WHAT'S NEW
SOLIDWORKS 2014 PRE-RELEASE

Contents

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Welcome to SolidWorks 2014

This chapter includes the following topics:

- **Highlights**
- **Top Enhancements**
- **For More Information**

Highlights

SolidWorks® 2014 includes many enhancements and improvements, most in direct response to customer requests. This release focuses on the following themes:

- **Focused design tools**: Powerful new features and time-saving shortcuts deliver greater productivity.
- **Integrated workflows**: Streamlined processes and information sharing remove the barriers to innovative design.
- **Increased performance**: Faster system response and reduced user interaction help you maintain design focus.
- **Enhanced visualization**: Stunning graphics and improved visualization tools create a more powerful collaborative environment.

Top Enhancements

The top enhancements for SolidWorks 2014 provide improvements to existing products and innovative new functionality.

Throughout this guide, look for the ★ symbol in these areas:

| SolidWorks Simulation | • **Automatic Conversion of Toolbox Fasteners to Bolts** on page 172
• **Contact Visualization Plot** on page 176
• **Importing Results from SolidWorks Plastics** on page 180 |

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: • **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.

2

User Interface

This chapter includes the following topics:

- **Accessibility**
- **Applying Standard Mates from a Context Toolbar**
- **FeatureManager Design Tree Response to Selections**
- **Including SmartMates in the CommandManager**
- **Selecting Items by Lasso**
- **Sticky Settings**
- **Text and Button Sizes**

Accessibility

An updated color scheme for certain icons makes the SolidWorks application more accessible to people with color blindness. Also, provides a more readable font.

Icons in the active PropertyManager use blue to indicate what you must select on the screen; faces, edges, and so on. Adding blue to the icons provides a sharp contrast in colors so that many SolidWorks users with color blindness can distinguish one icon from another.

Along with style updates to the main elements of the SolidWorks user interface, there is a more readable font. Turn on ClearType®, software technology developed by Microsoft®, to make the text on your screen easier to read.
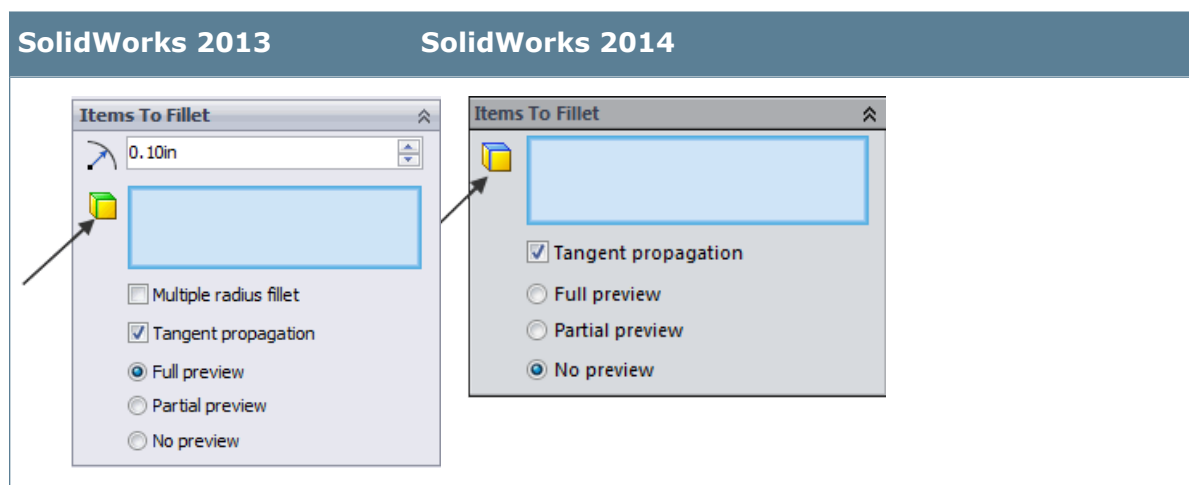
Turning on ClearType

To turn on ClearType:

1. From the Windows **Start** menu, click **Control Panel**.
2. Click **Appearance and Personalization**.
3. Click **Display**, and on the left, click **Adjust ClearType text**.
4. Select **Turn on ClearType**, click **Next**, and complete the wizard.

The text on your screen uses the new, easier to read SolidWorks font.

Applying Standard Mates from a Context Toolbar ⭐

You can apply standard mates in an assembly from a context toolbar.

The toolbar appears when you press **CTRL +** select mating geometry in the graphics area. Only mates that are appropriate for your selections are available.
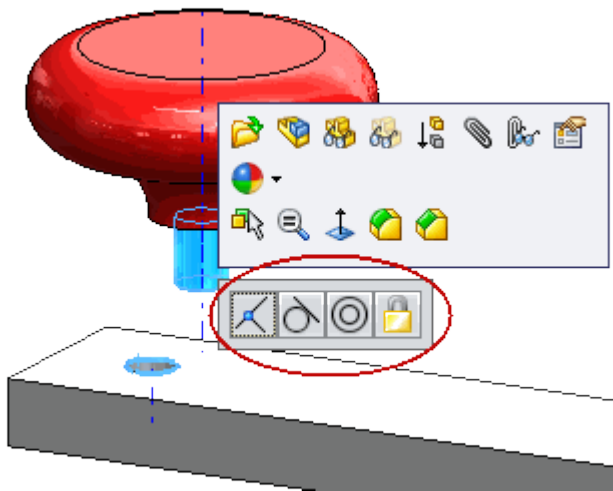
1. From an assembly, press **CTRL +** select entities to mate.
   The default mate is highlighted in the context toolbar.
2. Select a mate.
3. For distance or angle mates, enter the mate specification and click ✔ .

FeatureManager Design Tree Response to Selections

When you select an item in the graphics area, the FeatureManager® design tree expands and scrolls to the selected item. Clicking in the graphics area without selecting an item restores the scroll state of the FeatureManager design tree.

Before Selection	After Selection

Before Selection	After Selection
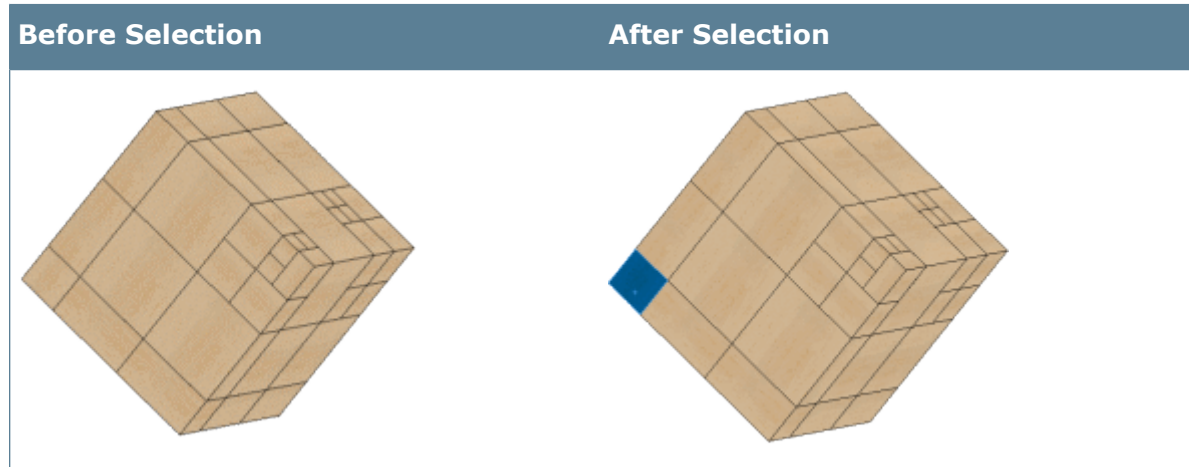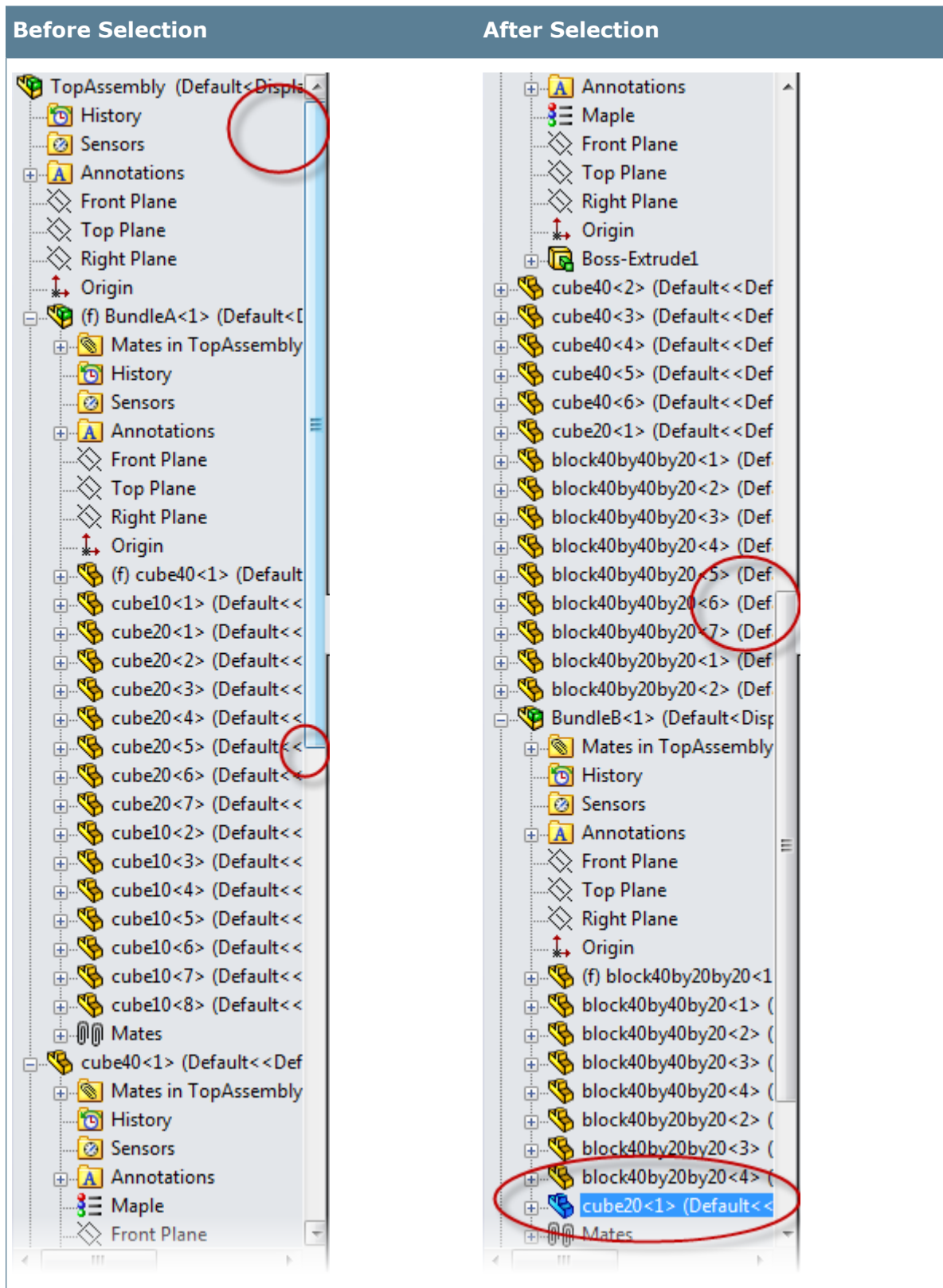
This behavior is on by default. To turn this behavior off, click **Tools** > **Options**, select **FeatureManager**, and clear **Scroll selected item into view**.

Including SmartMates in the CommandManager

You can include the **SmartMates** command when customizing the Assembly toolbar or the Assembly CommandManager.

To include the SmartMates command in the CommandManager:

1. Open an assembly.
2. Click **Tools** > **Customize**, or right-click the CommandManager and select **Customize**.
3. On the Commands tab, select **Assembly**.
4. Drag the **SmartMates** 🏷 command to the Assembly CommandManager, and click **OK**.

Selecting Items by Lasso ★

You can select items in the graphics area with a lasso.

> You can lasso-select items in sketches, drawings, parts, and assemblies.

To lasso-select items in an open document:

1. Set the default selection method to lasso selection by one of the following methods:

    - In the graphics area, right-click and click **Lasso Selection**.

      > The software toggles the default selection method between lasso selection and box selection. If **Box Selection**, rather than **Lasso Selection** is available when you right-click, then your default selection method is already set to lasso selection.

    - Click **Tools** > **Lasso Selection**.

    - Click **Tools** > **Options** or **Options** 🗒 (Standard toolbar). Select **Display/Selection**, and for **Default bulk selection method**, select **Lasso**.

2. Drag your pointer to select items.

    > You can lasso-select items without closing the lasso.

    > For clockwise lasso selection, the lasso selects only items contained in the lasso loop. For counterclockwise lasso selection, the lasso selects sketch entities in the lasso loop and items that cross the lasso.

The software supports counterclockwise lasso selection only in drawings or sketches.
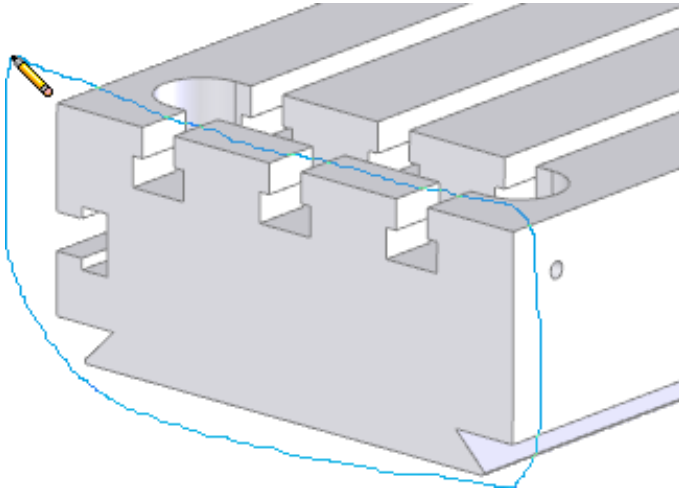
3. Release the mouse button.
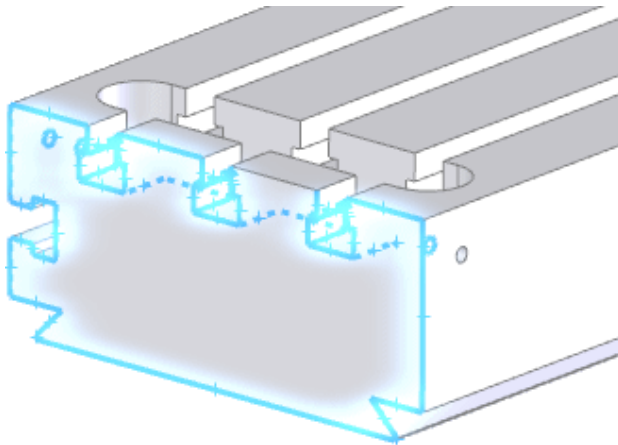
Sticky Settings

Settings in certain options persist across SolidWorks sessions and versions until you change them.

Settings in certain options persist after you click **OK** or when you accept PropertyManager options.

The following options persist across SolidWorks sessions and versions:

Knit Surface PropertyManager	When you click **Gap Control** and type custom values for the Gap Control dialog box, the **Gap Control** check box and the values you enter persist.
Edge Flange PropertyManager	The new settings under **Flange Length** and under **Flange Position** and the values that you enter persist.

Hole Specification PropertyManager (Hole Wizard)	The settings under **End Condition** and the **Depth** values that you enter persist.

