addressed by Feature Freeze. Examples of potentially time-consuming processes not addressed by Feature Freeze:


- Updating display appearances, especially on very large patterns
- Updating complex DimXpert dimension and tolerance schemes
- Updating the graphics (tessellation data) of very large, complex parts

Enabling Feature Freeze

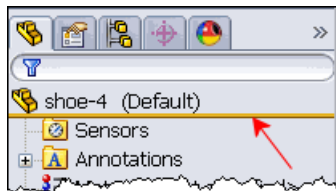
You turn on Freeze in System Options.

1. Open `install_dir\samples\whatsnew\Parts\shoe-4.sldprt`.



2. Click **Options**  (Standard toolbar) or **Tools > Options**.
3. On the System Options tab, click **General** and select **Enable Freeze bar**.
4. Click **OK**.


The yellow freeze bar appears near the top of the FeatureManager design tree, under the part name.

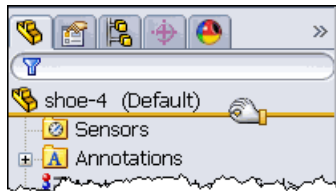



Freezing Features

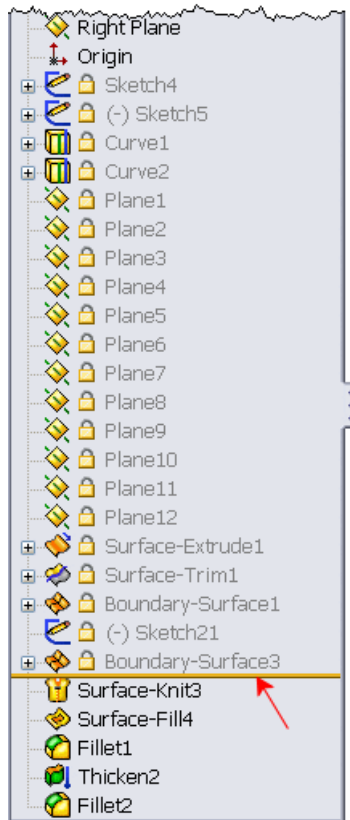
You drag the freeze bar to freeze features.


1. Move the pointer over the freeze bar.


The pointer changes to .



2. Drag the freeze bar down below **Boundary-Surface3** .

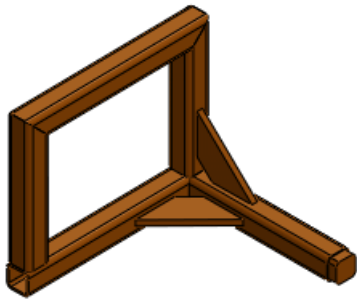
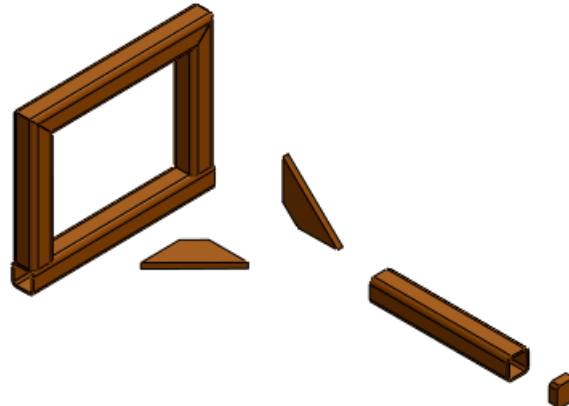


 When the freeze bar is at the top of the tree, you can also right-click a feature and click **Freeze** to freeze that feature and all features above it in the FeatureManager design tree.

Features above the freeze bar are frozen – you cannot edit them, and they are excluded from rebuilds of the model. Frozen features are indicated by an icon  and gray text.

Exploded Views of Multibody Parts

You can create exploded views of multibody parts.

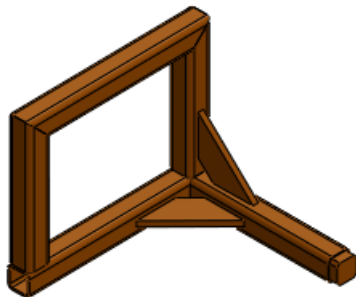
Collapsed**Exploded**

The process is similar to creating exploded views in assemblies.

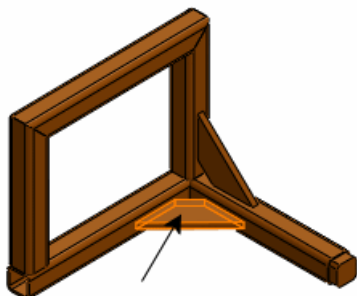
Creating an Exploded View


You can create exploded views of multibody parts.

1. Open `install_dir\samples\whatsnew\Parts\weldment01.sldprt`.

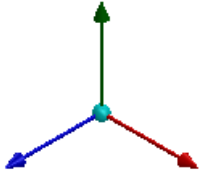


2. Click **Insert** > **Exploded View**.
3. In the graphics area, select **Gusset2**.



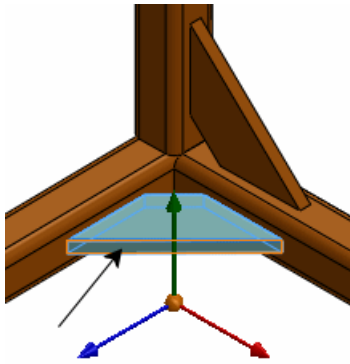
In the PropertyManager, under **Settings**, **Gusset2** appears in **Bodies of the exploded step** .

A triad appears in the graphics area, aligned with the X, Y, and Z axes of the model.

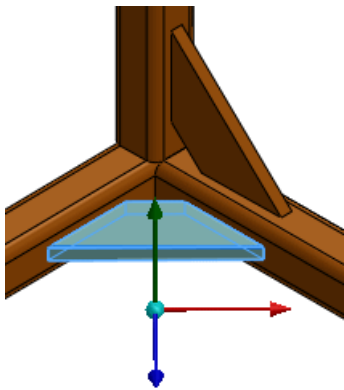


Next, change the alignment of the triad.

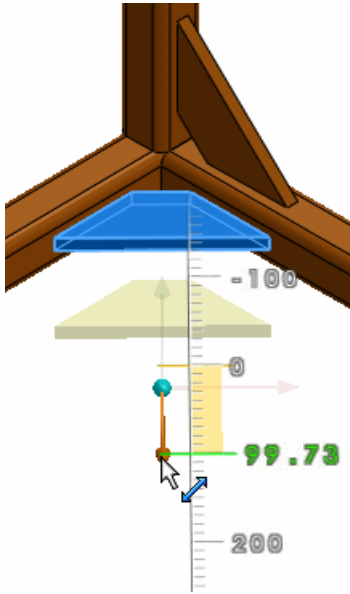
4. Right-click the center sphere of the triad and click **Align to**.
5. Select the narrow face of the gusset.



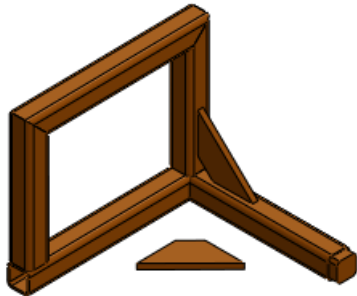
The blue handle aligns normal to the selected face.



6. Drag the blue handle to move the gusset forward.



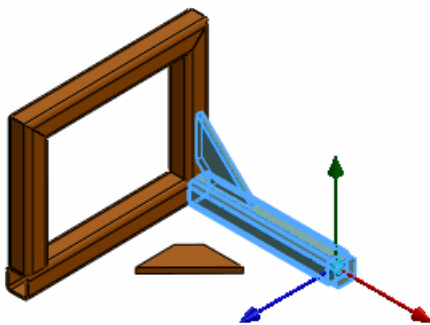
7. Drop the gusset approximately as shown.



Auto-spacing Bodies

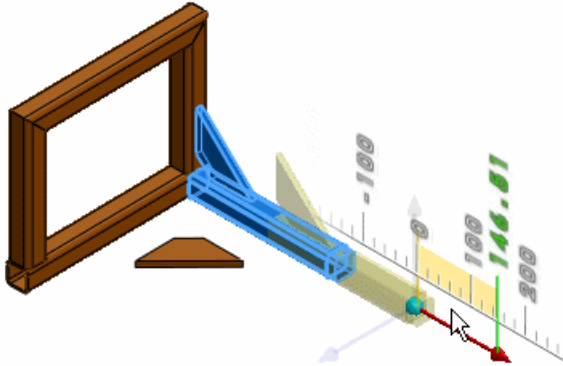
You can evenly space exploded groups of bodies.

1. In the graphics area, select the other gusset, the right-most structural member, and the end cap.

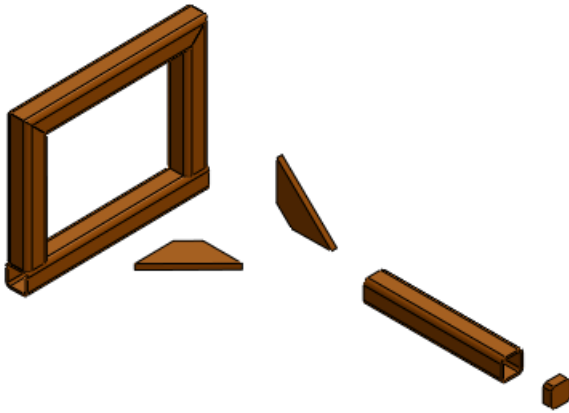


2. In the PropertyManager, under **Options**, select **Auto-space Solid Bodies after drag**.

3. Drag the red handle of the triad to position the bodies as shown.




When you drop the group of bodies in the position shown, one remains where you drop it, and the software automatically spaces the others further along the same axis.



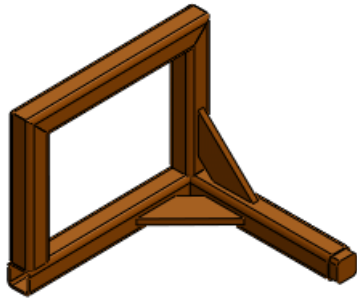
4. Click .

Collapsing an Exploded View

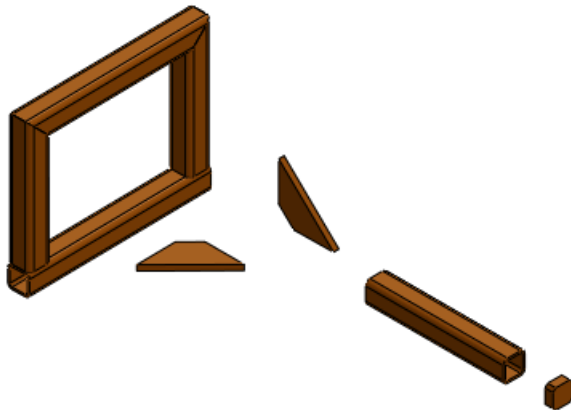
You can toggle between the exploded and collapsed views.

1. In the Manager Pane, on the ConfigurationManager tab , expand **Default<As Machined>**.

2. Double-click **ExplView**, or right-click it and click **Collapse**.



3. To return to the exploded view, double-click **ExplView** again, or right-click it and click **Explode**.



 Animation of **Explode** and **Collapse** is not supported in multibody parts.

Hole Wizard - Placement of Holes

You can precisely define the position of the first point in a Hole Wizard sketch. Previously, the first point was placed wherever you clicked when selecting the face where you wanted to drill the hole.

When you activate the Positions tab, the first sketch point and a shaded preview of the hole follow the pointer until you click to place the hole. As you move the pointer about the screen, you can take advantage of sketch snaps and inference lines to place the point precisely.

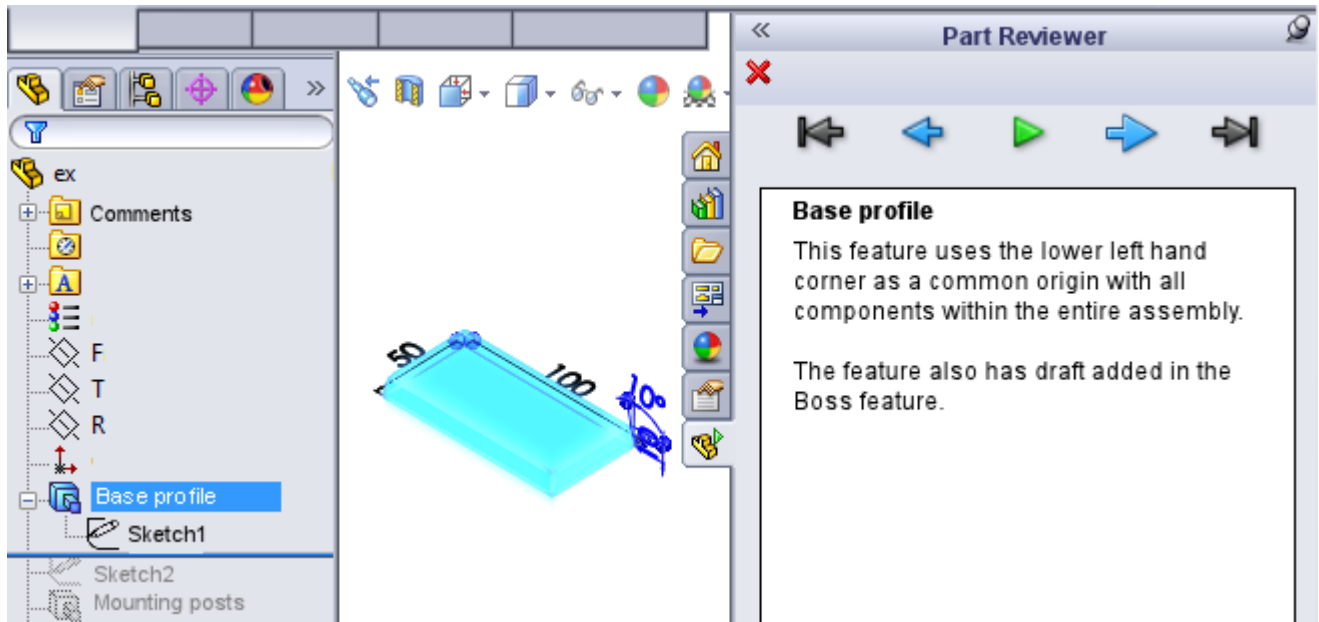
Part Reviewer

You can use Part Reviewer to review how parts are created feature-by-feature. Part Reviewer can help you learn best practices for creating various types of parts.