Text and Button Sizes

You can set sizes for text and buttons from the Toolbars tab of the Customize dialog box.

- Text in menus, tree items, dialog boxes, and PropertyManagers respects your SolidWorks software or operating system text size selection.
- You can set button sizes independent of the text size.
- The larger button sizes facilitate running the SolidWorks software on machines with high resolution displays.
- For low resolution displays, dialog boxes are automatically resized when they do not fit on the screen, and scrollbars appear automatically.
- To facilitate element selection on touch interfaces such as tablets, you can set up the larger size buttons and text from the **Options** 🔲 ▾ menu (Standard toolbar).

To view the text and button settings, click **Tools** > **Customize**, and in the dialog box, select the Toolbars tab.

3
SolidWorks Fundamentals

This chapter includes the following topics:

- **Accessing Recent Features Through History**
- **Application Programming Interface**
- **Equations**
- **Dismissing All Prompts for Missing Components**
- **Graphics**
- **Updating Assembly Graphics**
- **Opening Document Copies**
- **Pack and Go Toolbox Option**
- **Recent Documents**
- **Version Support for DXF and DWG Files**
- **Saving SolidWorks Files as Portable Network Graphics Files**

Accessing Recent Features Through History

From the **History** folder at the top of the FeatureManager design tree, you can access the features that you have most recently created or edited. You can also control the number of features to list in the **History** folder.

To access recent history:

- In the FeatureManager design tree, expand the **History** folder.
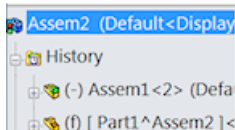
Setting History Length

You can set the number of edited features listed in the **History** folder.

To set the history length:

1. From an open document, in the FeatureManager design tree, right-click the **History** folder and click **History Options**.
2. In the PropertyManager, under **History Length**, for **Maximum**, type an integer between 1 and 99.

Application Programming Interface

See *SolidWorks 2014 API Help Release Notes* for late-breaking updates.

SolidWorks 2014 includes the ability to:

- Access mirror part feature data.
- Access flat-pattern folders in the FeatureManager design tree.
- Access sheet metal folders in the FeatureManager design tree.
- Access Fill Pattern feature data.
- Insert a Variable Pattern feature in the FeatureManager design tree.
- Create a new part document that mirrors another part document about a selected reference plane or planar face.
- Select assembly components by percent of assembly size.
- Get better error reporting when getting, setting, adding, or deleting custom properties.
- Attach an existing annotation to a drawing sheet or view.
- Replace the model in selected drawing views.
- Get library feature reference names.
- Get and set whether a part is a SolidWorks Toolbox part.
- Get a return code after rebuilding an assembly or drawing.
- Get and set the property values for a sunlight source.
- Access SolidWorks CommandManager tabs:
  - Get all of the tab names.
  - Get and set the visibility of tabs.
  - Get and set the active tab.
  - Get and set the index of tabs.
  - Send notification when a tab is about to be activated for:
    - Assembly document.
    - Drawing document.
    - Part document.

- Get and set whether to generate a graphics-only section view.
- Get and set whether to override default sheet metal feature parameters.
- Get and set whether to use a sheet metal feature gauge table.
- Get the actual entities associated with a sketch relation.
- Save various aspects of a part (sheet metal, faces, loops, and annotation views) to one or more DXF/DWG files, preserving the specified filename.
- Get and set the text format and font size of callouts.
- Change standard model views and reset them to their default settings.
- Get all visible entities, including silhouette edges, in drawing views.
- Send notification when a document is printed.
- Get the persistent ID of:
  - Component.
  - Feature.

- Change dimension tolerance values in specific configurations in a multi-configuration part or assembly.
- Derive the hole series feature data interface from IDispatch instead of IUnknown.
- Insert general table annotations in part and assembly documents.

Equations

Configurable Equations ⭐

The ability to apply configurations to equations and global variables is enhanced. You can configure equations and global variables in the same way as you configure dimensions.

Previously, you could only configure equations and global variables by suppressing or unsuppressing them in selected configurations.

Now you can create variations of equations and global variables, and apply them to configurations using the Equations and Modify dialog boxes. You can also configure equations and global variables in design tables, and in various PropertyManagers for parts and assembly features.

For example, when you create an equation in the **Distance** field of the Chamfer

PropertyManager, you apply configurations to it by using the configuration icon ⊞ that displays in the field. You can then edit the equation and assign it to different configurations using the Equation or Modify dialog boxes.

You can also specify a different unit of measurement for each configuration. For example, you can have one configuration where a global variable is expressed in millimeters and another where it is expressed in inches.

> In a design table, equations must be preceded by a single apostrophe and an equal sign ('=). The single apostrophe ensures, that if the equation is exported to Excel, it is interpreted as an equation and not as a text string.

You cannot use configurable equations and suppression/unsuppression in the same model. If you created a model with SolidWorks 2013 or an earlier version, you must remove all suppressed and unsuppressed equations and global variables, before adding configurable equations and global variables. You must also remove design tables, then save and re-open the document file with SolidWorks 2014.

Sketch View

A Sketch View is available in the Equations, Global Variables and Dimensions dialog box.

With this view, you can add, rename, edit, configure, and delete equations, global variables, and dimensions used in sketches.

Dismissing All Prompts for Missing Components

You can dismiss all prompts for missing components for the document you are opening.

Select **Suppress all missing components** in the Unable to locate file dialog box that appears when a document cannot locate a referenced file.

> This option affects only the current document. This option does not permanently dismiss all Unable to locate file dialog boxes for all documents.

Graphics

Accurate Environment Lighting

You can apply accurate environment lighting to clean up excessive noise in renderings.

The benefits of using accurate environment lighting are:

- Accurate environment lighting calculation from custom high dynamic range images, HDRI, used for scene environments.

Use accurate environment light when:

- Shadows in the current rendering are too noisy.
- You use a custom HDRI image to define the scene and you need more accurate lighting from the custom HDRI image.

Apply the accurate environment lighting when you refine renderings using the PhotoView 360 preview window.

> When you use this setting, it can significantly slow down rendering.

To apply accurate environment lighting:

1. With a model open, click **Tools** > **Add-Ins** and add in PhotoView 360.

2. From the DisplayManager, click **View Scene, Lights, and Cameras** 🖼️.

3. Right-click **Scene** and select **Edit Scene**.

> To view the document scene you must set the background appearance to **Use document scene**. Click **Yes** to set the background appearance, if the Background Display Setting dialog box appears.

4. On the Illumination tab, under **PhotoView Illumination** , select **Accurate environment lighting**.

5. Click ✔ .

Without accurate environment lighting	**With accurate environment lighting**

Refining Renderings

You can save renderings in progress directly from the PhotoView 360 Preview window.

You can use the extended refinement settings in the PhotoView 360 Preview window to refine the rendering beyond the specified quality setting in the PhotoView options. The quality of the rendering increases over time. You can also specify that the PhotoView 360 Preview window create an image that uses the full specified output image size in the PhotoView options.

> Use continuous refinement with the **Accurate environment lighting** option to get the best results.

To refine renderings:

1. Click **PhotoView 360** > **Preview Window**.
2. Choose how you want to do the rendering:

   - **Extended refinement** to force the preview to refine beyond the specified quality settings in PhotoView options.
   - **Full resolution preview** to force the preview to render an image that is saved at the size specified in PhotoView options.

3. Click **Save Preview Image** to save the result displayed in the PhotoView Preview window at any time.

Creating Graphics-only Section Views

You can quickly generate a section view by generating a graphics-only section view.

To create a graphics-only section view:

1. Open a part or assembly.

2. On the View toolbar, click **Section View** .
3. Under **Section 1**, select **Graphics-only section**.

4. Set other options and click .

> You cannot select a sectioned face or edge. You must retain the section cap color in a graphics-only section view. Pixels that lay within the same plane as the section plane or face are not hidden.

Exporting SolidWorks Models to *.lxo Files

You can export SolidWorks models, including models with motion studies, to `*.lxo` files. You can then import the `*.lxo` files into MODO®, the rendering product created by Luxology LLC.

To export a SolidWorks model:

Click **File** > **Save as**. In **Save as type**, select **Luxology (*.lxo)**.

To export a SolidWorks model and the information from a motion study:

Click **Save Animation** 🖼 (MotionManager toolbar). In **Save as type**, select **Luxology (*.lxo)**.

Flattened Floors in Scenes

You can flatten the floor of a spherical environment in scenes to improve the look of models that naturally rest on the ground or flat floors, particularly when performing view manipulations such as rotate or zoom.

If the model is too large or too small relative to something in the environment, you can adjust the environment size and scale the environment relative to the size of the model.

You can adjust the horizon height, essentially, from the equator down to the South pole. Imagine if the HDRI sphere scene is on the ocean. If you position the camera at a height, such that the horizon is in the middle of the image, then you can adjust the horizon height to line up with the horizon line. In this case, the gum drop is, essentially, a hemisphere.

If you position the camera very high from the surface of the ocean, then the horizon is toward the bottom of the image and you can adjust the horizon height to line up with the horizon line. In this case, the gum drop is like a sphere with a flat surface on one side that lines up with the floor.

To flatten floors in scenes:

1. From the DisplayManager, click **View Scene, Lights, and Cameras** 🖼.
2. Right-click **Scene** and select **Edit Scene**.
3. Under **Background**, select **Use Environment**.

   > You must select **Use Environment** to use the flatten floor option.

4. Under **Floor**, click **Flatten floor**.
5. Set the **Environment size** to specify the size of the HDRI scene sphere, the gum drop, that surrounds the model.
6. Set the **Horizon Height** to specify at what lattitude on the HDRI scene sphere the floor starts to flatten.

7. Click ✔ .

   > The Flatten floor option works best for environments that have a natural horizon line near the equator of the spherical environment image. These tend to be exterior environments. The Presentation Scenes folder now includes 10 new scenes which work well with the flatten ground control.

Performance Improvements to Rendered Animations

PhotoView 360 saves rendered animations to *.lxo files which contain all the data: the appearances, scenes, model geometry, and motion study definitions needed to process

rendered animations. Internal conversion to *.lxo files dramatically reduces the time required to process a rendered animation.

In previous versions of the SolidWorks software, for each frame of a motion study, the SolidWorks MotionManager solves the motion and saves the model at that frame. PhotoView 360 then loads the model and renders the model for each frame. For example, if you have a one second animation at 30 frames per second (FPS), the SolidWorks software saves a model 30 times and PhotoView 360 loads it and renders the model 30 times. Saving each frame and loading it into PhotoView 360 takes a significant amount of time.

With the performance improvements to rendered animations, because the SolidWorks software saves the motion study information in *.lxo files, when PhotoView 360 loads the model the first time, Photoview 360 has all the information about the entire motion study model to render it. In the example of a one second animation at 30 FPS, SolidWorks saves the model just once and PhotoView 360 loads the files once to render all the frames of the animation.

Selecting Bodies or Components for Section Views ★

You can select bodies or components to include in or exclude from section views.

To select bodies or components for section views:

1. Open *install_dir*\samples\whatsnew\fundamentals\jig saw\Jigsaw.SLDASM.
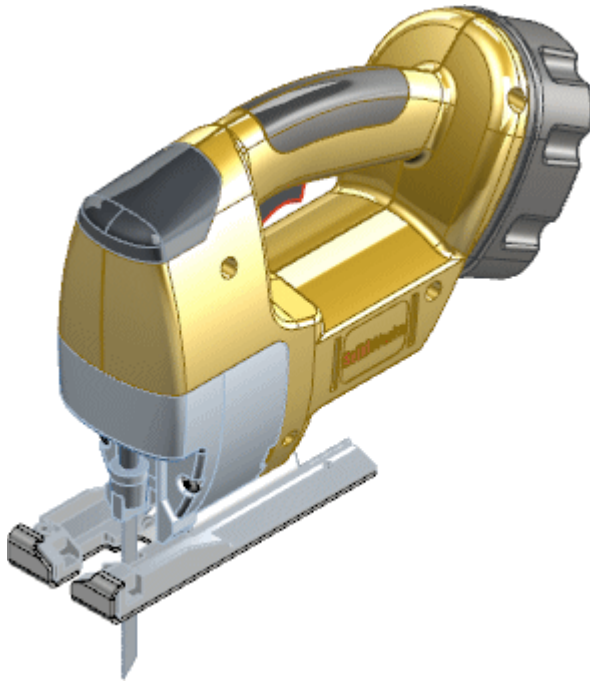
2. Click **Section View** (View toolbar) or **View** > **Display** > **Section View**.
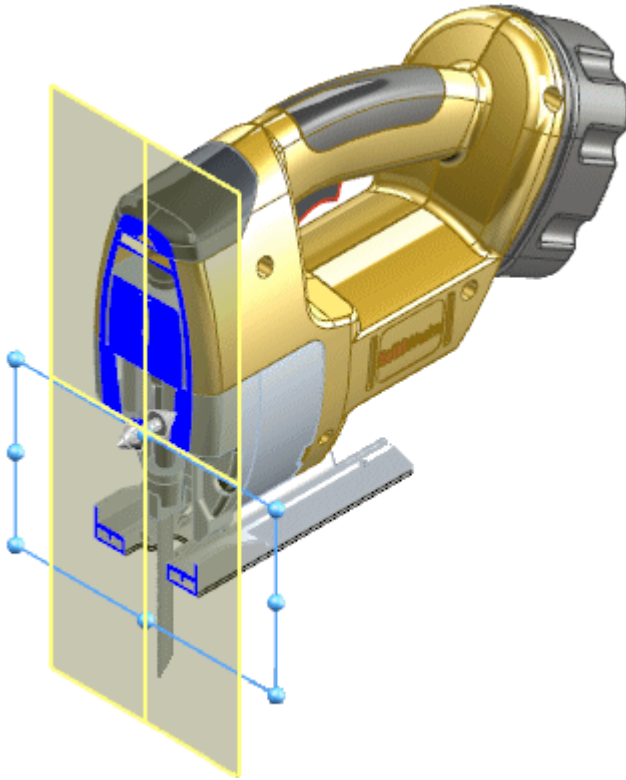
3. In the PropertyManager, under **Section 1**, click **Side** .
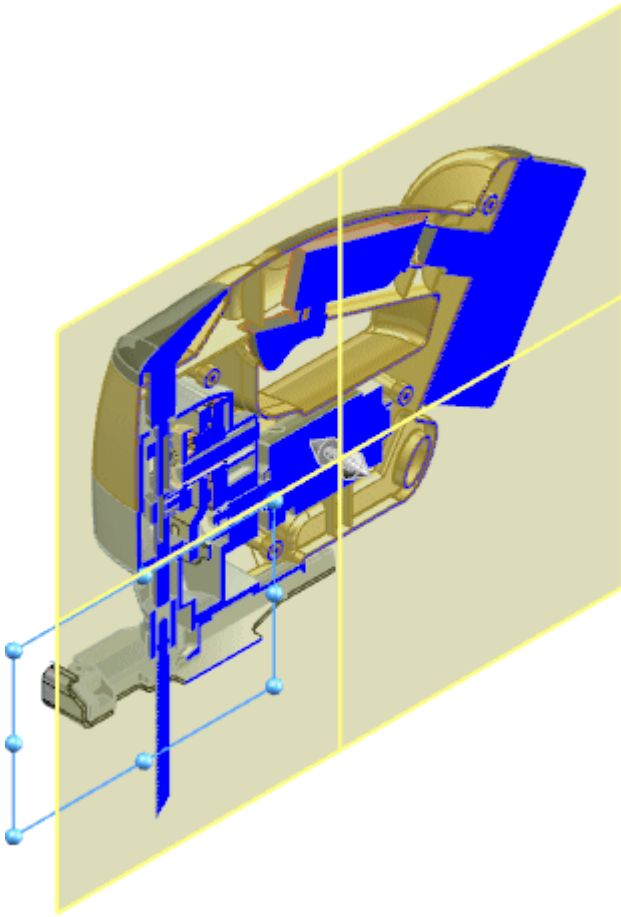
4. Click **Selected components**.
5. In the flyout FeatureManager design tree, select these items to add them to **Selected components**.
    - MOTOR DRIVE
    - SW0903A GEAR DRIVE
    - SW0903A CAM
    - NEEDLE ROLLER (NIH)
    - GEAR DRIVE SHAFT
    - SW0904 - PLUNGER ASSEMBLY