Part Reviewer sample parts are located in the SolidWorks Forum. These sample parts contain comments about their features to teach you how they were created and why they were used. You can also create your own parts and add your own comments.

To access Part Reviewer, click **Tools > Add-Ins**. Under **SolidWorks Add-ins**, select **SolidWorks Part Reviewer**.

The Part Reviewer tab  appears in the Task Pane.




Templates for Split Parts

In the Split and Save Bodies PropertyManagers, you can select an alternate part template to override the default template from **Tools > Options > System Options > File Locations**.

The selected template is applied to all the new part files you create during that **Split** or **Save Bodies** operation.

Previously, you could set a system option to prompt you for an alternate template whenever you created a part. Then you had to repeatedly specify that template (once for each body that you saved).

 In the Save Bodies PropertyManager, if you select **Create Assembly**, you can select an alternate assembly template.

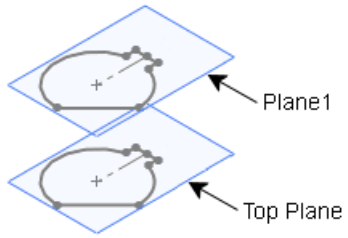
Reference Planes

Reference planes defined from the top plane are no longer rotated 180° from the top plane, and now behave the same as reference planes defined from the front or right planes.

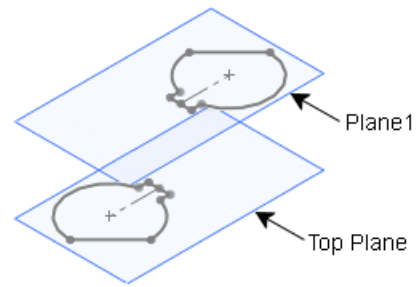
This change eliminates unexpected rotation of the model when viewing normal to the reference plane. It also eliminates flipping of copied sketches when you paste them onto the reference plane.

For example, suppose you create **Plane1** at an offset from **Top Plane**. Then you copy a sketch from **Top Plane** and paste it onto **Plane1**.

In SolidWorks 2012, the copy of the sketch is oriented as expected.



Previously, the copy of the sketch flipped 180°.



17

Routing

Available in SolidWorks Premium.

Enhancements include the ability to create spools, add splices to electrical routes, create custom routing templates, create BOMs with only routing components, and define dimensions in routing models to the end face of flanges.

This chapter includes the following topics:

- **Spools**
- **Splices**
- **Custom Route Property Templates**
- **Bills of Materials Options**
- **Dimensioning from Flanges**

Spools

Routing provides full support for spools.

You can create spools from existing pipe and tubing routes. Spools are sections of pipes or tubes and fittings that are manufactured separately and then connected during a final assembly or construction process. You can identify the spools uniquely by name, color, and line style, associate adjacent components with spools, create drawings of spools, import route specifications from P&ID files, generate BOMs, and export spool data to the PCF file format for ISOGEN applications.

Defining Spools

The command is available from the Piping and Tubing menus. This topic describes the steps from the Piping menu.

To define spools in a routing assembly:

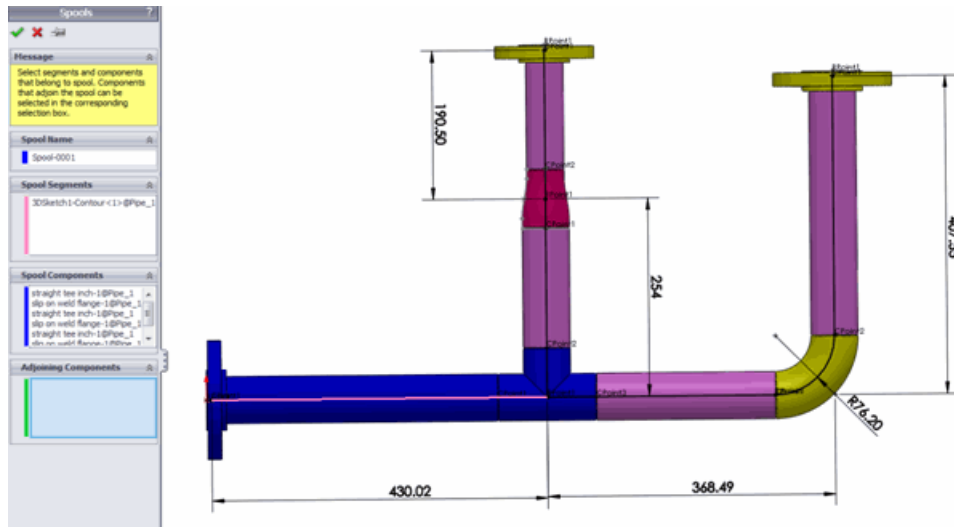
1. Click **Routing > Piping > Define Spools**.

You can also right-click the **Route** feature in the FeatureManager design tree and click **Define Spools**.

The Spools PropertyManager lists the name, color and line styles for the spool and its components.

- In the assembly, select the sketch entities that make up a spool.

Note: All segments and components in a spool must be contiguous. Gaps are not allowed in a spool. Also, you can only define one spool at a time.



- For adjoining components, click the PropertyManager, and in the assembly, select components that are external to the spool. These components are typically tees and elbows that will be connected to the spool in the manufacturing process.
 - Add or remove components by dragging and dropping the entities between Components and Adjoining Components in the PropertyManager.
 - Click .
- Spools are arranged in folders in the FeatureManager design tree of the route. A display state is also generated for each spool.

The creation of spool drawings, BOMs, import of P&ID data, and export of spool data to PCF is integrated with the piping and tubing functionality. You can select options for spools in each of the related PropertyManagers for these functions. For more information, see *SolidWorks Help: Spools*.

Splices

You can add splices to electrical routes.

Splices are typically modeled as cylinders containing a single connection point. You can add splices by splitting routes, by dragging and dropping splice components onto routes, and by adding splices from From-To lists. You can also create connector tables with splices, and add splices to BOMs.

Adding a Splice


This topic describes how to add a splice after splitting the route.


- Open a routing assembly in **Edit Route** mode.
- Right-click the sketch entity at the location where you want a split in the route and click **Routing > Electrical > Split Route**.

A split point is added where the sketch entities separate.

3. Select the split point and click **Routing > Electrical > Add a Splice**.
4. From the Design Library, drag a splice component onto the split point.
The splice supports various wire sizes. Select the configuration of the splice component to use based on the wire size.

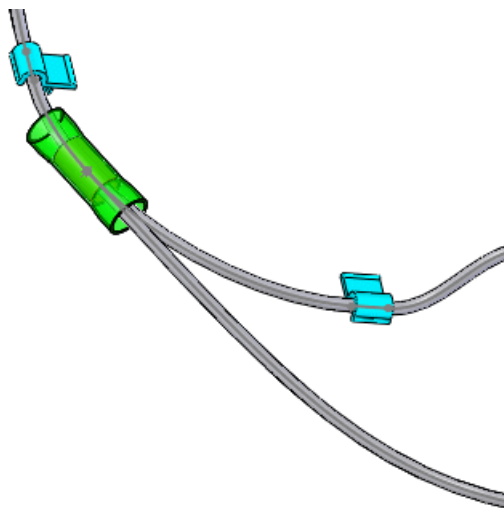


Add the command for **Add a Splice**  to the Electrical toolbar for easier access.

5. In the **Add Splice** PropertyManager, enter a name for the splice, and select the options to add a connector table and a BOM, if desired.
6. Click .

The splice is added to the FeatureManager design tree.

For more information, see *SolidWorks Help: Splices*.



Custom Route Property Templates

You can create route property templates that predefine parameters and defaults for a route. In a template you can specify details such as standard pipes and elbows and pipe size ranges. You can set parameters for route properties like standard lengths for pipes, whether to select weld gaps, and whether to insert standard couplings. You can add custom elbows. You can specify whether to always use elbows, always form bends, or prompt to select an elbow or bend along with the bend radii.

You can use the template to create an entire route or a single route segment.

When creating a route, the software automatically filters pipes and elbows based on the size and schedule defined in the template.

To create a route property template:

1. Click **Routing Tools > Routing Library Manager**. Then click the Route Properties tab.
2. In Select Properties Template, you can add a new template, copy an existing template or delete a template.

3. In Pipe Settings, you can select pipes from the Piping and Tubing database. You can set defaults for selected pipe including the schedule, a standard weld gap, pipe length, and whether to insert a standard coupling. You can also pick elbows from the database or create custom elbows.

When you build a new routing assembly, the entries from your default template appear. For example, it applies the default elbow at appropriate locations. If you have configured additional templates, you can select those from a list. You can also override a template and edit the entries.

For more information, see *SolidWorks Help: Routing Library Manager*.

Bills of Materials Options

You can manage routing components in BOMs. You can create BOMs that show only routing components. You can list all pipes and tubes individually in BOMs, or you can list all pipes and tubes of the same size as a single line item with a sum of the total pipe or tube length.

You can generate a BOM from the drawing view of an assembly. If an assembly includes any routing components, the Bill of Materials PropertyManager includes an extra section for Routing Component Grouping.



These options are available without the Routing add-in.

Selecting **Show only routing components in BOM** generates a BOM with only pipe, tube, wire and other routing components. No other types of components are included.

Selecting **Group pipes or tubes with the same diameter and schedule** lists all the pipes and tubes of the same size as one item, with the sum of the lengths and schedules for those components. For example, instead of showing that an assembly contains 6 units of a certain pipe that are 10 inches long and 8 more pipes that are 12 inches long, the BOM displays one item for the size pipe with a total length of 156 inches.

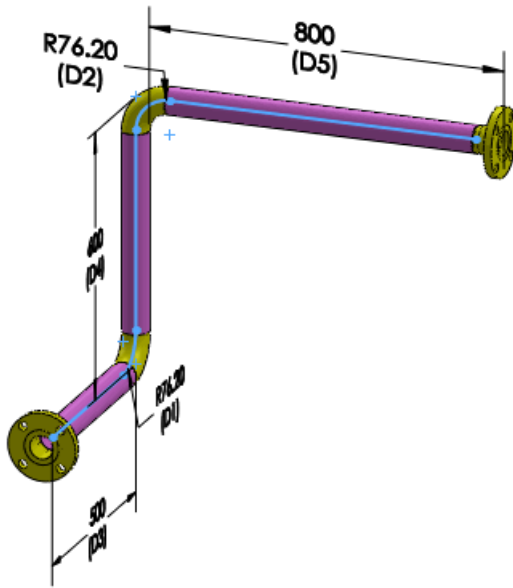
Selecting the options in different combinations produces different results. For example:

- Selecting both options produces a BOM that lists only routing components, and combines pipes and tubes of each size.
- Selecting the first option but not the second produces a BOM that lists only routing components, but lists all pipes, tubes and wires individually.
- Selecting the second option but not the first, produces a BOM that includes all the non-routing components in the assembly, but combines the pipe and tubes of each diameter and schedule.

Dimensioning from Flanges

In route models, you can define driving dimensions that start with the end face of a flange. Adding a dimension between a flange face and a route does not cause the sketch to be over defined.

In previous releases, you could not add dimensions between the end face of a flange and a route.

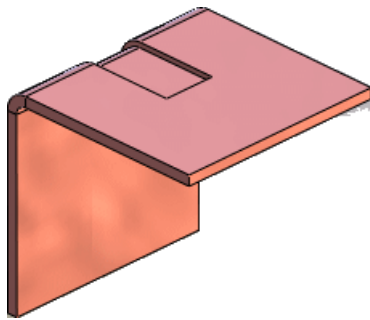


This chapter includes the following topics:

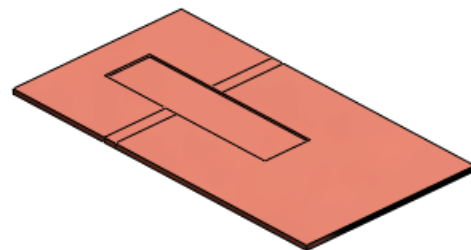
- **Cuts Across Bends**
- **Edge Flanges**
- **Flat Patterns**
- **Forming Tools**
- **Swept Flanges**

Cuts Across Bends

You can unfold a part that has cuts across bends when the cuts do not go through the entire part. This is helpful when making flat patterns for manufacturing.