6. Under **Selected components**, select **Exclude selected**.

    The selected components appear in their entirety and all other parts appear as sectioned.
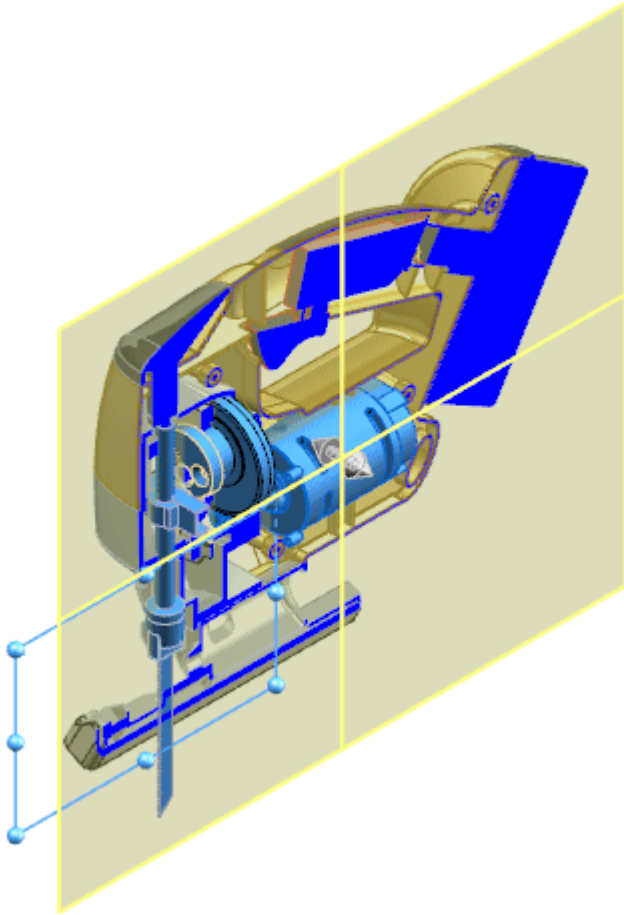
7. Click **Preview** to show the graphics-only preview of the section based on the section plane location and the components or bodies that you select.

Preview hides the section plane, reference plane and face outlines, and the selection plane.
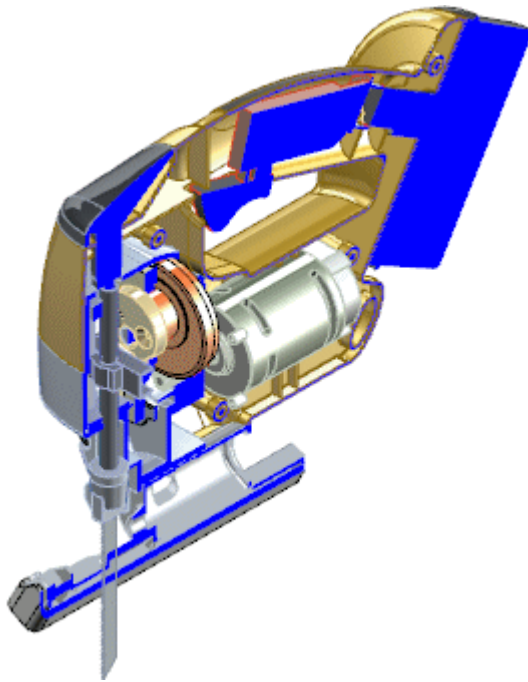
8. Select **Enable selection plane** to display a temporary plane that helps you select components that are not visible in the model or in the sectioned area.
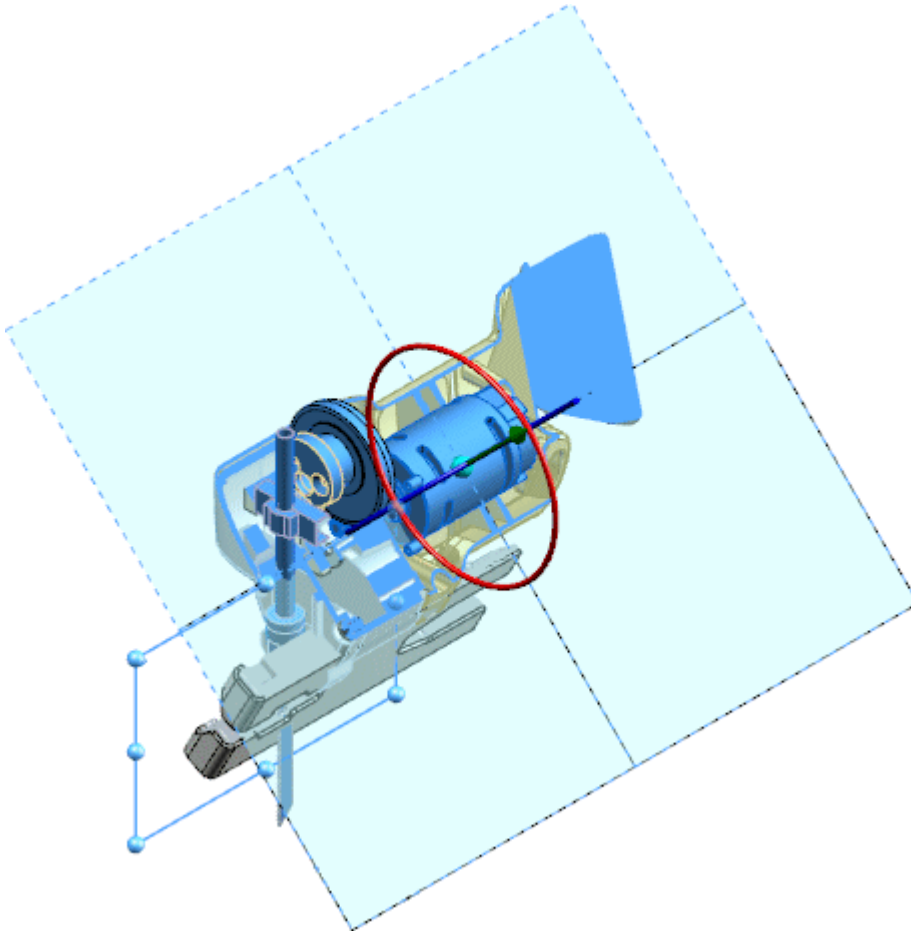
9. Drag the center ball of the triad to the right to view hidden components.
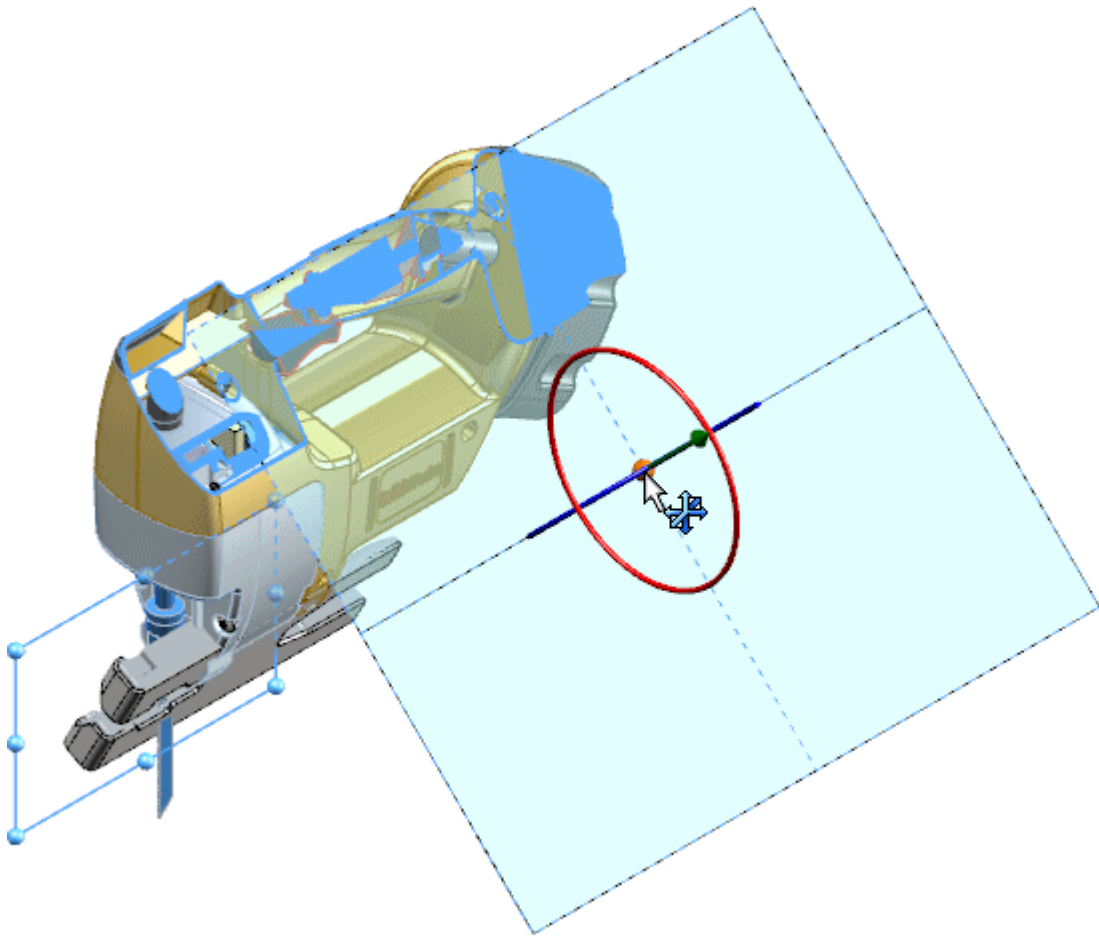
10. In the graphics area, click BEZEL - RIGHT to add it to the Selected components.
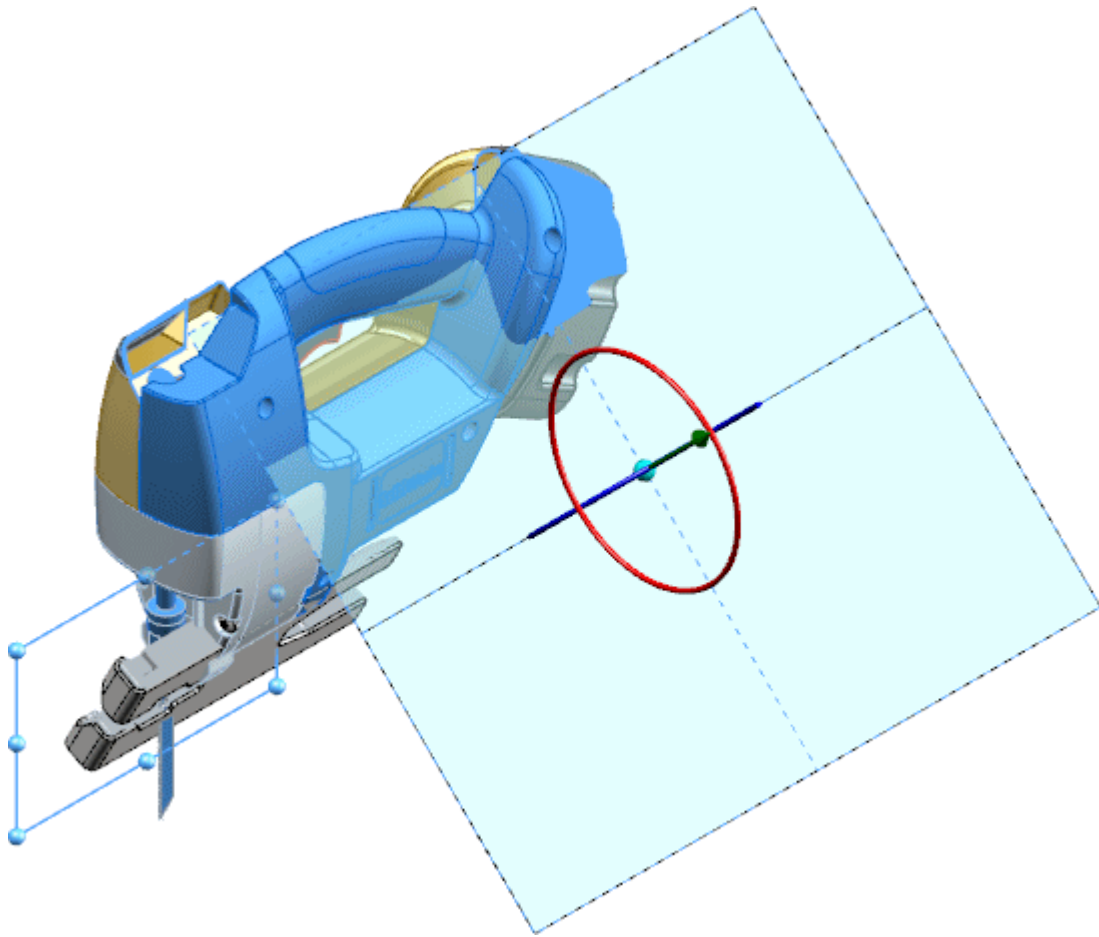
11. Click **Preview**.

12. Click ✔ .

> To edit the section view, right-click in the graphics area and select **Section View Properties**.

Solar Access Studies ⭐

With SolidWorks Professional, you can apply sunlight to SolidWorks models. Using solar access studies, you can simulate the movement of the sun as it passes over models of buildings, solar panels, and outdoor equipment.

You must include the following data when specifying sunlight for a model:

- The direction of north with respect to the model
- The geographic location of the model
- The time and date to simulate

You can create two types of solar access studies:

Fixed date, variable time   Displays the path of the sun on a specific date across a range of time.

Fixed time, variable date   Displays the position of the sun at a specific time of day across a range of dates.

Adding Sunlight to a Model

The sunlight you add to a model takes location, time, date, and atmospheric conditions into account.

To add sunlight to a model:

1. From a part or assembly, click **View** > **Lighting and Cameras** > **Add Sunlight**.
2. In the PropertyManager, for **Location, Time, and Date**, specify the options:
    a) For **North direction**, select a planar face or an edge.

    The north direction is perpendicular to a selected face, or collinear with a selected edge.

    > Click **Flip direction** to change the direction from which the sun is applied to the model.

    b) For **Location**, select a location from the list, or click **Specify Location** and enter values for **North latitude** and **East longitude**.
    c) Enter values for the time:

    Time zone from GMT   Specifies the time zone for the sunlight location relative to Greenwich Mean Time.

    Date   Specifies the date to apply the sunlight.

    Time of day   Specifies the time to apply the sunlight.
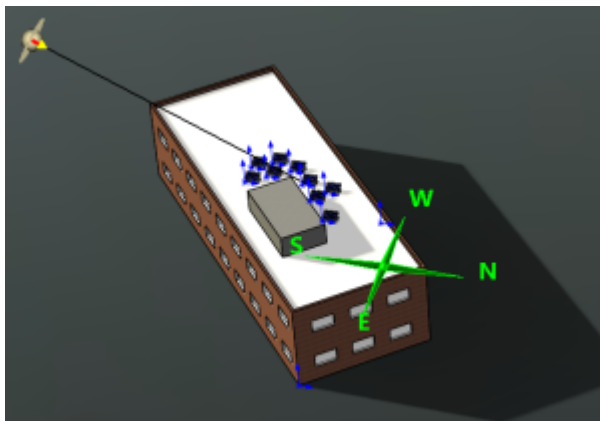
3. To set advanced settings such as haze, sun diameter, ground albedo, or sky gamma, click **Advanced**.

4. Click ✔ .

Creating Solar Access Studies

Using solar access studies, you can simulate how sunlight that is applied to a model changes over time. To create a solar access study, you must first add sunlight to a model.

To create a solar access study:

1. From a part or assembly, select the Motion Studies tab (lower portion of the graphics area).
2. Click **Animation Wizard** ⬚ (MotionManager toolbar).
3. In the wizard, select **Solar Access Study** and click **Next**.
4. Select the solar access study type:

Option	Description
Fixed date, variable time	Displays the movement of the sun over a time range on a selected date.
Fixed time, variable date	Displays the movement of the sun at a selected time of day over a range of dates.

5. Complete the rest of the wizard and click **Finish**.

6. To run the study, after creating the study, click **Play from Start** ⬚ (MotionManager toolbar).

   The speed of the animation depends on your settings. After you play the animation once, you can drag the timeline to view changes in the shadows as the sun moves.

Solar Access Study Tutorial

In this example, you optimize the placement of solar panels on the roof of a building.

The optimal location for roof-top solar panels is where they receive the most sunlight. After applying sunlight to a model of a building, you can create a solar access study to determine the placement of the solar panels. For this example, the building has an HVAC shed on the roof, which produces shade, and you cannot place the panels on the shed.
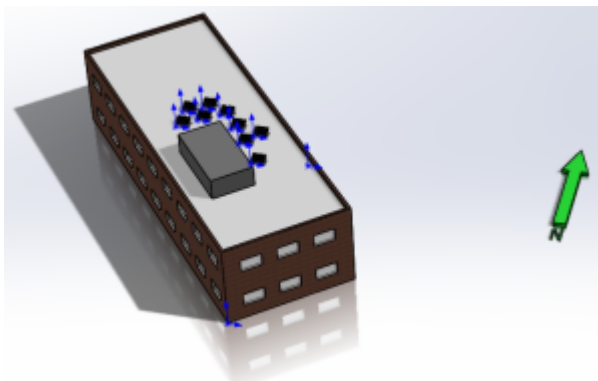
You perform the following tasks:

- Open a model of a building with solar panels on the roof.
- Apply sunlight to the model on the winter solstice.

- Create a fixed date, variable time solar access study.
- View the study results and adjust the solar panel positions to minimize shadows on the panels.

Opening the Model and Adding Sunlight

First, you open the model of the building and add sunlight.

1. Open *install_dir*\samples\whatsnew\fundamentals\solar_access\ building_and_solar_panels.SLDASM and save the model as my_building_and_solar_panels.SLDASM.

   The building has an HVAC shed and nine solar panels in arbitrary locations on the roof.
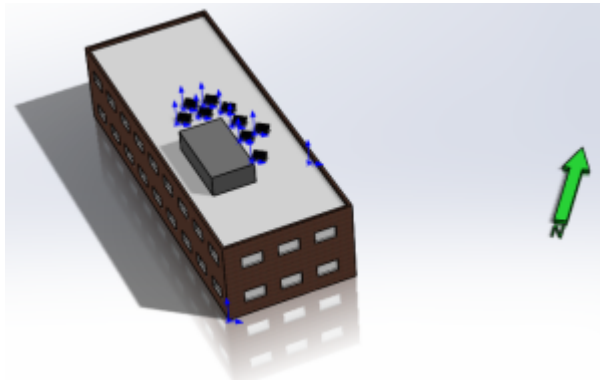
2. To turn on RealView graphics, click one of the following:

   - **View Settings** 🖥️ ▾ > **RealView Graphics** 🟡

   - **RealView Graphics** 🟡 (View toolbar)
   - **View** > **Display** > **RealView Graphics**.

3. To add sunlight, click **View** > **Lights and Cameras** > **Add Sunlight**.
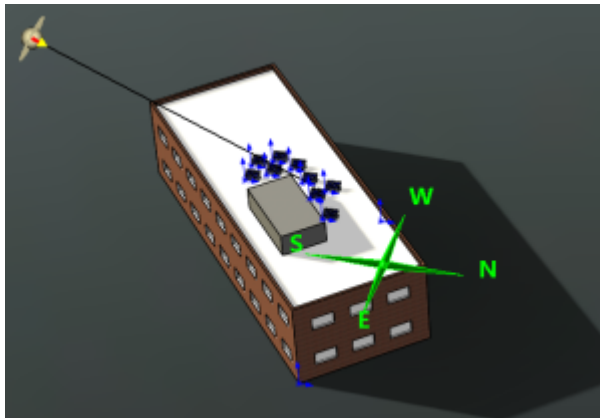
4. Under **Location, Time, and Date**, for **North direction**, select the north-pointing green vector located to the right of the model.

   The model includes the north vector as an aid for this example. For **North direction**, if you select an edge or line, the north direction is parallel to your selection. If you

select a planar face, the north direction is normal to your selection. You cannot select a direction that is normal to the scene floor.
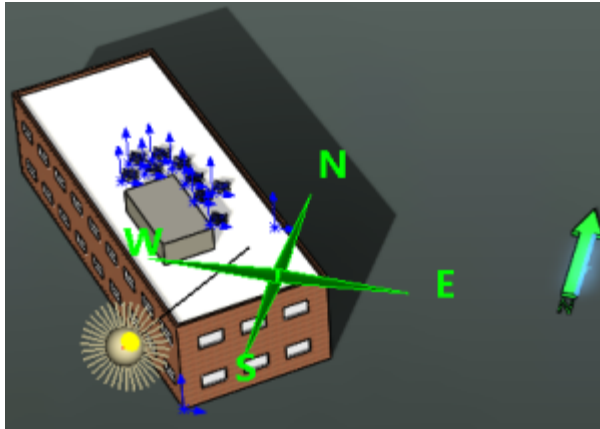
5. If north, **N**, points the wrong way, click **Reverse Direction** .
6. For **Location**, select **New York City, NY**.

7. For **Date**, enter `12/21/2013`, or the equivalent in your date format, and click .

    December 21 is the approximate date of the winter solstice in the northern hemisphere, the shortest day of the year. By optimizing for solar exposure on the day with the longest shadows, you can expect to optimize for solar exposure on all days of the year.
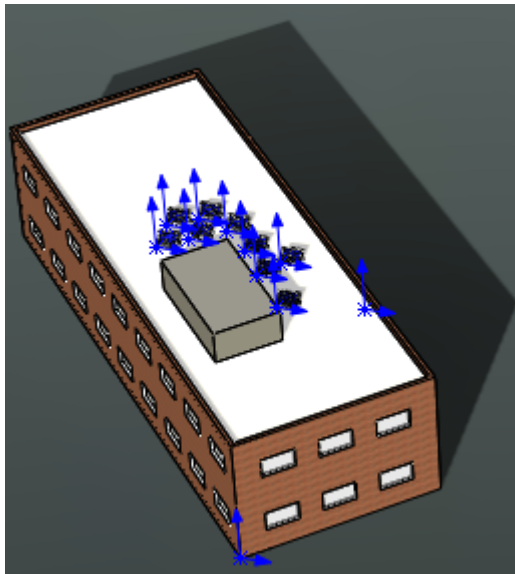
Creating a Solar Access Study

Next, you create a solar access study for the model.

1. Select the Motion Studies tab (lower portion of the graphics area).

2. Click **Animation Wizard** (MotionManager toolbar).

3. In the wizard, select **Solar Access Study** and click **Next**.

4. On the Select Solar Access Study Type page, select **Fixed date, variable time** and click **Next**.

    This type of study displays the movement of the sun over a time range on a selected date.

5. On the Fixed Date, Variable Time page, select the following and click **Next**:

    - For **Select the date**, enter `12/21/2013` or the equivalent in your date format.

      > This is approximately the shortest day of the year at the building location.

    - For **Select the time range**, select **Sunrise to sunset**.

6. On the Fixed Date, Variable Time page, for **Duration (seconds)**, type `10`, and click **Finish**.

    The duration specifies the duration of the animation you create.
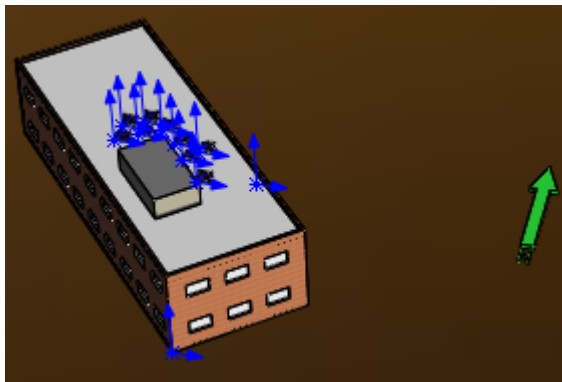
7. Click **Play from Start** ▶ (MotionManager toolbar).

    The speed of the animation depends on your settings. After you play the animation once, you can drag the timeline to view changes in the shadows as the sun moves.

Examining the Shadows

Next, you drag the timeline and examine the shadow exposure.

1. To zoom in on the roof area, click **Zoom to Area** 🔍 (View toolbar).

2. Drag the timeline and inspect the shadow movement throughout the day:

1 sec
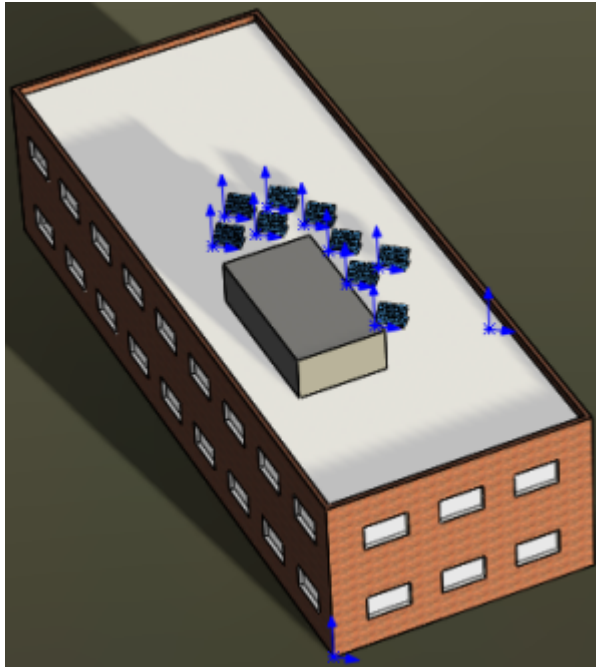
2 sec
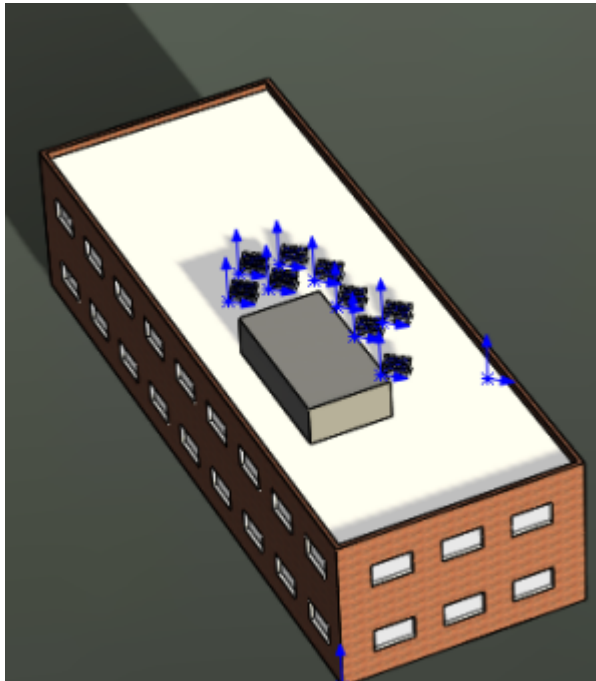
3 sec
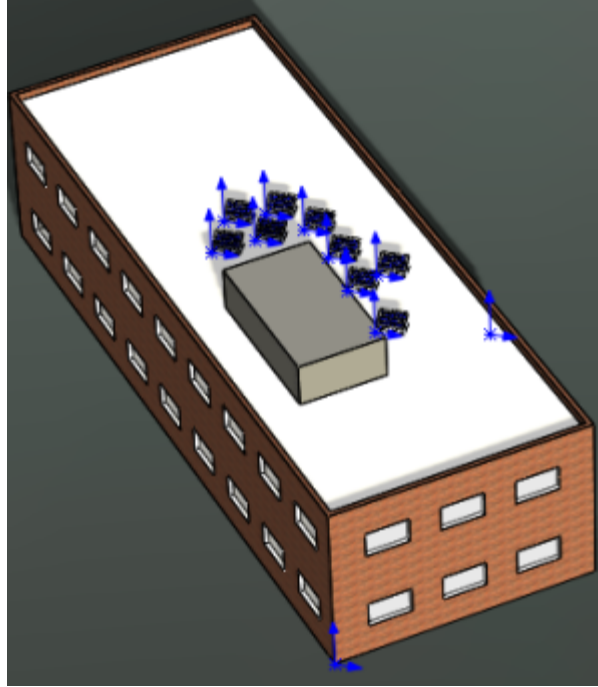
5 sec
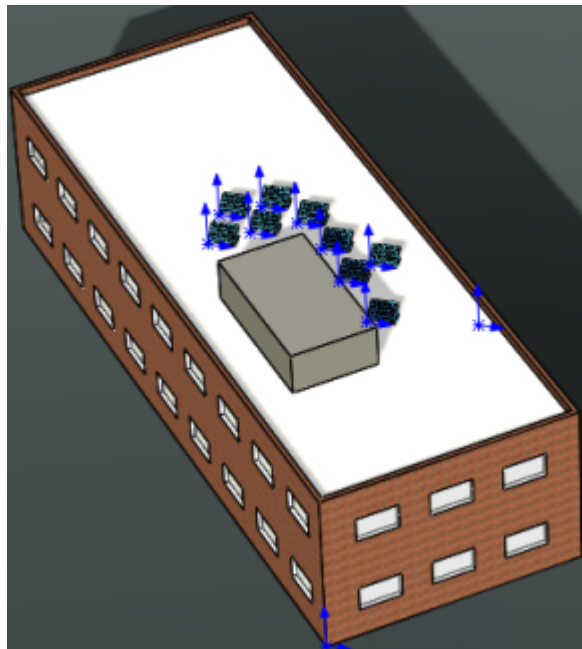
6 sec
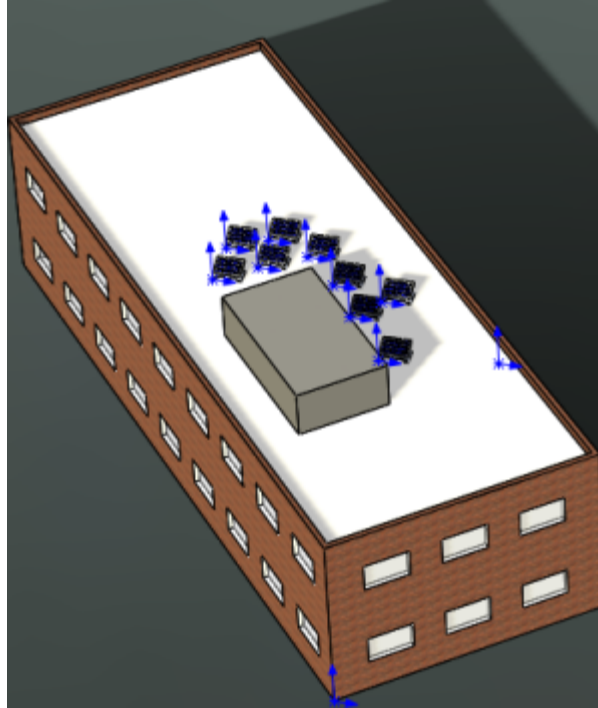
8 sec
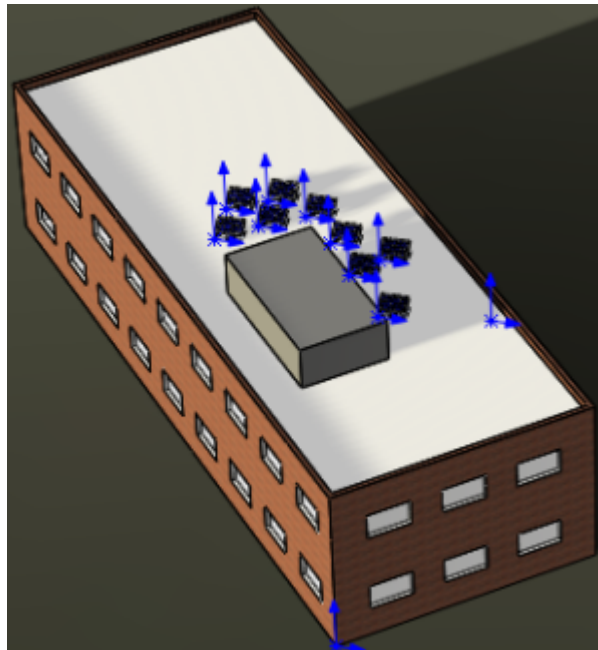
9 sec
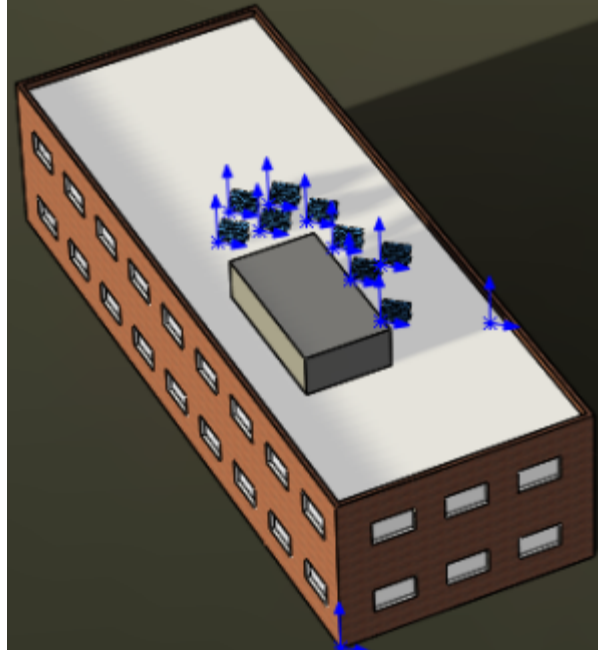
The best locations for the panels appear to be south-facing, away from the southwest corner of the building, and outside of the shadows of other panels and the HVAC.