Part with a cut across a bend. The cut does not go through the entire part.



Unfolded part

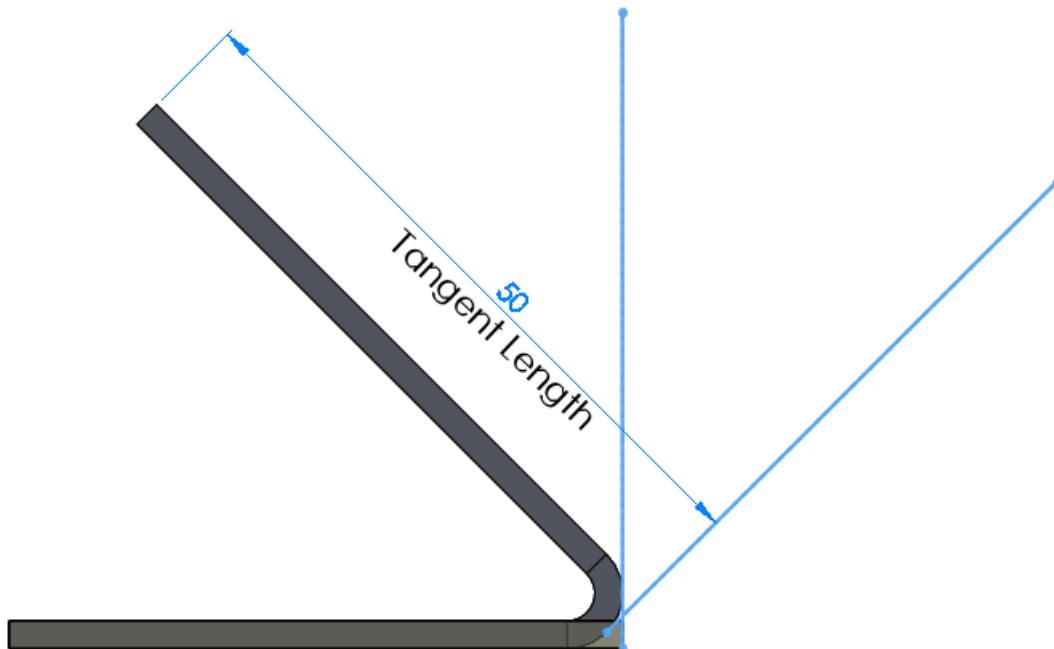
Edge Flanges

New options in the Edge-Flange PropertyManager let you create edge flanges based on tangent distance in flange length and create flange positions that are tangent to the adjacent side face attached to the selected edge. Additionally, when you create a flange length as **Up To Vertex**, you can create the flange parallel to the base flange as well as normal to the flange plane.

Tangent Bend Flange Length

You can set an edge flange length to **Tangent Bend**. This option is valid for bends that are greater than 90° and lets you use the tangent length of a flange as the basis for the length calculation. **Tangent Bend** lets you type in the tangent length without using additional calculations.

In the Edge-Flange PropertyManager, under **Flange Length**, click **Tangent Bend** .

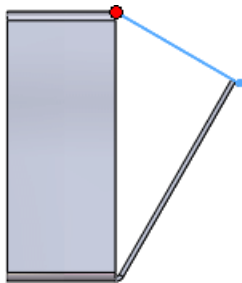


Up to Vertex Flange Length

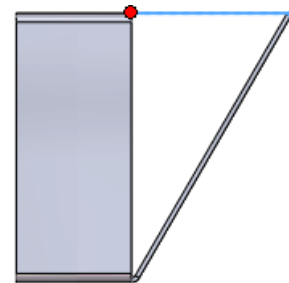
If you set an edge flange length to **Up To Vertex**, you can create it normal to the flange plane or parallel to the base flange.

In the Edge-Flange PropertyManager, under **Flange Length**, set **Length End Condition** to **Up To Vertex**, then select **Normal to flange plane** or **Parallel to base flange**.

Example of edge flange lengths using the options for **Up To Vertex**:



Normal to flange plane (existing functionality): The red vertex is coincident to the end face of the edge flange. (A sketch line is displayed for clarity.)



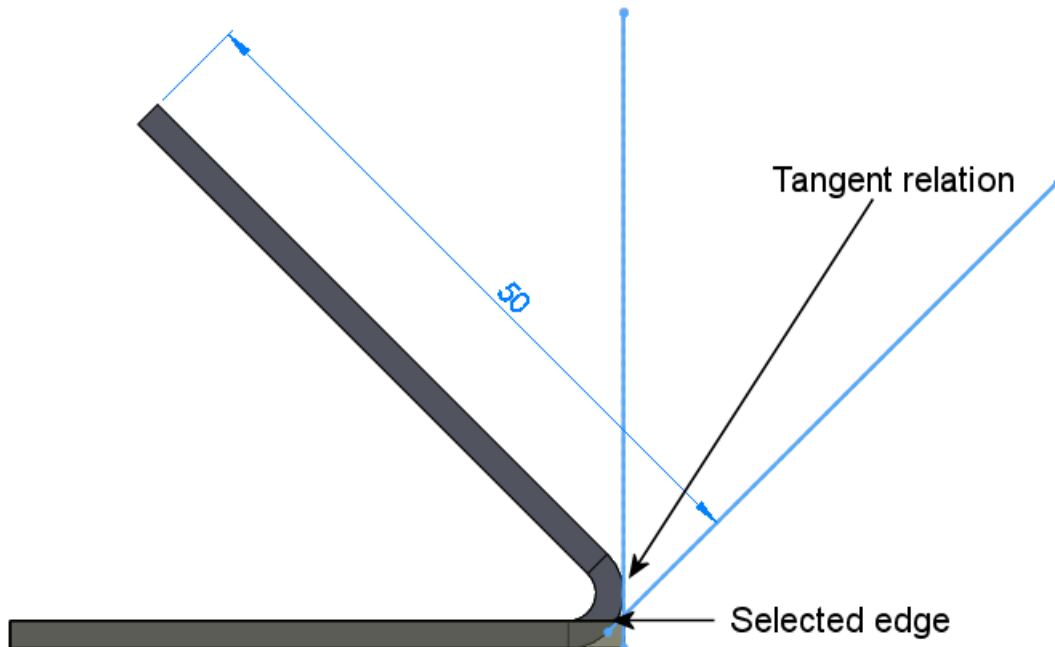
Parallel to base flange: The red vertex passes through a plane that is parallel to the face of the base flange. (A sketch line is displayed for clarity.)

Tangent to Bend Flange Position

You can set an edge flange position to **Tangent to Bend**. This option is valid for all flange length options and for bends that are greater than 90°.

The flange position will always be tangent to the side face attached to the selected edge, and the flange length will always maintain the exact length.

In the Edge-Flange PropertyManager, under **Flange Position**, click **Tangent to Bend**

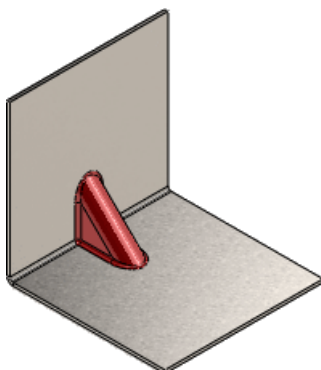


See *SolidWorks Help: Edge-Flange PropertyManager*.

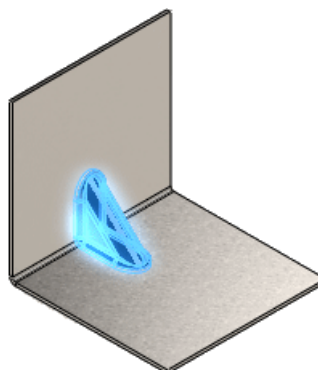
Flat Patterns

You can exclude faces from flat patterns, which is helpful when faces interfere with bends.

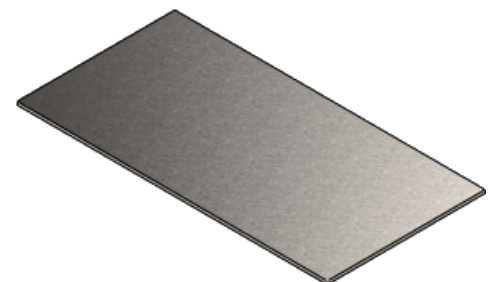
In the FeatureManager design tree, right-click **Flat-Pattern**  and click **Edit Feature** . In the graphics area, for **Faces To Exclude** select faces that you do not want in the flat pattern. You must select the front and back of each face that you want to exclude.



Original part



Faces to exclude - select the front and back of each face



Flat pattern

Forming Tools

Forming tools include several enhancements that improve the workflow.

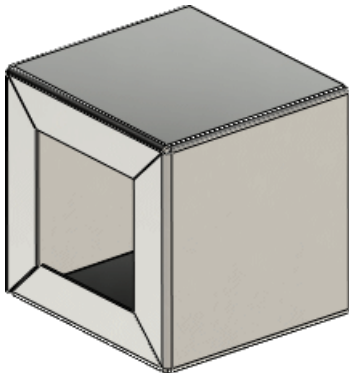
You can:


- Select configurations of the forming tools or target parts.
- Change the placement face where you locate the forming tools.
- Create links to forming tools, so that if you change the original forming tool, the changes propagate to target parts where you use the forming tool.
- Replace existing forming tools with different forming tools.
- Set the appearance of placement sketches in flat patterns.
- Assign punch IDs for use in punch tables.

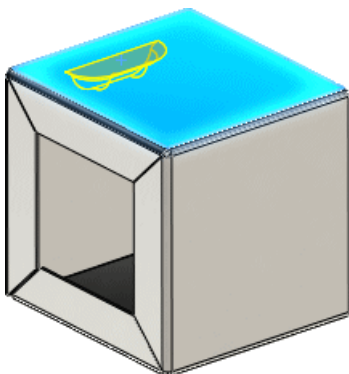
Inserting Forming Tools

When you insert forming tools, the Form Tool PropertyManager provides an easier interface for tasks such as selecting the placement face, flipping the tool, and choosing configurations.

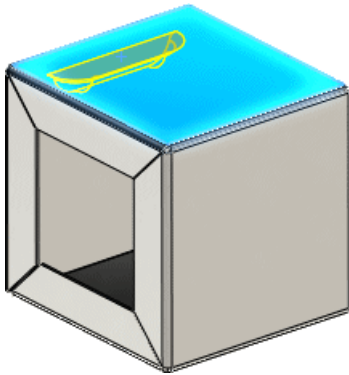
1. Open `install_dir\samples\whatsnew\sheetmetal\cover.sldprt`.
A sheet metal part opens.



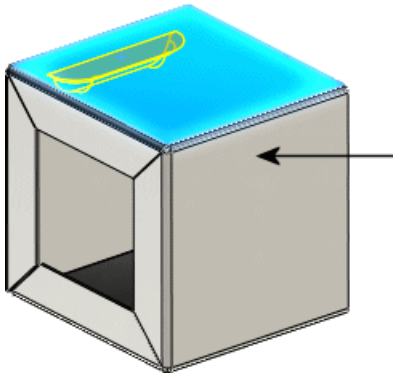
2. In the Task Pane, click the Design Library  tab.
3. Navigate to `forming tools/louvers`.
4. Drag **sample_louver** to the face shown.



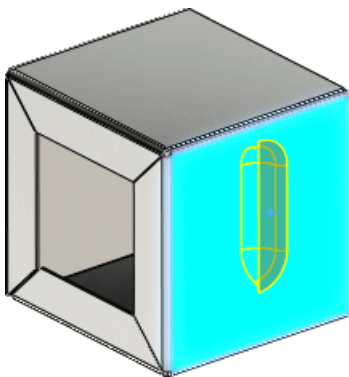
- In the PropertyManager, under **Configurations**, in **Form Tool Configuration**, select **W20**.
A longer louver appears.



- In the graphics area, select the face in the approximate area as shown.

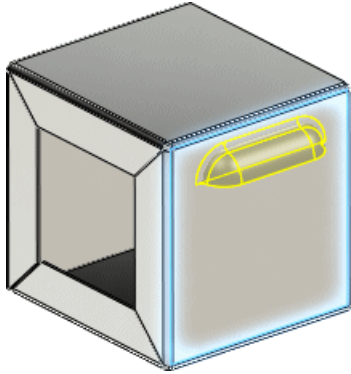


The forming tool moves to the new face and **Placement Face** updates in the PropertyManager.



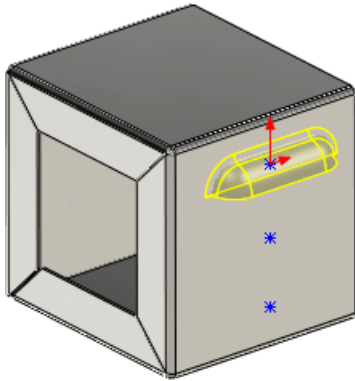
- In the PropertyManager:


- a) Under **Rotation Angle**, set the value to 270, then click **Flip Tool**.