Adjusting the Solar Panel Positions

Next, you adjust the solar panel positions to minimize the shadows.

1. Select the Model tab.

    You move the panels in the Model tab and test the solar study in the Motion Study 1 tab. If you move the panels in the Motion Study 1 tab, the panel position is modified only in the animation.

> You can click **Do no prompt again for this file** and click **No** if prompted to address missing images.

2. Drag the panels to minimize shadow over the course of the day.
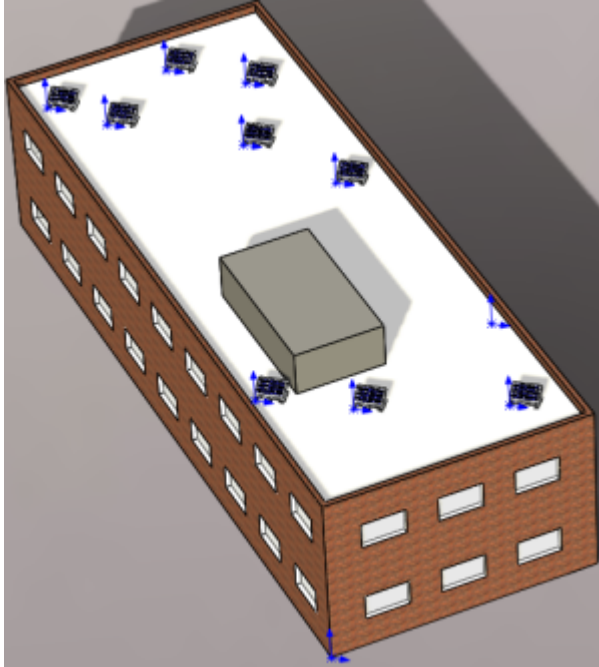
3. Select the Motion Study 1 tab, and click **Yes** when prompted to update the initial animation state.

4. Click **Play from Start** ▶ (MotionManager toolbar).

   The panel shadow exposure looks very good.

5. Save the model as `my_repositioned_building_and_solar_panels.sldasm`, and close the model.

Updating Assembly Graphics

You can set an option to update model graphics data for components when you save assemblies, to prevent display list data from becoming out-of-date.

To update assembly graphics:

1. Click **Options** 🔲 (Standard toolbar) or **Tools** > **Options**.
2. On the System Options tab, click **Assemblies** and select **Update model graphics when saving files**.
3. Click **OK**.

Opening Document Copies

After opening or modifying a document, you can save it as a copy and open the new copy automatically. References to the original document are not assigned to the copy.

To copy a document and open the copy:

1. With the document open, click **File** > **Save As**.
2. In the dialog box, select **Save as copy and open**.
3. For **File name**, type a new name.
4. Set other options and click **Save**.

Pack and Go Toolbox Option

You can include or exclude referenced Toolbox components when setting up Pack and Go.

Including Toolbox Components with Pack and Go

To include Toolbox components when you set up Pack and Go:

1. In SolidWorks, click **File** > **Pack and Go**.
2. In the dialog box, select **Include Toolbox components**.
3. Set the other options, and click **Save**.

Recent Documents ⭐

You can use the Recent Documents dialog box to open files with the same controls contained in the File Open dialog box.

You can also drag and drop a recent document into:

- An open part to create a derived part.
- An assembly to insert the component.
- A drawing to create a view of the model.
- An empty graphics area to open the file.
- Windows Explorer to copy the file to that location.

Browsing Recent Documents ⭐

To open a file from the Recent Documents dialog box:

1. Click **File** > **Browse Recent Documents** or press **R** to display the Recent Document browser.
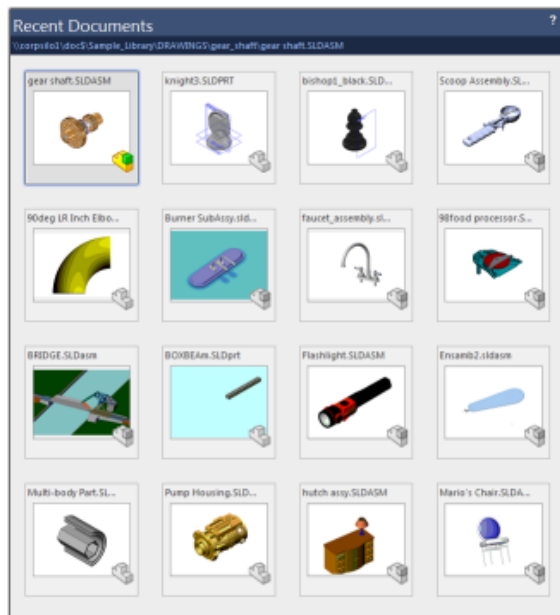
2. Hover over a document, click **Expand** ⌐ to set any of the options in the File Open dialog box, and click **Open** or **Open Read-only** to open the document.
3. Drag and drop a recent document tile into:
   - An open part to create a derived part.
   - An assembly to insert the component.
   - A drawing to create a view of the model.
   - An empty graphics area to open the file.
   - Windows Explorer to copy the file to that location.

> To close the browser without opening a document, click outside the browser or press **Esc**.

Version Support for DXF and DWG Files

You can import and export DXF and DWG files in AutoCAD® 2013 format.

Saving SolidWorks Files as Portable Network Graphics Files

You can save SolidWorks files as *.png files.

To save SolidWorks files as Portable Network Graphics files:

1. Click **File** > **Save as**.
2. In **Save as type**, select **Portable Network Graphics (*.png)**.
3. Optionally, for **Description**, type the text that describes your file.
4. Optionally, click **View PNG after saving**.
5. Optionally, click **Options** to select output options, and do the following:

   a) Select **TIF/PSD/JPG/PNG**.
   b) Under **Output as**, select **Image type**, **Remove Background**, and other options.
   c) Under **Print capture options**, set **DPI**, and **Paper size**.
   d) Click **OK**.

6. Click **Save.**

4
Installation

This chapter includes the following topics:

- **Installing SolidWorks Electrical**
- **Support for Environment Variables in the Option Editor**

Installing SolidWorks Electrical

SolidWorks Electrical is available from the SolidWorks Installation Manager.

You can choose to install, update, and remove SolidWorks Electrical using the SolidWorks Installation Manager or from a standalone installation package.

If you use the Installation Manager, you can choose to install:

- the client and server tools on the same computer or on different computers
- only the SolidWorks Electrical server tools

Previously, you could only install SolidWorks Electrical from a separate installation package.

Support for Environment Variables in the Option Editor

You can specify file and folder paths using an environment variable in the Administrative Image Option Editor.

All dialog boxes, that prompt for a file or folder path, include an option where you can assign a selected folder to an environment variable.

For example, if you select `C:\Program Files\SolidWorks Corp`, the file path is mapped to `%ProgramFiles%\SolidWorks Corp`.

The most common variables are `%ProgramFiles%` and `%SystemDrive%`.

You cannot convert all paths into environment variables. For example, `Z:\SolidWorks Data` cannot be mapped to an appropriate environment variable.

5
Assemblies

This chapter includes the following topics:

- **FeatureManager Structure for Large Design Review**
- **Including Rotations in Exploded Views**
- **Mates**
- **Mirroring Asymmetric Components**
- **Multiple Configuration Updates**
- **Patterns**
- **Selecting Components by Size**
- **Selecting Configurations from the Context Toolbar**
- **Shortcuts for Suppressing Items in Design Tables**
- **Toggling Between Flexible and Rigid States**
- **Updating Assembly Graphics**

FeatureManager Structure for Large Design Review

When you open a model in Large Design Review, the structure of patterns and assembly features are easier to view in the FeatureManager design tree.

The re-organization of these items include component and mirrored patterns and assembly features, such as cuts, holes, fillets, chamfers, well beads, belts, chains, and feature patterns.

You must save legacy assemblies in SolidWorks version 2014.

Including Rotations in Exploded Views ⭐

You can create exploded view steps that rotate a component with or without linear translation.

To create an exploded view in an assembly, click **Insert** > **Exploded View**.

To include a rotation in an exploded view step:

1. With the PropertyManager open, select the components to rotate in the explode step. The rotation and translation handles appear.
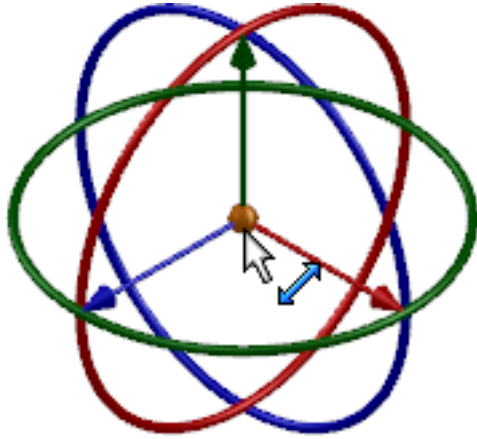
2. Select a rotational direction handle and rotate to a desired position.
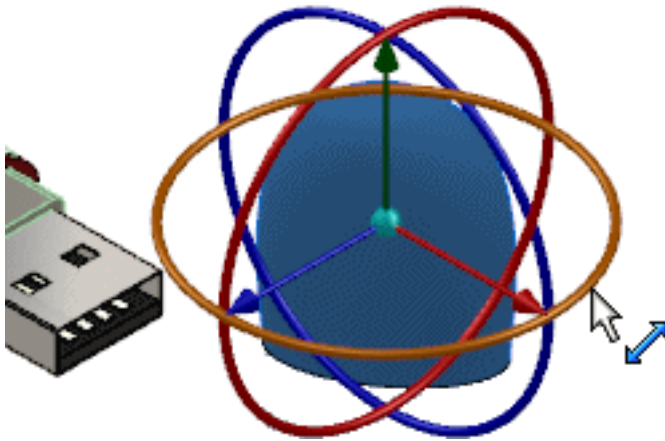
> You can include rotation and translation in the same explode step. You can also edit the explode step translation distance and rotation angle values in the PropertyManager.

3. To complete the explode step, click **Done**.

Viewing an Explode with Rotations

In this example, you open and explode an assembly that includes rotated exploded view steps. After you open the assembly, you can view the explode step details to see how to change values.

1. Open *install_dir*\samples\whatsnew\assemblies\explode_rotate\ usb_flash_drive_explode.SLDASM and save it as my_usb_flash_drive_explode.SLDASM.
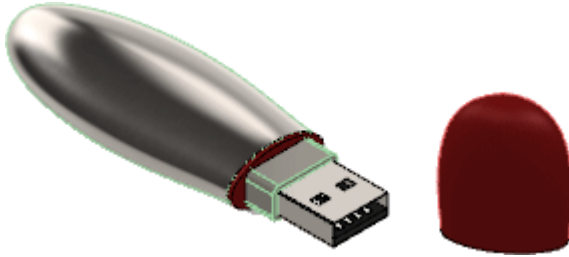
2. In the ConfigurationManager, expand the default configuration.

3. Right-click the exploded view ExplView1, and click **Explode**.

    Alternatively, to run an animation of the exploded view, you can click **Animate Explode**.

    The exploded view includes rotation and translation steps.

4. To view the details of the explode steps, right-click the exploded view and click **Edit Feature**.

    Since **Explode Step 3** includes translation and rotation, **Explode Distance** and **Explode Angle** have nonzero values. If you edit the explode step, you can edit these values.

5. Close the document without saving it.

Mates

Applying Standard Mates from a Context Toolbar ★

You can apply standard mates in an assembly from a context toolbar.

The toolbar appears when you press **CTRL +** select mating geometry in the graphics area. Only mates that are appropriate for your selections are available.
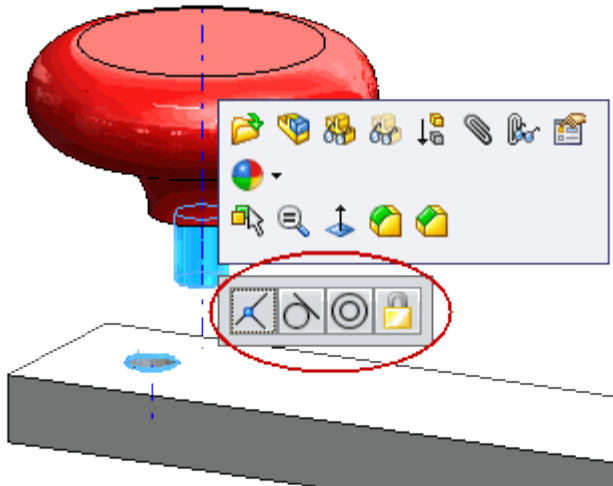
1. From an assembly, press **CTRL +** select entities to mate.
   The default mate is highlighted in the context toolbar.

2. Select a mate.

3. For distance or angle mates, enter the mate specification and click ✔.

Creating Slot Mates ✪

You can mate bolts to straight or arced slots and you can mate slots to slots. You can select an axis, cylindrical face, or a slot to create slot mates.
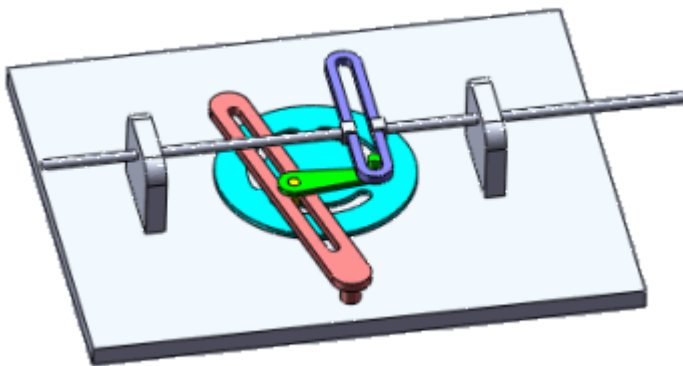
To create a slot mate:

1. Click **Mate** 🔗 (Assembly toolbar) or **Insert** > **Mate**.

2. In the PropertyManager, expand **Mechanical Mates** and select **Slot** 🔗.
3. For **Mate Selections**, select a slot face and the feature to mate to it:

- The face of another straight or angular slot
- An axis
- A cylindrical face

> You can select overlapping features.

4. Select a constraint option:

Option	Description
Free	Lets the component move freely in the slot.
Center in Slot	Centers the component in the slot.
Distance Along Slot	Places the component axis at a specified distance from the end of the slot.
Percent Along Slot	Places the component axis at a distance specified by the percentage of the slot length.

> To change the endpoint from which the distance is measured, select **Flip dimension**.

> For slot-to-slot mates, you can select only **Free** or **Center in Slot**.

5. Click ✔ .

Default Mate Types

When you select cylindrical faces or circular edges to mate with an axis, the software creates a concentric mate. When you select two parallel planar faces, the software creates a coincident mate.

> Some default mate types have a secondary default mate that is created when the primary default mate type coincident mate over-defines the assembly. For example, the software creates a parallel mate between two planar parallel faces when a coincident mate over-defines the assembly.

To mate an axis with a cylindrical face in an assembly:
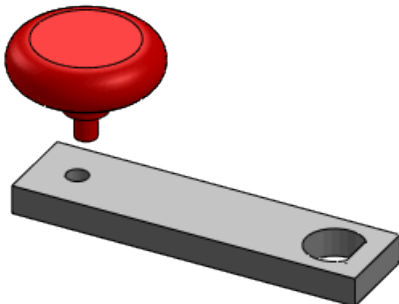
1. Select an axis and a cylindrical face or circular edge.

   To display temporary axes, expand **Hide/Show Items** [icon] (Heads-up View toolbar) and click **View Temporary Axes** [icon].

2. Click **Mate** [icon] (Assembly toolbar) or **Insert** > **Mate**.
   The mate defaults to a cylindrical mate.
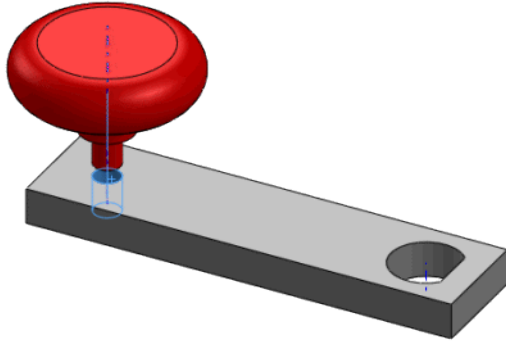
3. Complete the mate and click [icon] .

Including SmartMates in the CommandManager

You can include the **SmartMates** command when customizing the Assembly toolbar or the Assembly CommandManager.

To include the SmartMates command in the CommandManager:

1. Open an assembly.
2. Click **Tools** > **Customize**, or right-click the CommandManager and select **Customize**.
3. On the Commands tab, select **Assembly**.
4. Drag the **SmartMates** [icon] command to the Assembly CommandManager, and click **OK**.

Mirrored Components and Mates

Mirrored components include most advanced and mechanical mate types. In earlier releases, mirrored components included only mate types defined by two entities.

The following advanced and mechanical mate types are included in the mirror:

- Cam
- Hinge mates
- Linear coupler
- Path mates
- Symmetric mates
- Universal joints
- Width mates

Pinning the Mate PropertyManager

Clicking ✔ in the Mate PropertyManager saves your mate specification and closes the PropertyManager, unless you pin the PropertyManager open.

To specify more than one mate in the Mate **PropertyManager:**

1. Specify a mate.

2. In the PropertyManager, click 📌 before you click ✔ .

3. Specify the next mate and click ✔ .
4. Repeat Step 3 until you are done specifying mates.

5. To close the pinned PropertyManager, click ✖ .

Preventing Rotation in Concentric Mates

You can prevent the rotation of components that are mated with concentric mates by selecting the **Lock rotation** option. Locked concentric mates are indicated by an icon ◎ in the FeatureManager design tree.

> You cannot lock concentric mates when a referenced component is over-defined.

To lock a concentric mate in an assembly:

1. Open the Mate PropertyManager for a concentric mate:
   - For an existing mate, in the FeatureManager design tree, select the concentric mate and click **Edit Feature** 📷.
   - For a new mate, click **Insert** > **Mate**. Under **Standard Mates**, select **Concentric** ◎ and select the components to mate.

2. In the PropertyManager, select **Lock rotation** and click ✔ .

> To lock rotation on all concentric mates in an assembly, in the FeatureManager design tree, right-click the **Mates** folder and click **Lock Concentric Rotation**.

Setting the Sensitivity of SmartMates

You can set the speed at which the software applies SmartMates.

To set the SmartMate sensitivity:

1. Click **Options** 📋 (Standard toolbar) or **Tools** > **Options**.
2. Select **Performance**.
3. In **Assemblies,** for **SmartMate sensitivity**, drag the slider from left to right to decrease the speed.
4. Click **OK**.

Specifying Limit Mate Values in Design Tables

You can specify the lower and upper limit values for limit mates in design tables.

To specify limit mate values in a design table:

- Define columns for the limit mate and its upper and lower limits:

Option	Description
D1@LimitDistance1	Entering 1 in a cell specifies a limit mate for the table row.
$UPPERLIMIT_DISTANCE@LimitDistance1	Specifies the upper limit value for the limit mate.
$LOWERLIMIT_DISTANCE@LimitDistance1	Specifies the lower limit value for the limit mate.

Spherical and Curvilinear Mates ★

You can create mates using spherical bodies and surfaces.

You can create mates for spherical bodies with the following relationships:

- Tangential to nonlinear edges and sketches
- Tangent to nonplanar surfaces

You can create mates between elliptical, conical, and parabolic edges, sketch entities, and other geometry that are related as follows:

- Coincident to a point
- Tangent to a line, circle, cylinder, sphere, or ellipse
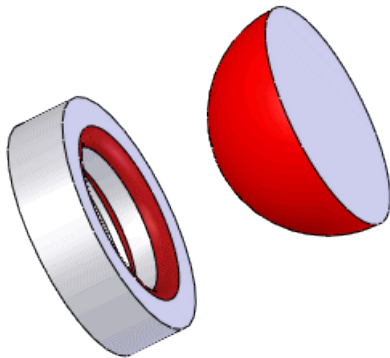
Creating Spherical Mates

In this example, you mate two spherical surfaces.
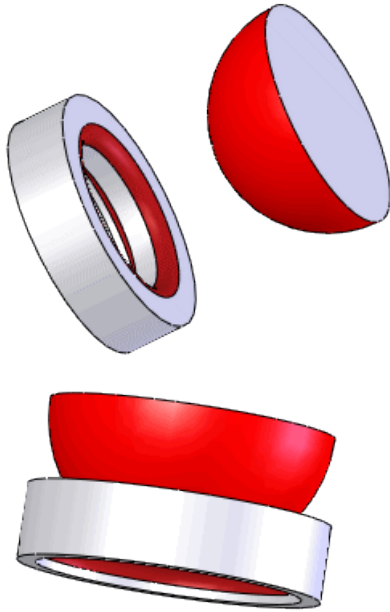
Opening the Assembly and Starting the Mates

First, you open an assembly, and then you set up a spherical mate.

1. Open *install_dir*\samples\whatsnew\assemblies\spherical_mates\ball
   ring.SLDASM and save it as my ball ring.SLDASM.
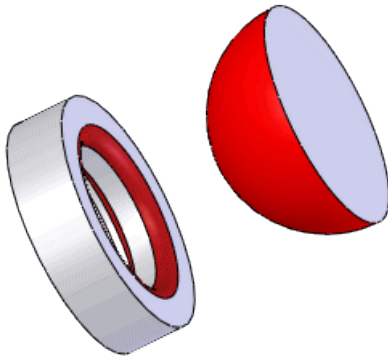
2. Click **Mate** 📎 (Assembly toolbar) or **Insert** > **Mate**.
3. Select the ring **Face<2>@ring-1** and the sphere **Face<1>@ball-1**.
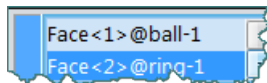
The default tangent mate is selected.
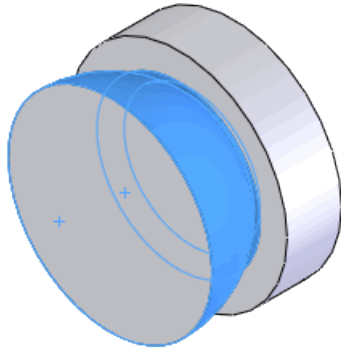
4. Click ✔.

5. Close the document without saving it.

Viewing Mates from Reference Geometry

You can open the **View Mates** window for any selected reference geometry, such as planes, axes, or other reference geometry.

To view mates for selected reference geometry:

1. Open an assembly.
2. Select mated reference geometry, for example, a mated plane or axis.
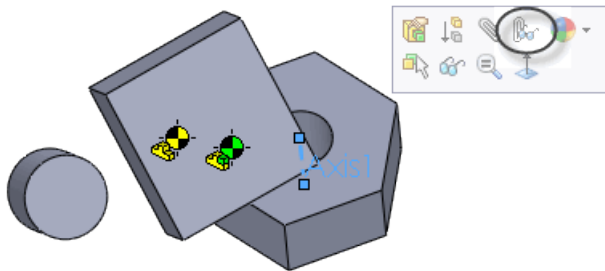
3. Click **View Mates** 🔧.

   The View Mates window lists mates that reference the selected reference geometry.

```
Axis1@Assem1-1/Part2-1                          ✕
┌─────────────────────────────────────────────────
│  ╱⊥  CoordinateSystem1 & Axis1 (Part2<1>,Coordi
│
```
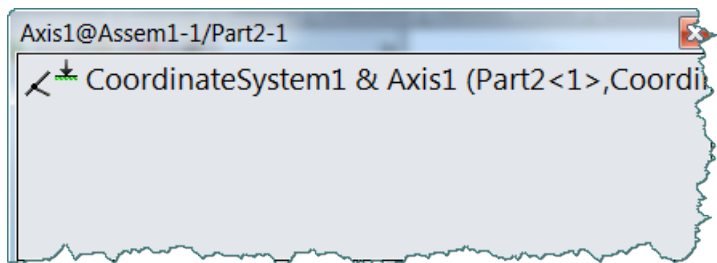
Mirroring Asymmetric Components

You can use a bounding box to position a mirrored component with respect to the mirror plane for mirroring asymmetric components.

> For mirroring asymmetric components, using bounding boxes to define the mirror plane provides better results than using the center of mass.

To mirror asymmetric components in an assembly:

1. Click **Mirror Components** ⬛ (Assembly toolbar) or **Insert** > **Mirror Components**.
2. In the PropertyManager, for **Selections**, select the **Mirror plane** and the

   Components to mirror and click **Next** 🔵 .
3. For **Mirror type**, select an option:

Option	Description
Bounding box	Positions the mirror so that the center of a bounding box for the selected components is mirrored about the mirror plane.
Center of mass	Positions the mirror so that the center of mass of the selected components is mirrored about the mirror plane.

4. Set other options and click ✔ .

Asymmetric Mirrored Components Examples

The following examples show the difference between a bounding box type mirror and a center of mass type mirror.

Bounding box type mirror

Center of mass type mirror
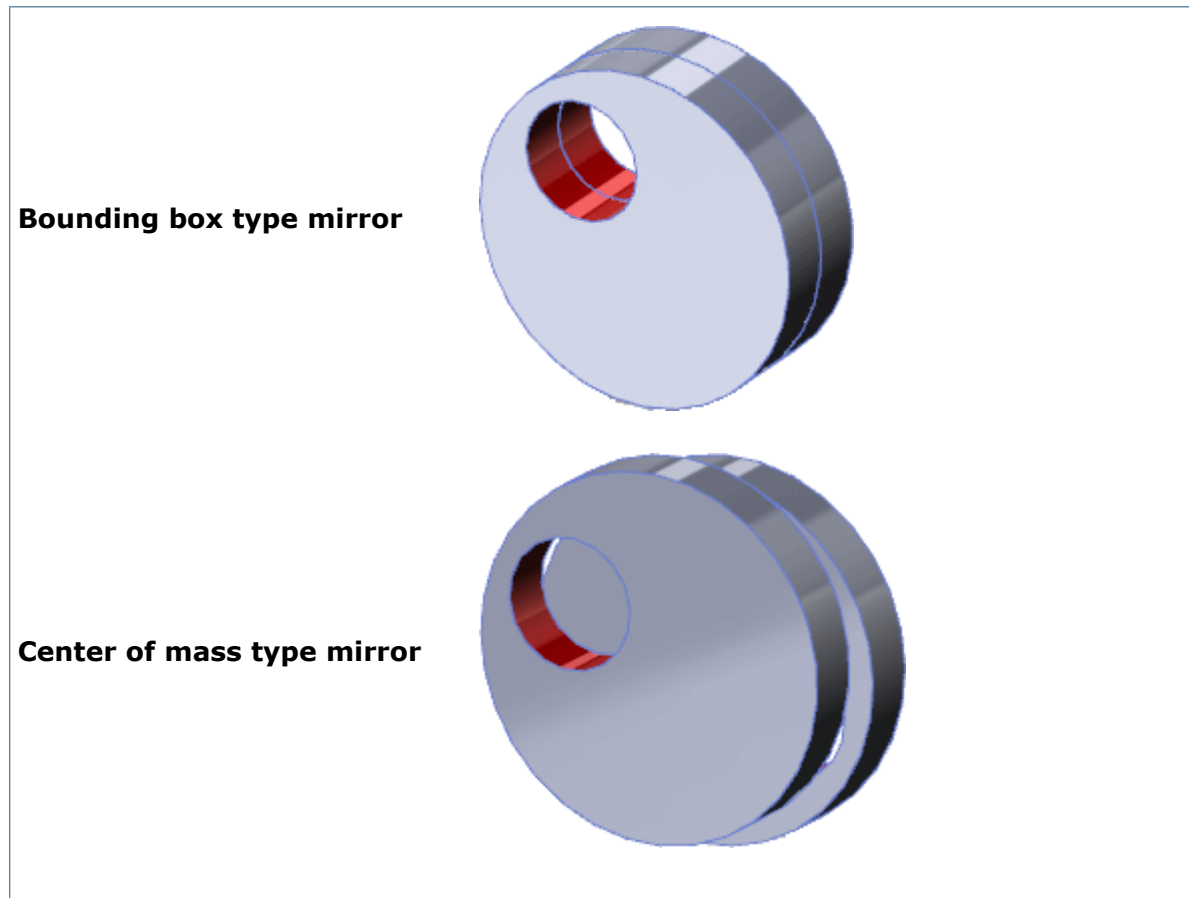
Multiple Configuration Updates

The software detects conflicts when an assembly contains subassemblies with a single-configuration component that is driven by an in-context relationship to a component with multiple configurations. The results appear in the AssemblyXpert dialog box.