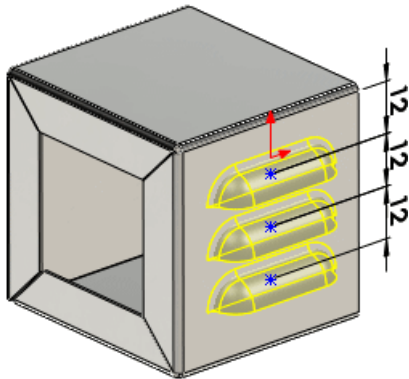


- b) Click the Position tab.

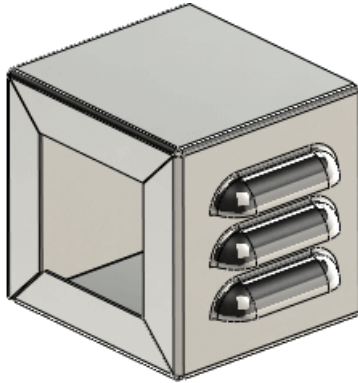
8. In the graphics area, select the areas shown to insert two more instances of the forming tool.



9. Click **Smart Dimension**  (Sketch toolbar) or **Tools > Dimensions > Smart**, and dimension the points as shown.



10. Click ✓ twice.



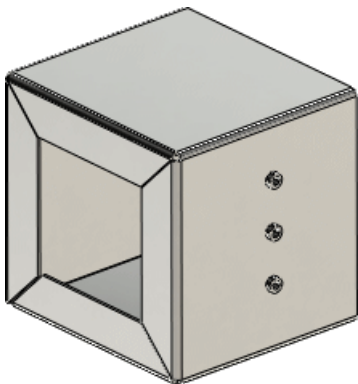
11. Keep this part open for the next procedure.

See *SolidWorks Help: Applying Forming Tools to Sheet Metal Parts*.

Replacing Forming Tools

After you insert a forming tool in a model, you can replace the forming tool with another one.

1. In `cover.sldprt`, in the FeatureManager design tree, right-click **sample_louver1(W20)** and click **Replace Form Tool**.
2. In the dialog box, navigate to `install_dir\Documents and Settings\All Users\Application Data\SolidWorks\SolidWorks 2012\design library\forming tools\embosses\extruded hole.sldprt` and click **Open**.
3. In the PropertyManager, click ✓.
The louver is replaced with the hole.










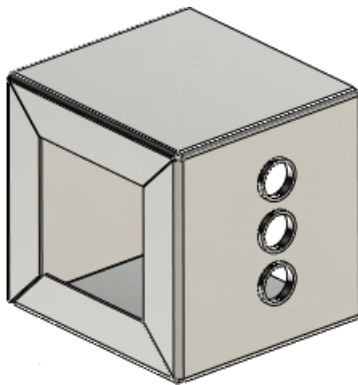
4. Keep this part open for the next procedure.

See *SolidWorks Help: Replace Form Tool PropertyManager*.

Linking Forming Tools

You can create links to forming tools, so that if you change the original forming tool, the changes propagate to target parts where you use the forming tool.


1. In `cover.sldprt`, right-click **extruded hole1(Default)**  and click **Edit Feature** .
2. In the PropertyManager:
 - a) Under **Link**, make sure **Link to form tool** is selected.
This sets the link between the forming tool and its original part.
 - b) Click .
3. In the Design Library, navigate to `forming tools\embosses`.
4. Right-click **extruded hole** and click **Open** .
The emboss opens.
5. In the FeatureManager design tree, right-click **Boss-Extrude1**  and click **Edit Sketch** .
6. In the graphics area, double-click the diameter dimension, 2.
7. In the Modify dialog box, type 8 and click .
8. Exit the sketch.
9. Save and close the part.
10. In `cover.sldprt`, click **Rebuild** (Standard toolbar) or **Edit > Rebuild**.
The part rebuilds with the emboss with a larger radius.






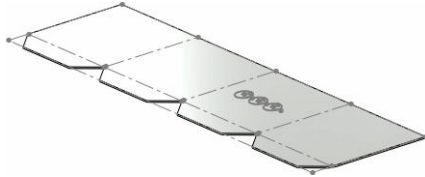
See *SolidWorks Help: Form Tool Feature PropertyManager*.



Transforming the Placement Sketch

You can choose how to display forming tools in flat patterns.

1. In `cover.sldprt`, in the FeatureManager design tree, right-click **extruded hole1(Default)** and click **Edit Feature** .
2. In the PropertyManager, under **Flat Pattern Visibility**:
 - a) Select **Override document settings** and **Show profile**.

- b) Clear any other options.
 - c) Click .
 3. In the FeatureManager design tree, right-click **Flat-Pattern1**  and click **Unsuppress** .
- The forming tool's profile (placement sketch) appears in the flattened part.




4. Right-click **Flat-Pattern1**  and click **Suppress** .
5. Keep this part open for the next procedure.

See *SolidWorks Help: Form Tool Feature PropertyManager*.





Creating Punch IDs

You can add punch IDs to forming tools for easier identification in punch tables.

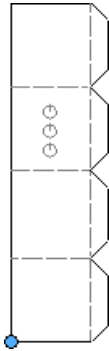
1. In the **Design Library**, right-click **extruded hole** and click **Open** .
The emboss opens.
2. Click **File > Properties**.
3. In the dialog box, on the Configuration Specific tab:
 - a) In **Property Name**, select **PunchID**.
 - b) In **Type**, select **Text**.
 - c) For **Value / Text Expression**, type `P1`.
 - d) Click **OK**.
4. Save the part.

Inserting a Punch Table


Punch tables are available with the flat pattern view of sheet metal parts and contain the form features and library features used in sheet metal bodies.

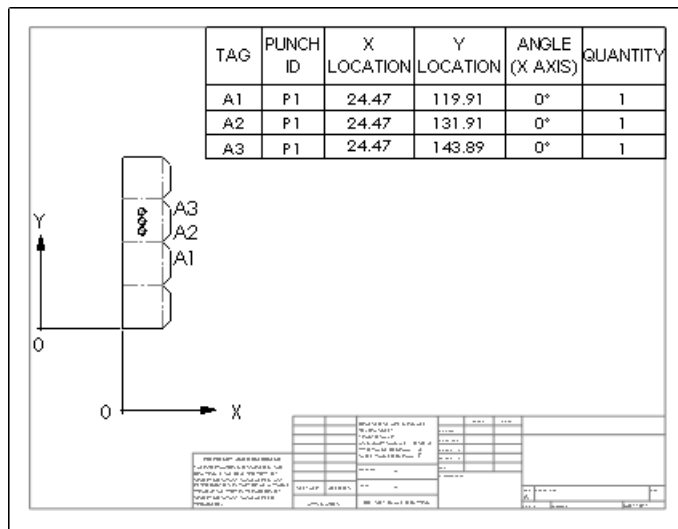
1. In `cover.sldprt`, click **Make Drawing from Part/Assembly**  (Standard toolbar) or **File > Make Drawing from Part**.
2. In the New SolidWorks Document dialog box, click **Drawing** , then click **OK**.
3. In the Sheet Format/Size dialog box, click **OK**.
4. In the View Palette, drag the **Flat pattern** view onto the sheet.
5. In the PropertyManager, click .
6. Click **Punch Table**  (Table toolbar) or **Insert > Tables > Punch Table**.
7. In the graphics area, click:

a) The point as shown for **Origin**.



b) Any face for **Edges/Faces/Features**.

8. In the PropertyManager, click .
9. In the graphics area, click to place the punch table.

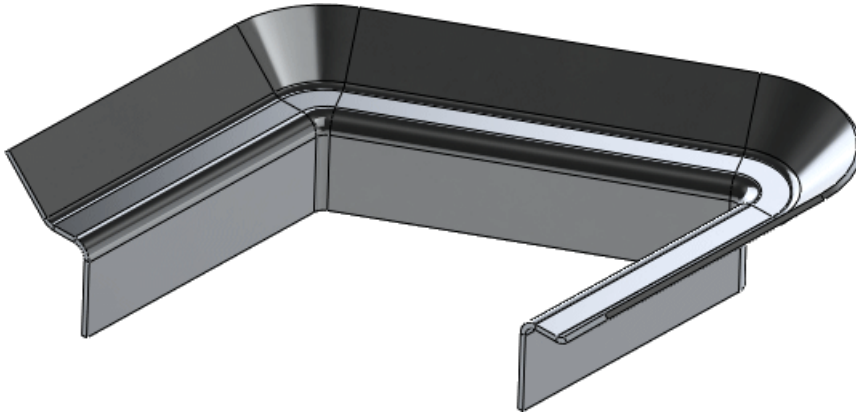


See *SolidWorks Help: Punch Tables*.

Swept Flanges

You can create compound bends in sheet metal parts using the **Swept Flange** tool.





The **Swept Flange** tool is similar to the **Sweep** tool; you need a profile and path to create the flange. To create a swept flange, you need an open sketch as the profile, and an open profile path or a series of existing edges in a sheet metal part.

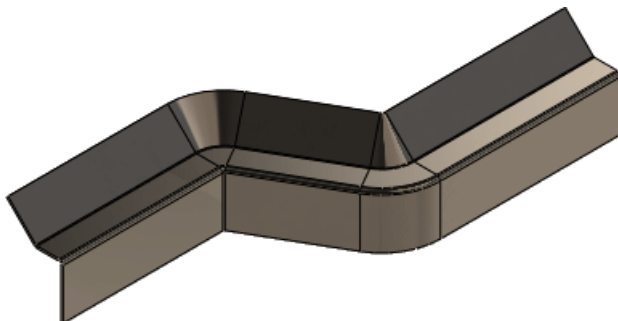


Any cuts, holes, chamfers, or fillets on the bend region of the swept flange do not appear in the flat pattern.

1. Open `install_dir\samples\whatsnew\sheetmetal\swept_bend.sldprt`.
A part with two sketches opens.



2. Click **Swept Flange**  (Sheet Metal toolbar) or **Insert > Sheet Metal > Swept Flange**.
 3. In the FeatureManager design tree, select:
 - a) **Sketch2** for **Profile** .
 - b) **Sketch1** for **Path** .
 4. Click .
- The swept flange is complete.



See *SolidWorks Help: Swept Flange and Swept Flange PropertyManager*.

19

Simulation

- SolidWorks Simulation is available in SolidWorks Premium.
- SolidWorks Simulation Professional and SolidWorks Simulation Premium are available as separately purchased products that can be used with SolidWorks Standard, SolidWorks Professional, or SolidWorks Premium.

This chapter includes the following topics:

- **Beams**
- **Connectors**
- **Parameters Interface**
- **Motion Optimization**
- **Creating a Design Study with Motion Parameters**
- **Optimization of a Catapult with Motion Features**
- **Nonlinear Studies**
- **Shells**

Beams

Enhancements include availability of beams in linear dynamics studies, imprinting of actual cross-sections for bonding connections, evaluation of torsional constants, and more realistic rendering of beam profiles.


Beams in Linear Dynamic Studies

Beam elements are supported in Linear Dynamic studies, which include: Modal Time History, Harmonic, Random Vibration, and Response Spectra analyses. Use beams to simplify problems with weldments or structural members, improve simulation performance, and reduce the file size of results.

Performing Dynamic Analysis with Beam Models

To perform a linear dynamic analysis for models that include beams:

1. Create a Linear Dynamic study, and select one of the study options depending on the loading and response parameters you want to access after running the study.
You can also copy an existing frequency study to a new dynamic study.
2. Set the frequency and dynamic properties of the study.
3. Define material. You can apply only **Linear Elastic Isotropic** material type to beams.
4. Define restraints. **Fixed Geometry, Immovable (No translation),** and **Use Reference Geometry** are available for beams.
5. Define Loads. You can choose from the following options:

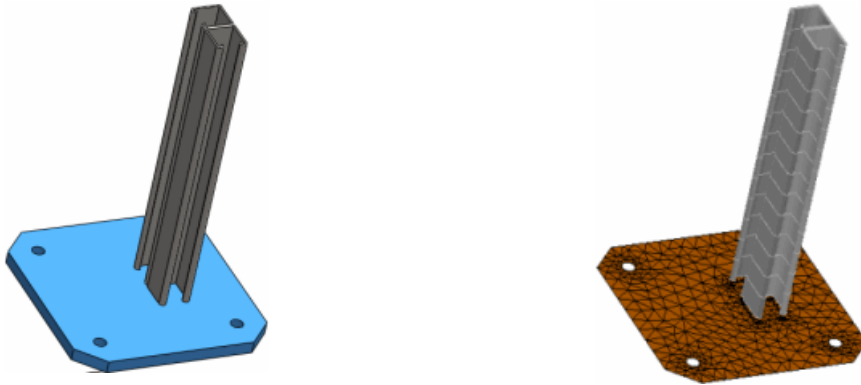
- **Time-dependent force or torque** Available for modal time history analysis studies. Select beam joints or the entire beam length to apply force or torque values.
- **Frequency-dependent force or torque** Available for harmonic and random vibration studies. Select beam joints or the entire beam length to apply force or torque values.
- **Uniform and selected base excitations**  For Random Vibration and Response Spectra Analysis, you can apply base excitation only in the global X, Y, Z directions.
- **Initial conditions** Available for modal time history analysis studies. Displacements, velocities, or accelerations at time = 0. You can apply initial conditions to beam joints or the entire beam length.

6. Define damping.
7. Define contact. You can select **Bonded** or **Allow Penetration**.
8. Mesh the model and run the study.
9. View the results:
 - You can generate time history graphs at selected nodes. Right-click **Results**, and select **Define Response Graph**.
 - You can plot stresses (axial, bending, highest axial and bending, torsional, and shear), displacements, velocities and accelerations for a specified solution step, or extreme values across all solution steps. Right-click **Results**, and select from the plot options.

Beam Bonding

When beam joints are connected to a neighboring face of a solid or shell, the beam's actual cross-section is meshed and imprinted on the solid or shell face, creating a more accurate bonding connection.

During meshing, elements are created in the imprinted beam section area. The end beam joint is connected to all elements inside the imprinted section area forming a more realistic representation of the joint.



If the beam's cross-section is partially touching on a solid or shell face, only the touching part of the cross-sectional area gets imprinted.

Enhanced algorithms automatically bond beam joints to touching solids or shell faces. When you set the **Global Contact** to **Bonded**, the software automatically bonds: (a) beam end joints to touching faces, and (b) beams that act as stiffeners to planar or curved surfaces that have touching geometries, or are situated within reasonable clearance.

Torsional Constants for Beams

SolidWorks Simulation computes the torsional constant and shear factor entries for beam sections. The shear center line of all beams is shown, along with the neutral axis.

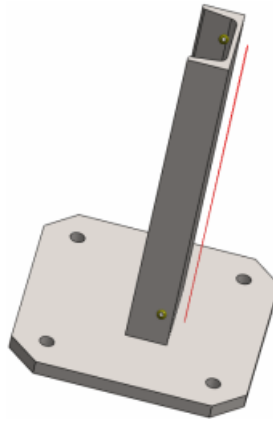
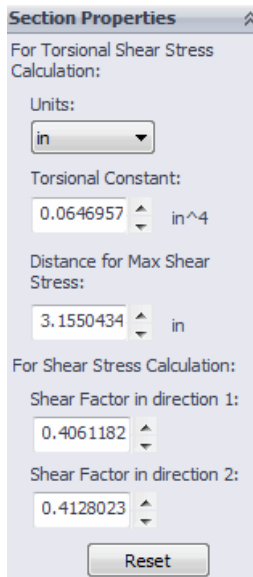
For torsional shear stress calculation, the program calculates the torsional constant, and distance from the center of the section to the point of maximum torsional shear for all beam cross-sections. For shear stress calculation, the program calculates the two shear factors that account for the nonuniform shear stress distribution across the beam's cross-section.

To view the section properties of a beam, right-click the beam definition and select **Edit Definition**. The torsional constant, distance for maximum shear stress, and shear factors are listed under **Section Properties** in the **Apply/Edit Beam** PropertyManager.



You can override the beam section properties calculated by the program with user-defined values.

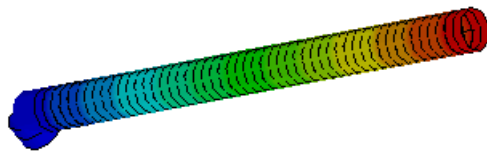
To view the shear center line of a beam, right-click **Joint group** and select **Edit**. Under **Results**, select **Display shear center**. A red line shows the shear center of beams in the graphics area.



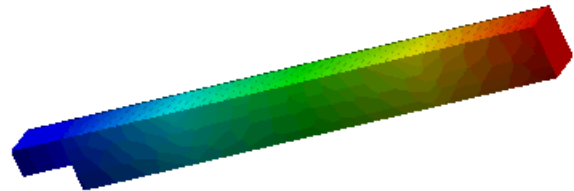
Beam Results

Beam mesh and result plots can now be shown on actual beam geometry. eDrawings supports the new plots.

In previous releases, mesh and result plots for beam studies were displayed on cylinders and not the actual beam geometry. You now have the option to display the results either on cylinders or on the beam geometry.



Previous releases of SolidWorks Simulation



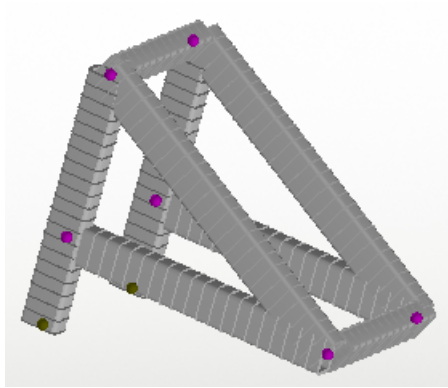
SolidWorks Simulation 2012

Displaying Mesh on Beam Geometry

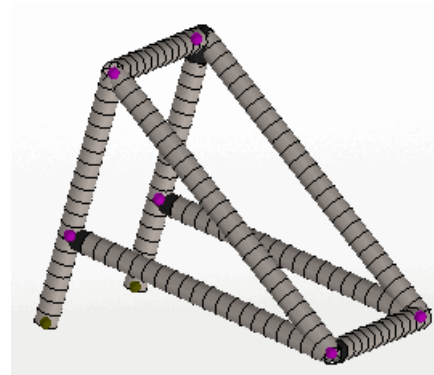
To display the mesh on actual beam geometry as opposed to cylinders:

1. In a Simulation study, click **Simulation** > **Options**.
2. On the Default Options tab, click **Mesh**.
3. Select **Render beam profile**, and click **OK**.

- In the Simulation study tree, right-click **Mesh** and click **Show Mesh**.




Render beam profile selected



Render beam profile cleared

Displaying Stress Plots on Beam Geometry

To display a stress plot on actual beam geometry:

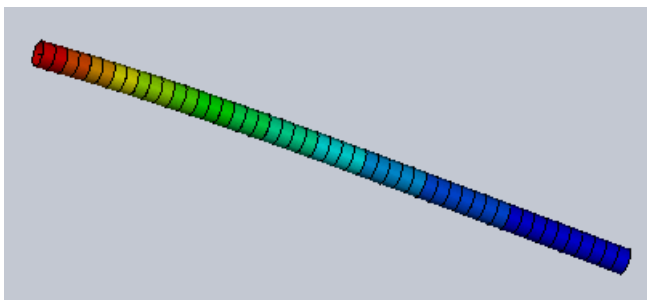
- In the Simulation study tree, right-click **Results** and click **Define Stress Plot**.
- In the Stress Plot PropertyManager, under **Display**, select **Render beam profile**.
- Click .



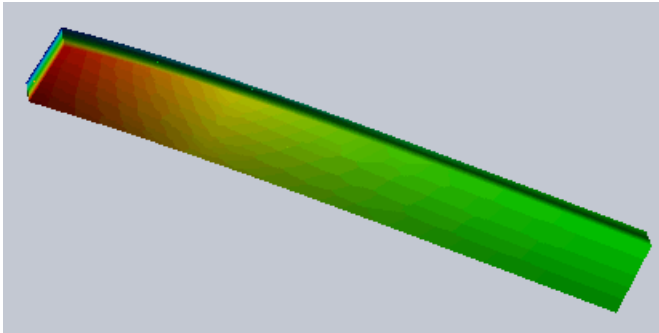
In the Stress Plot PropertyManager, under **Display**, in **Beam stress, Highest Axial and Bending** is now replaced by **Axial and bending**, which shows the combined uniform axial stresses and axial stresses resulting from bending. These plots now display the stress distribution throughout the beam, and no longer display a single color. In previous releases, a single color represented the maximum value across the beam's cross-section.

In stress plots, nodal as well as element results are available for beams. As before, element plots display on cylindrical cross sections.

If the **Render beam profile** option is cleared in the Stress Plot PropertyManager, the software assigns the *highest* stress value for the element and axial components. For example, you can select from **Highest axial and bending**, **Highest Bending in local direction 1**, or **Highest Bending in local direction 2**.



If the **Render beam profile** option is selected in the Stress Plot PropertyManager, bending stresses *vary* across the cross-section, as well as along the axis of the beam. For example, you can select from **Axial and bending**, **Bending in local direction 1**, or **Bending in local direction 2**, which indicates that the values vary throughout the cross-section.

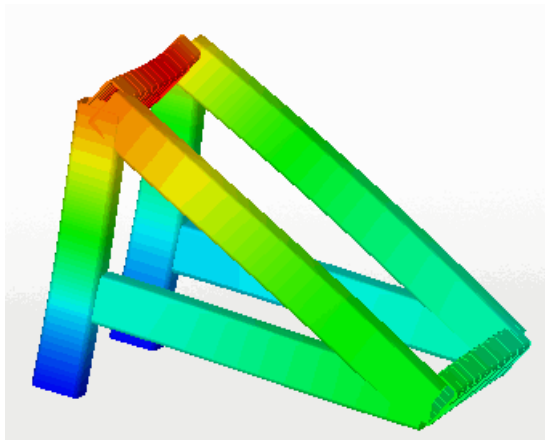


This option can be slow in studies with many beam elements. To improve performance in these instances, limit the rendering of the beam to the final results.

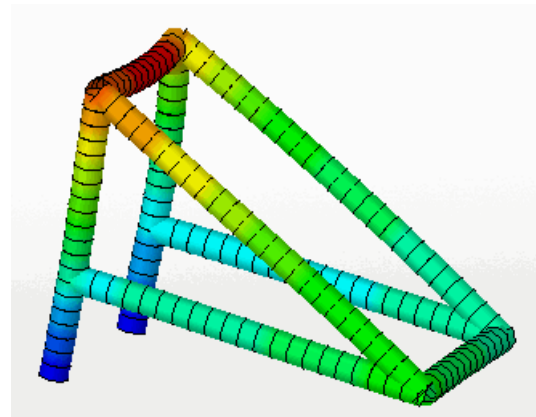
Displaying Displacement Plots on Beam Geometry

To display a displacement plot on actual beam geometry:

1. In the Simulation study tree, right-click **Results** and click **Define Displacement Plot**.
2. In the Displacement Plot PropertyManager, under **Advanced Options**, select **Render beam profile**.
3. Click .



Render beam profile selected



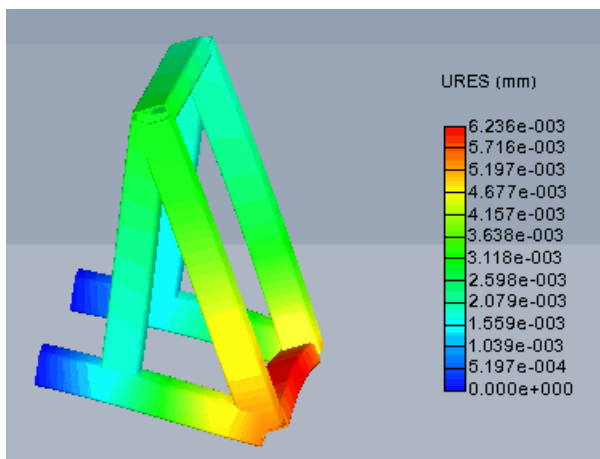
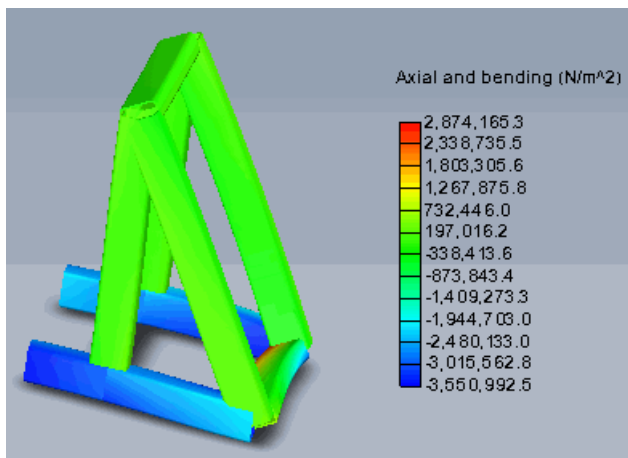
Render beam profile cleared

Displaying Beam Results in eDrawings

You can save beam results on actual beam geometry and display them in eDrawings as they appear in SolidWorks Simulation. Previously, beam results displayed on cylinders in eDrawings.

To save beam results on actual beam geometry and display them in eDrawings:

1. Create the beam plots in SolidWorks Simulation.
2. In the Simulation study tree, right-click **Results** and click **Save All Plots as eDrawings**.
3. In the Save As dialog box, enter a **Save in** location and **File name**. Click **Save**.
4. Open the file in eDrawings.

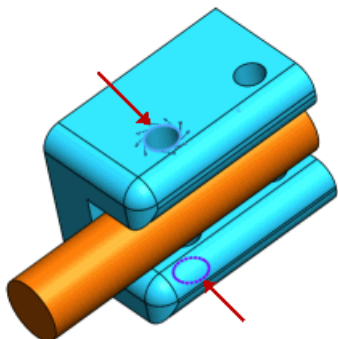


Connectors

Enhancements for connectors include improved accuracy for the edge weld connector, support of all connectors (except bolt and elastic support) for linear dynamic studies, and creation of a bolt or pin connector on the same component.

- The calculation of the weld sizes for the edge weld connector is more accurate due to the improvement of the bonding algorithm between shell edges and faces.
- All connectors, except the bolt connector and the elastic support, are supported in Linear Dynamic studies.

- You can import a frequency study with connectors to a new linear dynamic study.
- You can create a pin or a bolt connector by selecting cylindrical faces or edges of the same component.