The in-context relationships can arise from updates to equations or external references.

When the AssemblyXpert dialog box appears, to update the configurations, click **Fix It**.

Patterns

Curve and Sketch Driven Patterns

You can define curve driven and sketch driven patterns for assemblies.

Curve driven patterns can contain tangential lines to locate components driven by a curve. Sketch driven patterns can contain a centroid or a selected point to locate components driven by a sketch.

Previously, these patterns were available for parts only.

Click **Assembly** > **Insert** > **Component Pattern**, and select **Curve Driven** or **Sketch Driven** .

Pattern Driven Patterns

You can create a pattern using an underlying component pattern.

The **Feature Driven Pattern** has been renamed to **Pattern Driven Pattern** because it is no longer limited to features.

You can use a component, in addition to a feature, to drive a pattern driven pattern. The component can be in the same assembly or within a subassembly. The component can also be driven by an assembly-level pattern such as a Linear, Circular or Hole Wizard series, or by a feature driven pattern.

Click **Insert** > **Component Pattern** > **Pattern Driven** .

Selecting Components by Size

To select assembly components by size:

1. Click **Tools** > **Component Selection** > **Select by Size**.
2. In the dialog box, enter a value for the percentage of assembly size to select and click **OK**.

   To display a preview of your selections, select **Preview selection**.

   The components that are smaller than the percentage of assembly size you enter are selected.

Selecting Configurations from the Context Toolbar

When you select components or subassemblies, a context toolbar appears. When your selection has multiple configurations, you can change the configuration from the context toolbar.

To change the component configuration in an assembly:

1. Click the component or subassembly.
2. In the context toolbar, select a configuration from the list at the top.
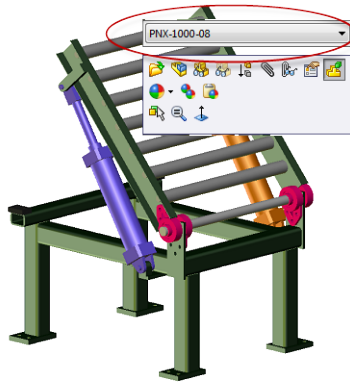
3. Click ✅.

Shortcuts for Suppressing Items in Design Tables

The string values `suppressed` and `unsuppressed` are valid values for selected design table columns. You can use shortcuts for these string values in design tables.

You can substitute the numerical value `1` for the string `suppressed`, and `0` for the string `unsuppressed` to achieve the same result. You can also enter the strings `s` for `suppressed` and `u` for `unsuppressed`. To enter shortcut values for `suppressed` or `unsuppressed` in design tables:

1. Type `1` or `s` for `suppressed`.
2. Type `0` or `u` for `unsuppressed`.

Toggling Between Flexible and Rigid States

You can toggle between the flexible and rigid states for subassemblies from the subassembly shortcut bar.

To make a rigid subassembly flexible:

1. Open an assembly containing a rigid subassembly.
2. In the FeatureManager design tree, click the rigid subassembly, and click **Flexible or Rigid** .

   You can also click **Flexible or Rigid** (Assembly toolbar).

   To make a flexible subassembly rigid, select the subassembly and click **Flexible or Rigid** .

> For models with multiple configurations, following this method affects only the active configuration.

Updating Assembly Graphics

You can set an option to update model graphics data for components when you save assemblies, to prevent display list data from becoming out-of-date.

To update assembly graphics:

1. Click **Options** [icon] (Standard toolbar) or **Tools** > **Options**.
2. On the System Options tab, click **Assemblies** and select **Update model graphics when saving files**.
3. Click **OK**.

6
CircuitWorks

Available in SolidWorks Premium.

This chapter includes the following topics:

Associating CircuitWorks Components with SolidWorks Models

You can associate and disassociate CircuitWorks® tree components with SolidWorks part or assembly files.

To associate a CircuitWorks component with a model:

1. Open an ECAD file in the CircuitWorks application.
2. From the CircuitWorks dialog box, in the CircuitWorks tree, expand the **Components** folder.
3. Right-click a component and click **Associate Model**.
4. In the Open dialog box, select the SolidWorks part or assembly file to associate with the component and click **Open**.

   To dissociate an associated component in the CircuitWorks dialog box, right-click the component and click **Dissociate Model**.

   > You are prompted to delete the dissociated SolidWorks model when you dissociate the model from a selected component.

CircuitWorks and SolidWorks Flow Simulation

With CircuitWorks, an add-in to SolidWorks Premium, you can create 3D models from the file formats written by most electrical computer-aided design (ECAD) systems. You

can collaborate to design printed circuit boards (PCBs) that fit and function in SolidWorks assemblies.

You can import the following CircuitWorks ECAD file PCB or component thermal properties to SolidWorks Flow Simulation:

- Dielectric and conductor density
- Specific heat
- Conductivity for PCBs
- Conductivity for volumetric heat sources from components

When you conduct thermal analysis with SolidWorks Flow Simulation, the SolidWorks model references the imported customer thermal properties.

Two options in the Flow Simulation tree facilitate setting up the Computational Fluid Dynamics (CFD) analysis for cooling simulation of electronic components:

- Right-click **Heat Sources** and select **Import volume source from model**. Select the heat sources to import in **Item properties**.
- Right-click **Printed Circuit Boards** and select **Import Printed Circuit Boards from model**. Select the PCBs to import in **Item properties**.

CircuitWorks Library Component Wizard

You can use the CircuitWorks Library Component Wizard to select SolidWorks components to add to the library and to set them up for use with CircuitWorks.

The wizard guides you to assign the component orientation and to specify how the CircuitWorks software determines the package name and part number from the component file name or its custom properties.

To open the CircuitWorks Component Library Wizard, from the CircuitWorks Component Library dialog box, click **File** > **Add Components**.

CircuitWorks Setup Wizard

You can use the CircuitWorks Setup Wizard to set up important CircuitWorks options for connecting your ECAD data to the CircuitWorks and SolidWorks applications.

The wizard guides you to set up a location for the CircuitWorks Library and to specify which ECAD file properties can be used to match to a SolidWorks model. You can also download SolidWorks models of electronic parts to use in circuit board design.

The CircuitWorks Setup Wizard starts the first time you add in CircuitWorks. You can also start the wizard from the CircuitWorks Options dialog box.

Exporting from SolidWorks to CircuitWorks

You can use the CircuitWorks Export Wizard to identify the circuit board features that make the resulting CircuitWorks model useful for export to an ECAD system.

To export a SolidWorks model to the CircuitWorks application:

1. From SolidWorks, click **Export to CircuitWorks** 💾 (CircuitWorks toolbar) or **CircuitWorks** > **Export to CircuitWorks**.

    You can also export a SolidWorks model by clicking **Import Model** 🔄 (Tools toolbar) in CircuitWorks.

    The CircuitWorks Export Wizard appears.

2. Follow the wizard to set up your circuit board model components for use with the CircuitWorks software.

    You can specify the following information:

Component Type	Specifies that the component is electrical, mechanical, or a circuit board.
Top of the Board	For a circuit board, specifies the feature that represents the top of the board.
Board Outline	For a circuit board, specifies the feature that represents the outline of the board.
Mechanical Outline	For a mechanical component, specifies the feature that represents the outline of the component.
Electrical Outline	For an electrical component, specifies the feature that represents the outline of the component.
Nonplated Holes	For a circuit board, specifies the features that represent nonplated holes.
Plated Holes	For a circuit board, specifies the features that represent plated holes.

| **Keep-out and Keep-in Areas** | For a circuit board, specifies the features that define keep-out or keep-in areas. |

3. Click **Finish**.

Importing Properties from ECAD Files

You can import ECAD file thermal and other properties as SolidWorks Model custom properties.

To import properties from an ECAD file into a SolidWorks model:

1. Open the ECAD file in the CircuitWorks application.

2. From the CircuitWorks application, click the CircuitWorks application menu , and then click **Options** .

    > You can also access CircuitWorks options from the SolidWorks application by clicking **CircuitWorks** > **CircuitWorks Options**.

3. Select **Properties** .
4. To add custom board properties to SolidWorks board models you build, select **Add these properties to the SolidWorks board model**.
5. To add custom component properties to the SolidWorks component models you build, select **Add these properties to the SolidWorks component models**.
6. Click **OK**.

Syncing Selected ProStep EDMD Changes

When syncing ProStep EDMD changes with your ECAD design system, you can select the changes to export. You can export sets of changes as an `.idz` file.

Before sending ProStep EDMD file information, you must enable ProStep EDMD file syncing in the CircuitWorks application.

To open a ProStep EDMD file in the CircuitWorks application, from SolidWorks, click **CircuitWorks** > **Open ECAD File**, select the file, and click **Open**.

To sync selected ProStep EDMD changes:

1. On the EDMD tab, click **Sync with ECAD** .
    The Sync with ECAD dialog box displays a summary of the messages to be sent with the ProStep EDMD data.

2. Select messages to send:

    -   To send most messages, click **Change** and clear the messages to withhold.
    -   To send a few messages, select clear **Change** and select the messages to send.

3. Click **OK**.

Exporting ProStep EDMD Changes as IDZ Files

To export ProStep EDMD changes as compressed `.idz` files:

1. To display the CircuitWorks Options dialog box, do one of the following:

   - From the CircuitWorks application, click the CircuitWorks application menu ,
     and then click **Options** .
   - From the SolidWorks application, click **CircuitWorks** > **CircuitWorks Options**.

2. In the CircuitWorks Options dialog box, click **ProStep EDMD** .
3. For **Default file format**, select ***.idz (compressed)** and click **OK**.

Specifying Component Names in the CircuitWorks Tree

You can specify how components are named in the CircuitWorks tree. For example, you can display the ECAD part number followed by the ECAD part name.

To specify component labels in the CircuitWorks tree:

1. To display the CircuitWorks Options dialog box, do one of the following:
   - In the SolidWorks application, click **CircuitWorks** > **CircuitWorks Options**.

   - From the CircuitWorks application, click the CircuitWorks application menu ,
     and then click **Options** .

2. Click **General** .
3. Under **CircuitWorks tree**, select options for **Show components in the tree using**:

Option	Description
First part	Specifies the first portion of the name by ECAD component name, or ECAD part number.
Delimiter	Specifies an optional character string to separate the first part of the name from the secondary part.
Second part	Specifies the optional second portion of the name.

4. Click **OK**.

Support for ProStep EDMD Files

You can import or export ProStep EDMD 3.0 files.

To open a ProStep EDMD 3.0 file:

1. From SolidWorks, click **Open ECAD File** (CircuitWorks toolbar) or **CircuitWorks** > **Open ECAD File**.
2. From the Open ECAD File dialog box, select the ECAD file and click **Open**.

3. To import the file into the SolidWorks application, close open SolidWorks models, and in the CircuitWorks dialog box, click **Build Model** and then click **Build**.
   The model is built and opened in the SolidWorks application.

4. To export the file back into the CircuitWorks application, from the SolidWorks software, click **Export to CircuitWorks** (CircuitWorks toolbar) or **CircuitWorks** > **Export to CircuitWorks**.

User Interface

You can multiselect items in the CircuitWorks tree and display or edit common properties. You can also access zoom and view options by right-clicking in CircuitWorks preview area.
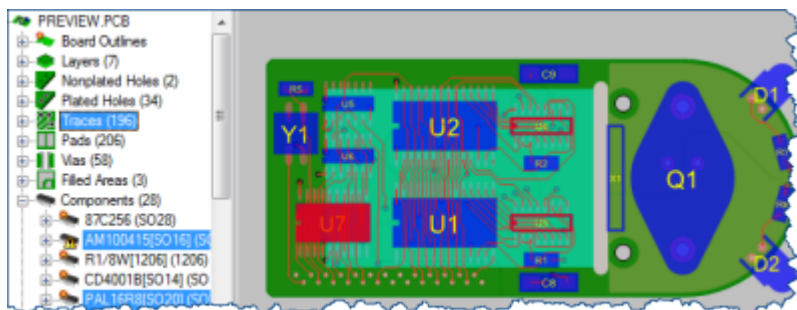
To access Zoom and View context menus:

1. From SolidWorks, click **Open ECAD File** (CircuitWorks toolbar) or **CircuitWorks** > **Open ECAD File**.
2. From the Open ECAD File dialog box, select the ECAD file and click **Open**.
3. In the CircuitWorks application, right-click the preview area.
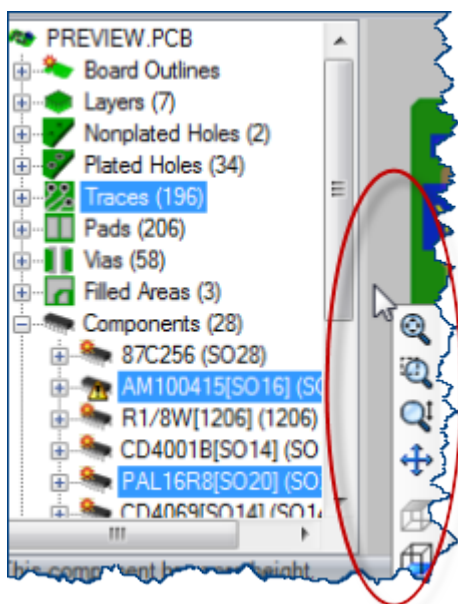
7
Configurations

This chapter includes the following topics:

- **Additional Properties for Toolbox Components**
- **Applying Configurations to Equations**
- **Selecting Configurations from the Context Toolbar**

Additional Properties for Toolbox Components

Three read-only properties, Part Name, Standard, and Specification, are available in Toolbox settings and appear in the Configure Component PropertyManager for standards that have an existing Designation, such as AS, DIN, GB, ISO, IS, and KS. The properties are available in drawing or assembly BOMs.

To display a Toolbox property:

1. Add a column to the BOM by double-clicking the top of a BOM column, just above the column heading.

    A dialog box appears with the **Column type** and **Property name** lists.

2. Select **TOOLBOX PROPERTY** from the **Column Type** list.
3. Select the property you want displayed in that column from the **Property name** list.

Applying Configurations to Equations

You can apply configurations to equations and global variables.

See **Configurable Equations**.

Selecting Configurations from the Context Toolbar

When you select components or subassemblies, a context toolbar appears. When your selection has multiple configurations, you can change the configuration from the context toolbar.

To change the component configuration in an assembly:

1. Click the component or subassembly.
2. In the context toolbar, select a configuration from the list at the top.
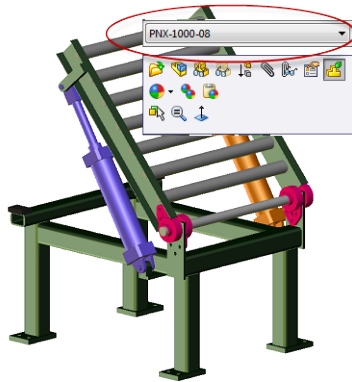
3. Click ✅.

8

SolidWorks Composer

SolidWorks® Composer™ software streamlines the creation of 2D and 3D graphical contents for product communication and technical illustrations.