Parameters Interface

The **Parameters** dialog box has an improved interface that allows you to create and link parameters to components from Simulation or Motion studies features.

Linking a Simulation (or Motion) Feature to a Parameter

You can now link parameters to Simulation or Motion study features from a unified interface. Use the parameterized Motion features in design studies to perform optimization or evaluate specific scenarios of your design.

To link a parameter to a Simulation or Motion study feature:

1. Click **Insert > Design Study > Parameters**, or click the Design Study tab and under **Variables**, select **Add Parameter**.
2. In the **Parameters** dialog box, under **Name**, type a name for the parameter.
3. Under **Category** select:
 - **Simulation**, to link to a Simulation feature, or
 - **Motion**, to link to a Motion feature.
4. Click the Simulation or Motion study tab and select the feature that you want to link to the parameter from the study tree.



Motion features that can be linked to parameters include: motors, springs, dampers, contact, and study properties. You can only link one motion feature to a parameter.

The selected feature appears in **Item Selection**. A list of valid components appear in **Component Link**.

5. In **Component Link**, select one component to link to the parameter.
6. Under **Properties**, type a numeric value in the **Value** field, or select **Expression** and enter an appropriate expression for the parameter type.

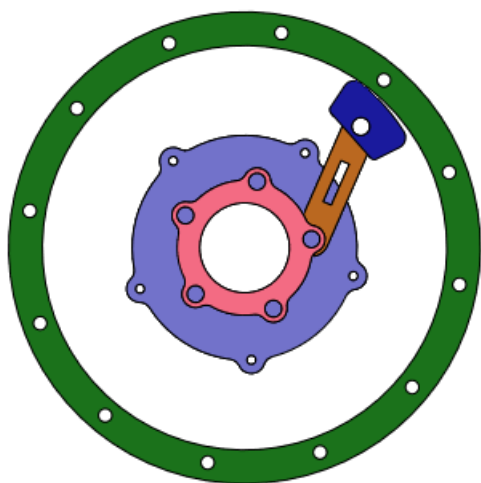
You can enter comments in **Comment (optional)**. Your comments appear in the **Comment** field under **Parameters**.

7. Click **Apply**.
An asterisk appears under **Linked** to show that the highlighted parameter is now linked.
8. You can continue adding new parameters or click **OK** to exit the dialog.

Motion Optimization

You can create a SolidWorks Simulation design study to optimize motion studies features.

- From the Parameters dialog box you can create and link variables to features from Motion Analysis type studies.
- In a design study, you can specify constraints and goals that are linked to motion study results through sensors. You can perform an optimization or evaluate specific design scenarios.
- For each design iteration, you can view the motion animation within the design study. For any iteration of the design study, you can view detailed motion results in the motion study.



Creating a Design Study with Motion Parameters


With a Design study, you can evaluate several design scenarios based on parameterized features from motion analysis-type studies, and also optimize your design based on defined goals.

1. Create a motion analysis-type study that includes features to parameterize in a design study (i.e., springs, dampers, motors, contact, or force).
2. Click **Insert** > **Design Study** > **Parameters** to create Motion-type parameters, and link them to the desired features of the motion analysis-type study.

Parameters are added as variables in the Design study.

See also [Linking a Simulation \(or Motion\) Feature to a Parameter](#) on page 152.

3. Define sensors to add as **Constraints** or **Goals** in the Design study by doing on the following:

- Attach sensors to result plots of the motion analysis-type study. In the Results/Sensor PropertyManager, select **Create Sensor for Design Study/Optimization**, or
 - Add sensors by right-clicking **Sensors** in the FeatureManager design tree and selecting **Add Sensor**. Under **Sensor Type**, select **Motion Data**. Select the motion study and the motion results plot to attach a sensor.
4. Create a Design study.
 5. Add parameters as **Variables**. Specify a range, or set discrete values for the variables. Variables change values with each iteration of an evaluation or optimization design scenario.
 6. Specify **Constraints** through predefined sensors, or add new sensors to specify conditions that your design must satisfy. For evaluation design studies, use constraints to monitor results.
 7. Click **Run**, or define **Goals** for an Optimization study.
 8. Specify **Goals** through predefined sensors, or add new sensors to specify design objectives. If you define multiple goals, you can set the relative weight of each goal.
 9. After you run the design study, on the **Results View** tab, view the values of variables, constraints, and goals for each scenario. The optimal scenario column is green, scenarios that violate the constraints are red.
 10. Click **Play**  to play back motion results for each iteration.

Optimization of a Catapult with Motion Features

You define the optimal spring constant of a catapult that launches a projectile to hit a target with the minimum impact force.

You create a design study to find the required spring constant of a catapult designed for launching a projectile that hits a target at a certain distance.



To open the model, go to
install_dir\samples\whatsnew\Optimization\catapult.sldasm.

 You need to install the Motion Simulation add-in to complete the tutorial.

Review the Motion Analysis study features:

- Linear spring
- Gravity load
- Two body contact sets
- Two result plots for tracking the contact force between the projectile and the target, and the X - component of the projectile's linear displacement.
- Two sensors that track the motion analysis result plots.

 To create sensors that track motion results, in the **Sensor/Results** PropertyManager, select **Create Sensor for Design Study/Optimization**.

Defining the Design Study Variable

You define the spring constant as a parameter, and you add it as a variable in the design study.

1. Create a new Design Study.
2. On the **Variable View** tab, expand **Variables**, and select **Add Parameter**. In the **Parameters** dialog box:
 - a) Under **Name**, type a name for the parameter, i.e. *Spring_Constant*.
 - b) Under **Category**, select **Motion**.
 - c) In the Motion study tree, double-click **LinearSpring2**.
The spring feature is highlighted in **Item Selection**, and all available components are listed in **Component Link**.
 - d) Select the component **Spring Constant (Spring)** to link to the parameter.
 - e) For **Value**, type 17500.
 - f) Click **Apply** and **OK**.
An asterisk under **Linked** shows that the selected component is now linked to the parameter.
3. Add the parameter under **Variables**.
4. Select **Range with step**. For **Min**, type 17400 (N/m), for **Max**, type 17600 (N/m), for **Step**, type 25 (N/m).


Defining Constraint and Goal


You define the sensor that tracks the X-component of the projectile's trajectory as a constraint. As a goal, you want to minimize the impact force of the projectile when it hits the target.

1. On the **Variable View** tab, in **Constraints**, select the displacement sensor. For condition, select **Is between**. For **Min**, select -0.1 and for **Max**, 0.1.
This condition enforces the limits for the X-component of the projectile's displacement, so that it can hit the target.
2. In **Goals**, select the force sensor. For condition, select **Minimize**.
As a goal, you want to minimize the contact force of the projectile when it hits the target. If you do not define any goals, you can run the design study as an evaluation design study. There is a total of nine active design scenarios

You are now ready to run the optimization study.

Running the Optimization

1. Select **Optimization** and click **Run**.
The program invokes the Motion Analysis study at each iteration, and reports the values for each iteration.
2. On the **Results View** tab, select **Scenario1** and click **Play from Start** . The motion animation appears in the graphics area. Scenarios that do not satisfy the constraints are highlighted in red. In this scenario, the projectile misses the target.

3. Select **Optimal (4)**, which is the optimal scenario highlighted in green, and click **Play from Start** .

For the projectile to hit the target, the required spring constant of the catapult is 17475 N/m.

	Optimal (4)
Spring	17475 N/m
Displacement6	-0.089413
Force2	119.206823



You can switch to the Motion Analysis study tab, to view motion result plots corresponding to the active design scenario.

Nonlinear Studies

Improvements in the areas of contact convergence and I/O data handling enhance the overall quality of nonlinear analysis.

- Performance improvements ranging from 15% to 30%, for most studies, in reduction of overall solution time. Some studies have shown over 50% performance improvement.
- The surface-based option for bonding is extended to nonlinear studies.

To apply surface-based bonding, in the Nonlinear - Static dialog box, under Incompatible bonding options, select **More accurate (slower)**.

- The convergence and solution accuracy of the surface-based contact algorithm are improved. As a result, nonlinear problems with the following features are improved:
 1. Surface-based bonding contact and connectors
 2. No penetration contact with friction
 3. Combination of shrink-fit and no penetration contact (differentiate the two types of contact in the database)

Shells

Enhancements for shells include:

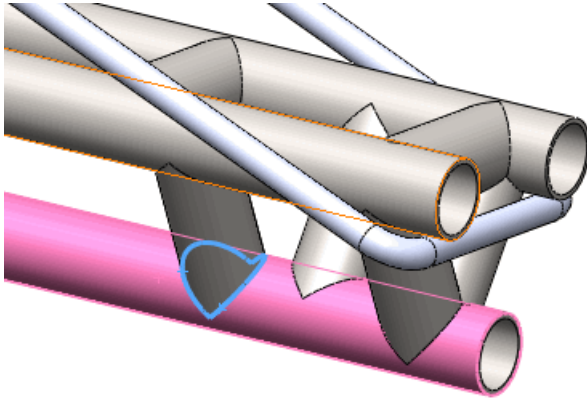
- Automatic detection of contact pairs between touching shell edges connected to faces is expanded to include combinations with non-planar faces and circular edges.
- Improved accuracy of the bonding algorithm between shell edges and solid faces for static, frequency, buckling, and nonlinear studies.
- Transfer of temperature and pressure data from SolidWorks Flow Simulation.

Auto-bonding Shell Edges to Faces

The automatic detection of touching shell edges connected to solid or shell faces is improved to include more combinations with non-planar faces and circular shell edges.

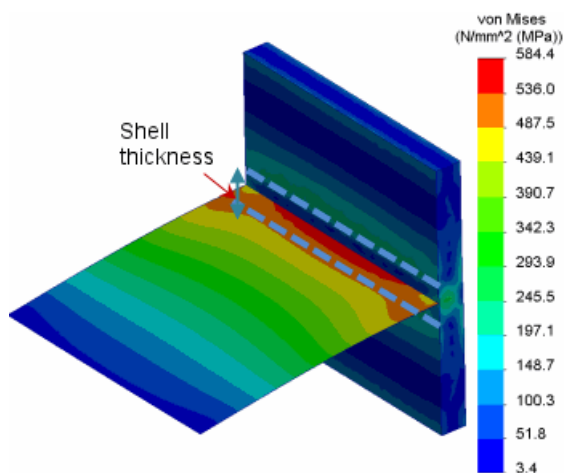
The software automatically bonds the detected contact pairs between shell edges and faces when you define bonded contact either globally or locally.

The Contact Sets PropertyManager has a new option: **Find shell edge - solid/shell face pairs**.



Edge-Wing Solution for bonding Shell Edges to Solid Faces

The algorithm of bonding a shell edge to a solid face is improved to consider the shell thickness. When you define a global or local bonding contact, the software virtually imprints the thickness of the shell edge on the touching solid faces, and applies bonding automatically.



T- type bonding between a shell and a solid. The new bonding algorithm produces more accurate stress results at the vicinity of contact, because it takes into account the thickness of the shell.

Import Flow Simulation Data

You can import results from Flow Simulation analyses performed with thin solid or sheet metal parts to shells in SolidWorks Simulation. Imported data include temperature, pressure, shear stresses, and convection coefficient. You can view a graphic plot of the imported data in SolidWorks Simulation.

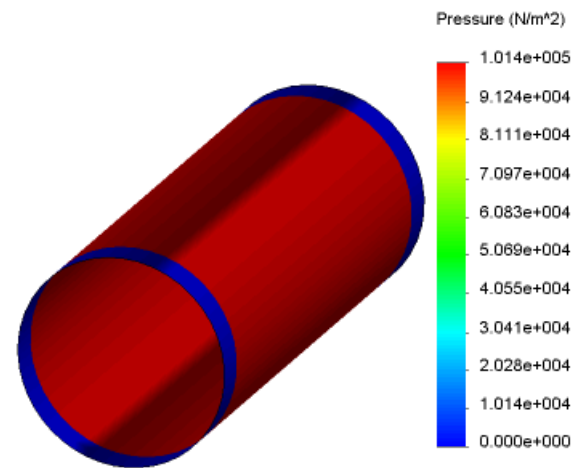
1. In the Simulation study tree, right-click **External Loads**, and select **Flow Effects**.

2. Select **Include fluid pressure effects from SolidWorks Flow Simulation**, and browse to select the Flow Simulation results file (.fld).



If your model consists of thin solid parts, before you import the Flow Simulation results, define shells from selected solid faces with thickness equal to the thickness of the solids.

3. To view the imported Flow Simulation data, mesh the model, right-click the flow data definition under **External Loads**, and select **Show Plot**.



You can also view Flow Simulation data imported from solids in Simulation study trees. New plots for fluid shear stresses are now included.


This chapter includes the following topics:



- **Dimensions**
- **Linear and Circular Sketch Patterns**
- **Shortcut Menus**
- **Sketch Rotation**


Dimensions


Inserting Dimensions

New menu items are available on the shortcut menu when you right-click in a sketch.

Sketch Numeric Input  Displays numeric input fields to specify sizes when creating sketch entities. This is the same option in **Tools > Options > System Options > Sketch** called **Enable on screen numeric input on entity creation**.

Add Dimension  Displays dimensions when creating sketch entities. Instead of selecting **Add dimensions** in sketch PropertyManagers, you can right-click in the graphics area and click **Add Dimension** .

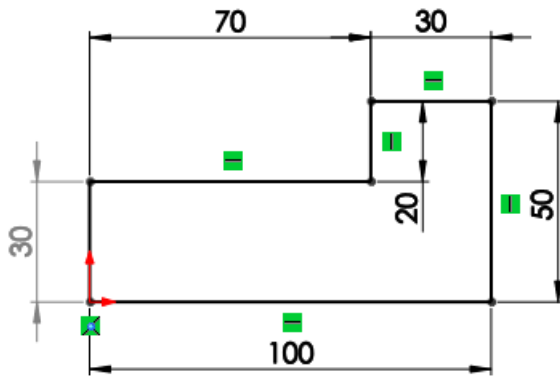
Sketch Dimension Driven  Creates driven dimensions when creating sketch entities. This is helpful when you want some dimensions to be driving and others to be driven because you can switch between the dimension types. In SolidWorks 2011, you could only create driven dimensions after you placed the dimensions.

For **Add Dimension** and **Sketch Dimension Driven** to appear in the shortcut menu, when you are using a sketch tool, right-click in the graphics area and click **Sketch Numeric Input** .

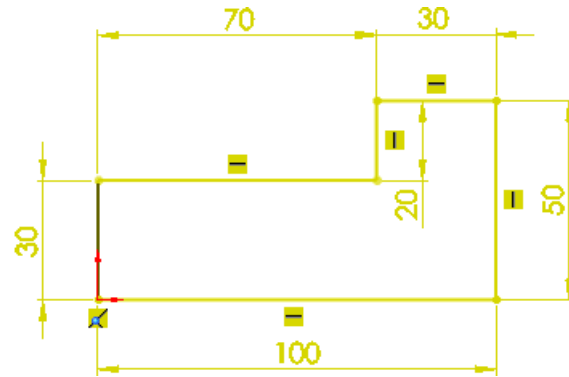
See *SolidWorks Help: Displaying Dimensions and Inserting Driven Dimensions*.

Over Defined Sketches

When you sketch the last entity to create a closed-profile sketch, it may clash with inferred relations from the other sketch entities. In this case, the dimension for the last sketch entity causes the sketch to be over defined. In SolidWorks 2012, the clashing dimension is changed to a driven dimension.




SolidWorks 2012: The dimension on the left vertical line that causes the sketch to be over defined is changed to driven



SolidWorks 2011: The dimension on the left vertical line causes the sketch to be over defined

Undo

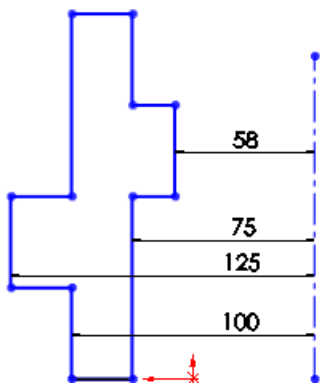
When creating dimensions with the **Smart**, **Horizontal**, or **Vertical Dimension** tools, you can undo previous selections by pressing **Esc**.

For example, when dimensioning multiple entities using the **Smart Dimension**  tool, you can press **Esc** to undo the last selection. This is helpful if you accidentally select an entity that you did not want to dimension.

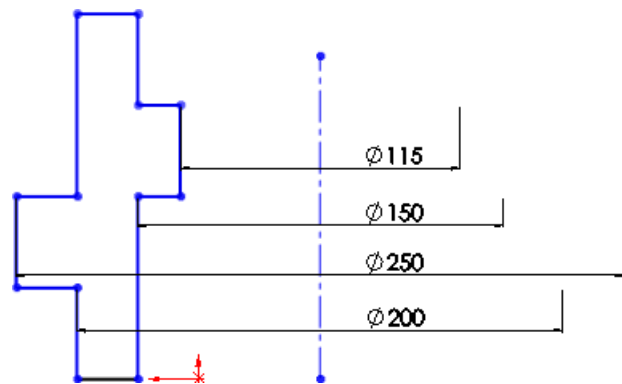
Using Centerlines to Create Radial and Diametric Dimensions

You can create multiple radial or diametric dimensions without selecting the centerline each time.


This type of dimensioning is helpful when you create sketches for revolved geometry that require several diameter dimensions.



Multiple radial dimensions



Multiple diametric dimensions

1. In a sketch with a centerline and lines or points, click **Smart Dimension**  (Dimensions/Relations toolbar) or **Tools > Dimensions > Smart**.
2. Select the centerline and a line or point.
3. To create a radius, move the pointer to the near side of the centerline. To create a diameter, move the pointer to the far side of the centerline.
4. Click to place the dimension.

The pointer changes to  for radial dimensions and  for diametric dimensions.

5. Select other lines or points in the sketch to create additional dimensions.

Linear and Circular Sketch Patterns

Downstream Features

If you rebuild a linear or circular sketch pattern that has downstream features that depend on the pattern, relations or references to the pattern are maintained.

Dimensions in Linear and Circular Patterns

Options in the Linear Pattern and Circular Pattern PropertyManagers have changed.

In the Linear Pattern PropertyManager, the option **Add spacing dimension** is renamed to **Dimension X spacing** (for **Direction 1**) and **Dimension Y spacing** (for **Direction 2**). The option **Add angle dimension between axes** is renamed to **Dimension angle between axes**. These options are available when creating or editing linear patterns.

In the Circular Pattern PropertyManager, the option **Add spacing dimensions** is split into two different options:

Dimension radius	Displays the circular pattern radius.
Dimension angular spacing	Displays the dimension between the pattern instances.

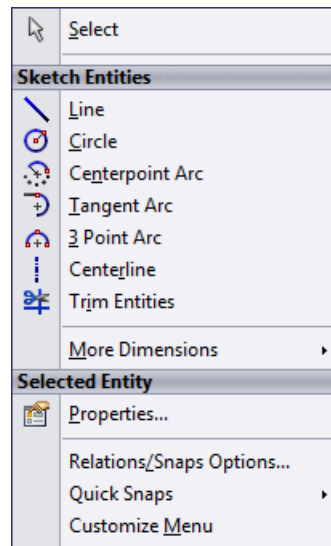
These options are available when creating or editing circular patterns.

Shortcut Menus

When sketching, the shortcut menu is longer so it displays more sketch and relations tools for ease of use and improved workflow.




SolidWorks 2012



SolidWorks 2011

Additionally, you can customize the shortcut menu by selecting which tools should be immediately accessible in the shortcut menu and which tools should be accessible after you expand the shortcut menu.

In a sketch, right-click to display the shortcut menu. In the shortcut menu, click , then **Customize Menu** to edit the menu.

Sketch Rotation

You can set an option to automatically rotate 2D sketches to be normal to the sketch plane.

Click **Tools > Options > System Options > Sketch** and select **Auto-rotate view normal to sketch plane on sketch creation**. Whenever you open a sketch on a plane, the view rotates so it is normal to the plane.

This chapter includes the following topics:

- **Increased Input Control**
- **Adding Materials**

Increased Input Control

SolidWorks Sustainability includes new fields that let you input more detailed information for your environmental studies.



In SustainabilityXpress, these fields appear as non-editable labels, showing default values.

You can specify the following:

- Amount of recycled content present in the material
- Amount of energy consumed during the manufacturing process
- Type of transportation and distances traveled during shipping
- Amount of time the product is designed to last as well as how long it will be used
- Product's End of Life disposition

Specifying Recycled Content

The **Recycled content** value in the Material section of the Sustainability Task Pane shows the percentage of recycled content for the selected material.

The information for the material in the sustainability database determines how the value is presented and whether you can modify it.

1. With a part or assembly open, click **Tools > Sustainability**.
2. In the Sustainability Task Pane, expand **Materials** and choose a material **Class** and **Name**.

Material

Class:
Aluminium Alloys

Name:
1060 Alloy

Recycled content:
75.00 %

Weight: 1.20 g

If the sustainability database contains primary (virgin), secondary (recycled), and trade mix (default mix of virgin and recycled content) values for the material you chose, the **Recycled content** field shows the percentage of recycled content.

- To change the amount of recycled material, specify a new percentage.
The control becomes yellow and the **Use Trade Mix** button becomes active.
- To revert to the default trade mix, click **Use Trade Mix**.

Specifying Manufacturing and Assembly Details

You can now specify additional details for use in evaluating the sustainability of manufacturing or assembly processes. New controls let you specify the expected lifetime of a part or assembly, the processes used to manufacture it, and the energy used in each process.

Defining the Manufacturing Process for a Part

- With a part open, click **Tools > Sustainability**.
- Expand the **Manufacturing** section of the Sustainability Task Pane.

The screenshot shows the 'Manufacturing' task pane with the following settings:




- Region:** Asia
- Built to last:** 2.00 Year
- Process:** Sand Casted
- Electricity:** 10.07 kWh/kg (Total electricity: 20.14 kWh)
- Natural Gas:** 13.00 BTU/kg (Total natural gas: 26.00 BTU)
- Scrap Rate:** 5.00 %

- Select the manufacturing region from the **Region** drop-down list or by clicking the region in the map.
- Under **Built to last**, enter a time increment and value to represent the expected life span of the part. For example, 10.00 and Year, indicating 10 years of life expectancy.
- Under **Process**, select the manufacturing process to be used.

The initial values in the editable fields for electricity use, natural gas use, and scrap rate are calculated based on the part's material and the manufacturing process you select.

The totals under each field are calculated based on the part's weight.

6. Correct any of the following values:

-  Amount of electricity used, in kilowatt hours per kilogram
-  Amount of natural gas used, in BTUs per kilogram
-  Scrap rate (i.e., the percentage of material that is discarded as scrap)

7. To recalculate the totals, click away from the field.



If you change the process, the field values and totals are recalculated with default values.

Defining the Assembly Process

1. With an assembly open, click **Tools** > **Sustainability**.
2. In the Sustainability Task Pane, use the **Task List** to ensure that materials are defined for all parts, then click **View Results**.
3. In the **Assembly Process** section, select the manufacturing region from the **Region** drop-down list or by clicking the region in the map.



Assembly Process

Region:
Asia



Built to last:
1.00 Year

Energy required for assembly process

4. Under **Built to last**, enter a time increment and value to represent the expected life span of the assembly. For example, 10.00 and Year, indicating 10 years of life expectancy.



If the **Built to last** value of the whole assembly is 10 years and the **Built to last** value of one of the assembly's components is 5 years, the components whose value is five years will be included twice in the impact calculation of the assembly.

5. If you want to specify the energy usage to build the assembled product, select **Energy required for assembly process**.
6. In the **Fuel type** drop-down list, select **Electricity** or **Natural Gas**.
7. In the **Amount** field, type the amount of energy you expect to be used.

A dialog box titled "Energy required for assembly process" with a checked checkbox. Below it is a "Fuel type:" dropdown menu with "Natural Gas" selected. At the bottom is an "Amount:" field with "8.00 BTU" and a spinner control.

Specifying Part and Assembly Use Parameters

A new **Use** section of the Sustainability Task pane lets you specify where the product will be used. For assemblies, this is where you now indicate the energy needs of the product over its lifespan.

For a part, in the **Use** section, select the manufacturing region from the **Region** drop-down list or by clicking the region in the map.

To provide assembly Use parameters:

1. With an assembly open, click **Tools > Sustainability**.
2. In the Sustainability Task Pane, use the **Task List** to ensure that materials are defined for all parts, then click **View Results**.
3. In the **Use** section of the Sustainability Task pane, select the manufacturing region from the **Region** drop-down list or by clicking the region in the map.

A dialog box titled "Use" with a "Region:" dropdown menu showing "North America". Below the dropdown is a world map with North America highlighted in blue. At the bottom is an unchecked checkbox labeled "Energy needs over lifespan:".

4. If you want to specify the energy used during the lifetime of the product, select **Energy needs over lifespan**.

A dialog box titled "Energy needs over lifespan:" with a checked checkbox. Below it is a "Type of energy:" dropdown menu with "Diesel" selected. At the bottom is an "Amount:" field with "2.00 gal" and a spinner control, followed by a unit dropdown menu with "Hour" selected.

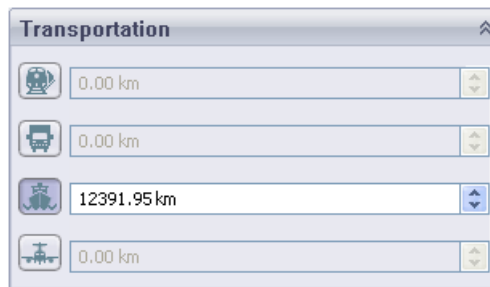
5. In the **Type of Energy** drop-down list, select from:
 - **Electricity**
 - **Natural Gas**
 - **Diesel**

- **Gasoline**
 - **Kerosene**
 - **Light Fuel Oil**
6. Specify the **Amount** of energy and select a time increment. For example, selecting **Diesel**, typing 2.00 and selecting **Hour**, indicates that the product will use 2.00 gallons of diesel fuel per hour during its lifetime.

Specifying Shipping Details

For both parts and assemblies, the **Transportation** section of the Sustainability Task Pane lets you identify shipping methods and distances.

1. In the Sustainability Task Pane, expand **Transportation**.



Default shipping distances are based on the **Use Region** selected.

The units are given in kilometers or miles, depending on whether the units for the part's weight are metric.

2. Click a button to enable editing of its value.
You can edit any or all of the following:

 **Train**

 **Truck**

 **Boat**

 **Plane**

3. Specify the distance traveled for the selected transportation mode.
Changing a value highlights the new value in yellow and enables the **Reset to Defaults** button, which you can use to return to the default transportation settings for the current **Use Region**.

Transportation

0.00 km

0.00 km

1391.95 km

24873.50 km

Reset to Defaults

Specifying End of Life Disposal Methods

The **End of Life** section lets you specify how the product will be disposed of when it reaches the end of its life. You can adjust the percentages of the product that will be recycled, incinerated, or sent to a landfill.

1. Scroll to the **End of Life** section of the Sustainability Task Pane.



End of Life

18.00%

60.00%

22.00%

Default values based on the material are listed for:

-  **Recycled**
-  **Incinerated**
-  **Landfill**

2. Change a value in any of these fields.

End of Life

18.00%

80.00%


2.00%

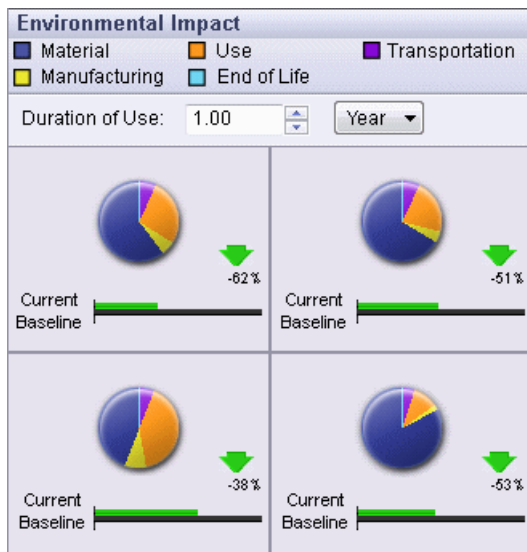
Reset to Defaults

The values are linked so that the total of the percentages is always 100%. A change to the **Recycled** or **Incinerated** fields modifies the value of the **Landfill** field. A change to the **Landfill** field modifies the value of the **Recycled** field.

3. Click **Reset to Defaults** to return to the original values.

Environmental Impact Dashboard Changes

On the **Environmental Impact** dashboard, a new **Duration of Use** field lets you specify the length of time a product will be used. A **Home** button  makes it easy to return to the initial view of the dashboard.



Duration of Use

When sustainability is calculated, the **Duration of Use** value is used in conjunction with the **Built to last** value you specify in the **Manufacturing** section.

To get a more accurate picture of the impact of a design, the **Duration of Use** value is divided by the **Built to last** value. The result is rounded to the next whole number.




For example, if a product is built to last four years and its duration of use is 10 years, the calculation is $10 / 4 = 2.5$, rounded up to 3. This means that to have the use of the product for 10 years, you will need a quantity of three. So the net impact on all environment results is three times that of a single product.

Conversely, if a product is built to last for 10 years and its duration of use is one year, the calculation is $1 / 10 = .01$, rounded up to 1. This means you will only need a quantity of one to satisfy the duration of use.



Duration of Use and **Built to last** have been added to reports and the Custom Column choices for Assembly Visualization.

Navigation

The **Previous**  and **Next**  buttons from earlier versions of Sustainability have been replaced by a **Home**  button.

Clicking **Home** when you are viewing environmental impact details such as **Carbon Footprint** or **Energy Consumption** returns you to the initial display of the **Environmental Impact** dashboard, showing the summary pie charts.

Bar Graphs

Arrows have been added to the bar graphs below the pie charts on the dashboard Home page. They emphasize the increase ▲ or decrease ▼ in environmental impact.


Adding Materials

You can access additional materials for Sustainability studies using the Sustainability Extras folder in the SolidWorks Materials dialog box.

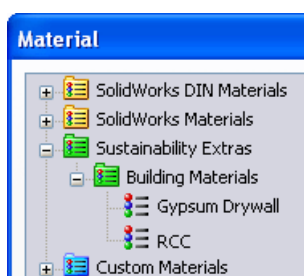
You can download additional materials and request materials that are not yet available.

Updating the Materials Database

When a new version of the Sustainability database is available, you can download it from the Material dialog box.

1. With a part open, in the FeatureManager design tree, right-click **Material**  and select **Edit Material**.
2. In the Material dialog box, in the left pane, click **Sustainability Extras**.
The Properties tab in the right pane shows the latest version of the GaBi Sustainability Database and when you last checked for updates.
3. Click **Check for Updates**.
The Check for Updates dialog box tells you whether an update to the database is available.
4. If an update is available, click **Update**.
5. When the download is complete, click **OK**.

The database version is updated and the added materials are listed under **Sustainability Extras**.



You can edit the material as you would any other material. If you make changes, they are preserved when you update the database again.


When you use Sustainability, the materials you have added appear in the following places:

When you select a material for a part The **Class** drop-down list identifies the added material class with a Sustainability Extras prefix.

In the Find Similar Material dialog box When you select **-any-** as a **Material Class**, the results that match the specified criteria include materials from the Sustainability Extras folder.

Requesting Materials

If the Materials dialog box does not contain a material you need, you can request that the material be added.

1. In the FeatureManager design tree, right-click **Material**  and select **Edit Material**.
2. In the Material dialog box, in the left pane, click **Sustainability Extras**.
3. On the Properties tab in the right pane, to see a listing of available materials, click **Check online listing**.

A Web page opens, listing the SolidWorks materials that are currently defined in the sustainability database and what you can model with them.

4. To request that a material be added to the database, click **Submit a Request**.

The **Request a Material** form on the SolidWorks Sustainability Website appears. The form includes a list of recent material requests and their status.

5. Complete and submit the form.
6. When you are notified that the material you requested has been added, click **Check for Updates** to download the latest updates.

Available in SolidWorks Professional and SolidWorks Premium.

This chapter includes the following topics:

- **Performance and Customization Enhancements**

Performance and Customization Enhancements

To permit more flexible Toolbox customization and to improve performance, Toolbox component information is now stored in each part file rather than in a Microsoft Access database. For example, from Toolbox Settings, you can assign components to folders you create and you can rename files. You can also modify components within the default standards.

To open Toolbox Settings, click **SolidWorks version** > **SolidWorks Tools** > **Toolbox Settings** from the list of all programs on the **Start** menu of your computer.

See *SolidWorks Help: Configuring Toolbox* for more information.

Customizing Toolbox

You can customize Toolbox content from the Customize Hardware page of Toolbox Settings by copying, pasting, deleting, and renaming Toolbox folders.

In this example, you create a new Toolbox folder, add content to it, rename content, and delete content from it.



Your Toolbox administrator might have already customized Toolbox for you. Consult your Toolbox administrator before you perform this exercise.

First, open Toolbox Settings to begin hardware customization.

To open Toolbox Settings, click **SolidWorks version** > **SolidWorks Tools** > **Toolbox Settings** from the list of all programs on the **Start** menu of your computer.

From Toolbox Settings, click **2 - Customize Your Hardware**.



Creating and Renaming Toolbox Folders

Next, you create a new Toolbox folder and rename it.

1. Right-click **Toolbox** standards and click **New Folder**.
A folder named **New Folder** appears at the bottom of the Toolbox Standards tree.
2. Click the folder name, and for **Title**, type `My Company Hardware`.

Copying and Pasting Toolbox Folders

Next, you copy the ANSI Inch bolts and screws into your custom hardware folder.



If your Toolbox administrator has already deleted ANSI Inch tools, you can follow along with this example by performing the tasks with another Toolbox standard.

1. Select the ANSI Inch standard if it is not already selected for your Toolbox content:
 - a) Click **Toolbox Standards** in the standards tree, and in the Customize Hardware pane, select **ANSI Inch**.
2. Copy Toolbox content in the **Toolbox Standards** tree. For example:
 - a) Expand **ANSI Inch**.
 - b) Right-click **Bolts and Screws** and click **Copy**.



To move a folder or standard rather than copy it, you can click **Cut** instead of **Copy**.

3. Right-click the **My Company Hardware** folder in the **Toolbox Standards** tree and click **Paste**.

A copy of the **Bolts and Screws** folder appears under **My Company Hardware**.



When you copy and paste Toolbox content, a number is appended to the Toolbox filenames so that they remain unique. Cutting and pasting does not modify Toolbox filenames.



Copying standard Toolbox content from one folder to another does not affect smart fastener assignment. For each hole type, you can reassign smart fasteners.

Renaming Toolbox Content and Deleting Folders

Next, you modify the filename for Toolbox content in your custom Toolbox content folder and delete unnecessary content.

For this example, suppose your assemblies reference two types of heavy hex bolts. To differentiate the heavy hex bolts in your custom Toolbox content folder from those in the ANSI Inch folder, you can change the filename and title assigned to the heavy hex bolts in your custom Toolbox content folder. In addition, suppose your company never uses round head bolts. You can remove round head bolts from your custom Toolbox content folder.

1. In the **My Company Hardware** folder in the **Toolbox Standards** tree, expand **Bolts and Screws**.
2. Expand **Hex Head** and select **Heavy Hex Bolt**.
3. In the **Heavy Hex Bolt** properties pane, change the filename and title:
 - a) For **Filename**, type `My Favorite ANSI Inch Hex Bolt.sldprt`.



All Toolbox filenames must be unique. You cannot assign a filename that conflicts with the name of another Toolbox file.

- b) For **Title** change **Heavy Hex Bolt** to `My Favorite ANSI Inch Hex Bolt`.



The part **Title** affects only the name displayed in the **Toolbox Standards** tree and not the part filename.

4. Remove the round head bolt folder from the **My Company Hardware** folder:
 - a) Right-click the **Round Head Bolts** folder and click **Delete**.
 - b) Click **OK**.
5. Close Toolbox Settings.
6. When prompted to save changes, do one of the following:
 - To save your changes, click **Yes**.
 - To discard your changes, click **No**.

You can perform similar operations to move folders around, rename folders and tools, and delete unnecessary tools or folders.




If you do not use the ANSI Inch standard at your company, select **Toolbox Standards** and clear the **ANSI Inch** standard after you complete this example.

This chapter includes the following topics:

- **Cut List Properties**

Cut List Properties

When you create subweldments, or when you save weldment bodies to new parts, the cut list properties from the parents propagate to the subweldments or new parts.

In the Cut-List Properties dialog box, the **Value / Text Expression** of the properties appear as **Linked to Parent Part-** *parent* **.sldprt**. You cannot edit the cut list properties unless you break the reference to the parent part. To open the Cut-List Properties dialog box, in the FeatureManager design tree, right-click a **Cut-List-Item**  and click **Properties**.

SolidWorks Workgroup PDM

This chapter includes the following topics:

- **Rebuilding Workgroup PDM Vault Metadata**
- **Accessing the Workgroup PDM Vault from Windows 7 and Windows Vista**
- **Pack and Go Improvements**

Rebuilding Workgroup PDM Vault Metadata

The **Rebuild Vault** option in the SolidWorks Workgroup PDM VaultAdmin tool lets you rebuild vault metadata from key base files.



Before rebuilding a vault, the recommended best practice is to create a backup or ensure that your regularly scheduled backup is up to date.

You can improve the startup time of the vault by running the **Rebuild Vault** option once or twice a month on the system where the vault was created.

You should also run the **Rebuild Vault** option before migrating Workgroup PDM vault data to Enterprise PDM or exporting from a Workgroup PDM vault.

To rebuild the vault:

1. Log in to the Workgroup PDM vault as a user with administrator privileges.
2. On the Vault Settings tab, under **Vault Lock**, click **Locked** to lock all users out of the vault.
3. Click **Rebuild Vault**.
4. In the Repair options dialog box, select rebuild options and click **OK**.
5. When warned that the vault will be shutdown, click **Yes**.

Accessing the Workgroup PDM Vault from Windows 7 and Windows Vista

The SolidWorks Open dialog box allows Windows 7 and Windows Vista users to access Workgroup PDM vaults.

A secondary dialog box lets you search in the Workgroup PDM vault and check out files or get the latest file. When you select a file, the SolidWorks Open dialog box lets you open the file as if it were a local file.

To access Workgroup PDM vault files:

1. In SolidWorks, click **File > Open**.
2. In the Open dialog box, click **Browse vault**.
3. In the Select from Vault dialog box, select a file and do any of the following:

- To preview the file, select **Thumbnail**.
- To check out the file without using other options, select the **Check Out** check box and then click **Checkout**.
- To see additional Workgroup PDM options, click **Check Out Advanced** or clear **Check Out** and click **Select Advanced**.

The Get Local Copy and Open Document dialog box opens.

When you check out a file directly or select **Open/Check Out** from the Get Local Copy and Open Document dialog box, the SolidWorks Open dialog box reappears with the file listed as a local document.

4. Select additional options from the SolidWorks Open dialog box options and click **Open**.

Pack and Go Improvements

When you run Pack and Go, a progress bar reports the steps in the Pack and Go process. The **Cancel** button remains active so you can cancel after beginning the operation.

In addition, Pack and Go provides improved support for non-ASCII characters in file names.

In the Pack and Go dialog box, the progress bar and **Cancel** button become active when you click **Save** to start the process of creating the package.