These products are available:

- SolidWorks Composer
- SolidWorks Composer Check
- SolidWorks Composer Path Planning
- SolidWorks Composer Player
- SolidWorks Composer Player Pro
- SolidWorks Composer Sync
- SolidWorks Composer Enterprise Sync

This chapter includes the following topics:

- **Automatically Hiding Annotations**
- **Composer APIs**
- **Coordinate Labels**
- **Wrapping Annotation Text**

Automatically Hiding Annotations

You can automatically hide annotations when the anchor is hidden in the current scene.

In the Properties dialog box, under **Auto hide**, set the appropriate option.

Option	Description
Disable	Disables **Auto hide**.
Standard	Enables **Auto hide** if the annotation's anchor is hidden by any object, even the object linked by the annotation.
Advanced	Enables **Auto hide** if the annotation's anchor is hidden by any object other than the linked object. The number of annotations might impact system performance.

Composer APIs

Several new APIs have been added to the SolidWorks Composer API.

API	Description
CreateView()	Creates a custom view that captures only the camera's position.
CreateCameraView()	Creates a view that captures the entire viewport state.
DeleteView()	Deletes the specified view.
UpdateView()	Updates the specified view.

Coordinate Labels

Coordinates have properties that let you customize the label's text with the properties of the actor.

In the Properties dialog box, under **Text**, use the **Text** property to specify which property or meta-property appears as the coordinate label's text. Use the **Parent (level)** property to specify the actor whose **Text** value appears as the coordinate label's text.

These properties help you avoid creating multiple labels to display simple information.
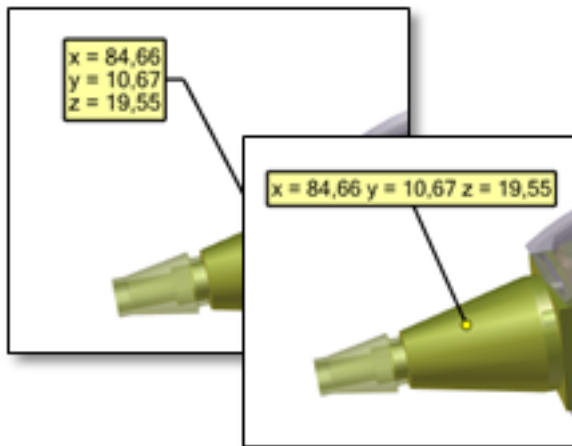
Wrapping Annotation Text

You can wrap text in annotation labels and callouts.

In the Properties dialog box, under **Text**, select **Wrap**. You can specify the **Wrap width** in millimeters. You can also drag the resize anchor in the text box to resize the width.

9
SolidWorks Costing

Available in SolidWorks Professional and SolidWorks Premium.

This chapter includes the following topics:

- **Converting Circular Cut Paths to Drilled Holes**
- **Costing Options**
- **Costing Reports**
- **Creating a Limited-access Costing Template**
- **Custom Properties for Costing**
- **Generating Costing BOMs**
- **Including Percent Scrap in Sheet Metal Costing**
- **Modifying Machined Part Setup**
- **Adding Template Comments to Drill, Mill, or Turn Operations**
- **Organization of the Setup Folder**
- **Setting Model Materials**
- **Simplifying Cost Estimates**
- **Updating Template Material Cost Data**
- **Using Multibody Parts to Define Removed Material**
- **Volume Features**

Converting Circular Cut Paths to Drilled Holes

To facilitate costing of sheet metal parts from machining templates, you can convert circular cut paths into drilled holes.

You can also revert drilled holes back to cut paths.

To convert a circular cut path to a drilled hole for a sheet metal part:

1. Click **Costing** (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.

2. In the Costing Task Pane, click **Costing Options** (upper left corner).

3. In the dialog box, select **Use machining template for sheet metal parts** and click **OK** twice.

4. To set up the cost estimate, in the Costing Task Pane, select the template and material.

> Keep **Plate** as the **Stock Body** selection.

5. In the CostingManager, expand the **Cut Paths** folder, right-click a circular cut path, and click **Change to Drill**.

> To revert the drilled hole to a cut path, expand the **Holes** folder, right-click the drilled hole, and click **Change to Cut Path**.

Costing Options★

You can define or override system-level costing options in the Costing Options dialog box.

You can select sheet metal Costing options to do the following:

- Estimate the cost of sheet metal parts from a machining template.
- Apply fixed custom costs to sheet metal features, such as bends, hems, or library features.

You can select machining Costing options to:

- Recognize removed material as volume features.
- Assign a default machining operation to volume features, or calculate the cost of volume features by assigning a per volume cost to the removed material.
- Assign the default surface finishing operation for milling and turning on machined parts.
- Override template settings for offset-based finishing. You can type a value for the offset from the machined surface for finishing and semifinishing operations.

To access the Costing Options dialog box, in the Costing Task Pane, click **Costing Options** (upper left corner).

Estimating Sheet Metal Part Costs from a Machining Template

To include the cost of plate geometries in sheet metal parts, you can use a machining template for sheet metal part costing.

To use a machining template for costing a sheet metal part:

1. Click **Costing** (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.

2. In the Costing task pane, click **Costing Options** (upper left corner).
3. In the dialog box, select **Use machining template for sheet metal parts**.
4. Set other options and click **OK**.

> You can select **Save as Costing default settings** to retain the Costing Options dialog box settings for future use.

Costing Reports

Customizing Costing Report Headers

When creating Costing reports, you can enter form data in the Costing Report dialog box, such as your company logo and contact information. You can also enter information about the customer quote you are preparing. Your entries appear in the report headers.

To customize report headers in a costing report:

1. Click **Costing** 🔲 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing** and calculate the cost of the part.

2. At the bottom of the Costing Task Pane, click **Generate Report** 🔲.

3. In the Report Options dialog box, expand **Company information**, select the options, and type the data.

Option	Description
Company	Specifies your company name.
Address	Specifies your company address.
URL	Specifies your company website.
Logo	Uploads a file of type `.jpg`, `.gif`, `.png`, or `.tif` to the report header. You can upload images of sizes up to 300 x 150 px.

> You can also enter your name and contact information on the report form.

4. Expand **Quote information**, select the options, and type the data.

   - **Quote number**
   - **Request for quote number**
   - **Customer company**
   - **Customer contact name**
   - **Customer contact information**
   - **Quote prepared by**

5. To view the published report, select **Show report on publish**.

6. Click **Publish**.

   The report appears.

7. Close the report.

Exporting Costing Reports as Microsoft Excel Files

You can export costing reports as either `.docx` or `.xlsx` files.

To export a costing report as a Microsoft `.xlsx` file:

1. At the bottom of the Costing Task Pane, click **Generate Report** ![icon].
2. In the dialog box, for **File type**, select **.xslx**.
3. Select **Show report on publish**.
4. Set other options and click **Publish**.
   The report appears in Microsoft Excel.
5. Close the report.

> To generate a Microsoft Word file, select **.docx** for **File type**.

Creating a Limited-access Costing Template ★

You can create a limited-access Costing template to share with other users. You can select limited-access templates in the Costing Task Pane, but you cannot view or edit them in the Costing Template Editor application.

To create a limited-access Costing template for a part:

1. To open a Costing template, do one of the following:

   - From the Windows **Start** menu, click **All Programs** > **SolidWorks *version*** > **SolidWorks Tools** > **Costing Template Editor *version***.
   - With a part open:

     1. Click **Costing** ![icon] (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.
     2. In the Costing Task Pane, select a template to save, and click **Launch Template Editor**.

2. In the **Costing Template Editor** application, in an open template, click **Save** ![icon] > **Save As Limited Access Copy**.

> You might have to click **Save As** ![icon] instead of **Save** ![icon].

3. In the Save As dialog box, for **File name**, enter a name and click **Save**.
   The template is saved as a protected `.zip` file.
4. Close the **Costing Template Editor** application.

Custom Properties for Costing

You can assign cost-specific custom properties to SolidWorks documents.

Cost - Material Name	Specifies the type of material used when costing the part.
Cost - Stock Type	Specifies the type of stock used to cost the part.
Cost - Template Name	Specifies the name of the template used to estimate the part cost.
Cost - Material Cost	Specifies the calculated cost of material used.
Cost - Manufacturing Cost	Specifies the calculated manufacturing cost.
Cost - Time of Cost Calculation	Specifies the time at which the cost was most recently calculated.
Cost - Stock Size	Specifies the size of the stock used to manufacture the part.
Cost - Total Cost	Specifies the calculated cost of the part.

For more information, see *SolidWorks Help*: *Adding Costing Information to File Properties*.

Generating Costing BOMs

You can generate assembly or drawing BOMs that display component costs.

To generate a costing BOM:

1. Click **Insert** > **Tables** > **Bill of Materials**.
2. Select the Costing BOM template `bom-costing.sldbomtbt`.

3. Select other options and click ✔ .

Including Percent Scrap in Sheet Metal Costing

To include scrap material in the cost of a sheet metal part, you can designate a percentage of the part as scrap.

To designate a portion of a sheet metal part as scrap:

1. Click **Costing** 🖼 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.
2. In the Costing Task Pane, under **Blank Size**, type a percentage value for **Percent scrap**.

    The value you type for **Percent scrap** designates a portion of the material defined in your **Area to cost** selection as scrap material.

Modifying Machined Part Setup

You can specify machined part setup time on the Machine tab of the machining template.

For each machine, you can specify the following:

Load/Unload Time   Specifies the time it takes to load a part into a machine and then unload it for each setup operation.

Operation Setup Time   Specifies the time it takes to set up fixtures and other machining process setup operations.

Setup Distribution   Specifies options for applying the operation setup time to costing.

To modify setup parameters for machined parts:

1. To open a Costing machining template, do one of the following:

   - From the Windows **Start** menu, click **All Programs** > **SolidWorks *version*** > **SolidWorks Tools** > **Costing Template Editor *version***.
   - With a part open:

     1. Click **Costing** (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.
     2. In the Costing Task Pane, select a template, and click **Launch Template Editor**.

2. If your machining template is not open, in the **Costing Template Editor** application, click **Open** , select the machining template and click **Open**.

3. Select the Machine tab, and, for each machine, enter data in the appropriate columns.

   > To set the setup operation parameters for cutting operations, under operations, select **Cut (Plate Stock)**.

4. Save the template and close the application.

Adding Template Comments to Drill, Mill, or Turn Operations

You can add comments to mill, drill, or turn tools in the machining templates.

To add comments to the machining templates:

1. To open a Costing machining template, do one of the following:

   - From the Windows **Start** menu, click **All Programs** > **SolidWorks *version*** > **SolidWorks Tools** > **Costing Template Editor *version***.
   - With a part open:

1. Click **Costing** 🪙 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.
2. In the Costing Task Pane, select a template, and click **Launch Template Editor**.

2. If your machining template is not open, in the **Costing Template Editor** application, click **Open** 📂, select the machining template and click **Open**.
3. Select the Drill, Mill, or Turn tab, and type comments in the **Comments** column.
4. Save the template and close the application.

Organization of the Setup Folder ✪

The CostingManager Setup folder for machined parts has been reorganized to display setup costs for operation setup, custom setup, and loading or unloading parts.

Operation Setup includes fixturing, setup, and programming.

Custom Setup includes painting, anodizing, or other custom operations.

Load and Unload Setup includes the loading and unloading of parts into and out of fixtures.
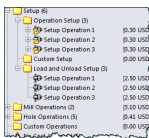
Setting Model Materials

You can set the model material for a part from the material you select in the Costing Task Pane.

To set the material for a part from Costing Task Pane selections:

1. Click **Costing** 🪙 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.
2. In the Costing Task Pane, under **Material**, select the values for **Class** and **Name**.
3. To set your material selections as the model material, click **Set Material**.

> Part material settings are automatically selected in the Costing Task Pane when the part material is also a material listed in the selected template.

Simplifying Cost Estimates ✪

You can estimate the cost of machined parts, relying on a selected template only for material selection.

To simplify the cost estimate, select the following machining options in the Costing Options dialog box, and calculate the cost of the removed material:

Use customizable volume feature recognition	When applied to the material to be removed from a machined part, specifies that volume features determine the amount of material to be removed.
Cost per volume removed	Lets you assign a fixed cost per unit volume of removed material, rather than calculate the cost based on tool sizes and removal rates.
Override template settings for offset-based finishing	Calculates the per-unit-volume cost of semifinishing and finishing, based on an offset distance value rather than on template data. The offset distance value is the operation offset distance from the machined surface.

To set up options for simplified cost estimates for a part:

1. Click **Costing** (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.

2. In the Costing Task Pane, click **Costing Options** (upper left corner).

3. In the dialog box, for **Machining options**, select the following:

   a) Under **Removed material processing**, select **As customizable volume feature recognition**.

   b) Under **Volume feature calculation**, select **Cost per volume removed** and type cost values for **Roughing**, **Semi-finishing**, and **Finishing**.

   c) Under **Surface finishing for mill or turn operations**, select **Override template settings for offset-based finishing** and type values for **Semi-finishing** and **Finishing**.

4. Set other options and click **OK** twice.

Updating Template Material Cost Data

You can export machining template material property data to an `.xlsx` file and modify the cost data in the spreadsheet. You can upload the material cost data from the `.xlsx` file to the Costing machining template.

To update material cost data in a Costing machining template:

1. To open a Costing machining template, do one of the following:

   • From the Windows **Start** menu, click **All Programs** > **SolidWorks *version*** > **SolidWorks Tools** > **Costing Template Editor *version***.
   • With a part open:

      1. Click **Costing** (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.
      2. In the Costing Task Pane, select a template, and click **Launch Template Editor**.

2. If your machining template is not open, in the **Costing Template Editor** application, click **Open** , select the machining template and click **Open**.

3. Select **Stock Material** and click **Export**.

4. To save the material properties data as an `.xlsx` file, type a name and click **Save**.

   The file opens in Microsoft Excel, where you can modify the cost data.

   > You must not add rows or columns to the spreadsheet.

5. To import the updated material cost data:

   a) In the machining template, select **Stock Material** and click **Update**.

   b) Select the edited `.xlsx` file and click **Open**.

   Updated costs are highlighted in the template **Cost** column.

6. Save the template and close the application.

Using Multibody Parts to Define Removed Material ⭐

You can use multibody parts to define the removed material in a part.

You can assign operation costs to bodies that define the removed material. Alternatively, you can assign a cost per volume removed. For example, you can compute the cost of post manufacturing operations on cast parts by assigning operation costs to the bodies.

To assign operation costs or the cost per volume removed to a multibody part:

1. Click **Costing** 🪙 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.

   The default multibody template is selected in the Costing task pane. Default sheet metal and machining templates are in effect, depending on the type of body.

2. Assign a custom operation to the bodies:

   a) Under **Body List**, select the bodies to assign a custom operation to.

   b) For **Type of body**, select **Custom/Removed Material**.

   c) Under **Operation Options**, select **Custom operation** and select a drill, turn, or mill operation.

   The selected machining template assigns the drilling, turning, or milling costs.

3. Apply cost per volume to the bodies:

   a) Under **Body List**, select the bodies to assign the custom cost per unit volume to.

   b) For **Type of body**, select **Custom/Removed Material**.

   c) Under **Operation Options**, select **Custom cost per volume** and type a value.

   > You can assign different costs per volume for different sets of bodies in a multibody part.

4. After completing the Costing setup for all of the bodies, you can begin or update the cost estimation.

Estimating Post-machining Costs for Cast Parts

When you finish a cast part, you might have costs associated with post-machining operations, such as drilling or milling. In this example, you open a cast part, define the removed material, and estimate the cost of removing material from the cast part.

Cast Stock Part

Removed Material
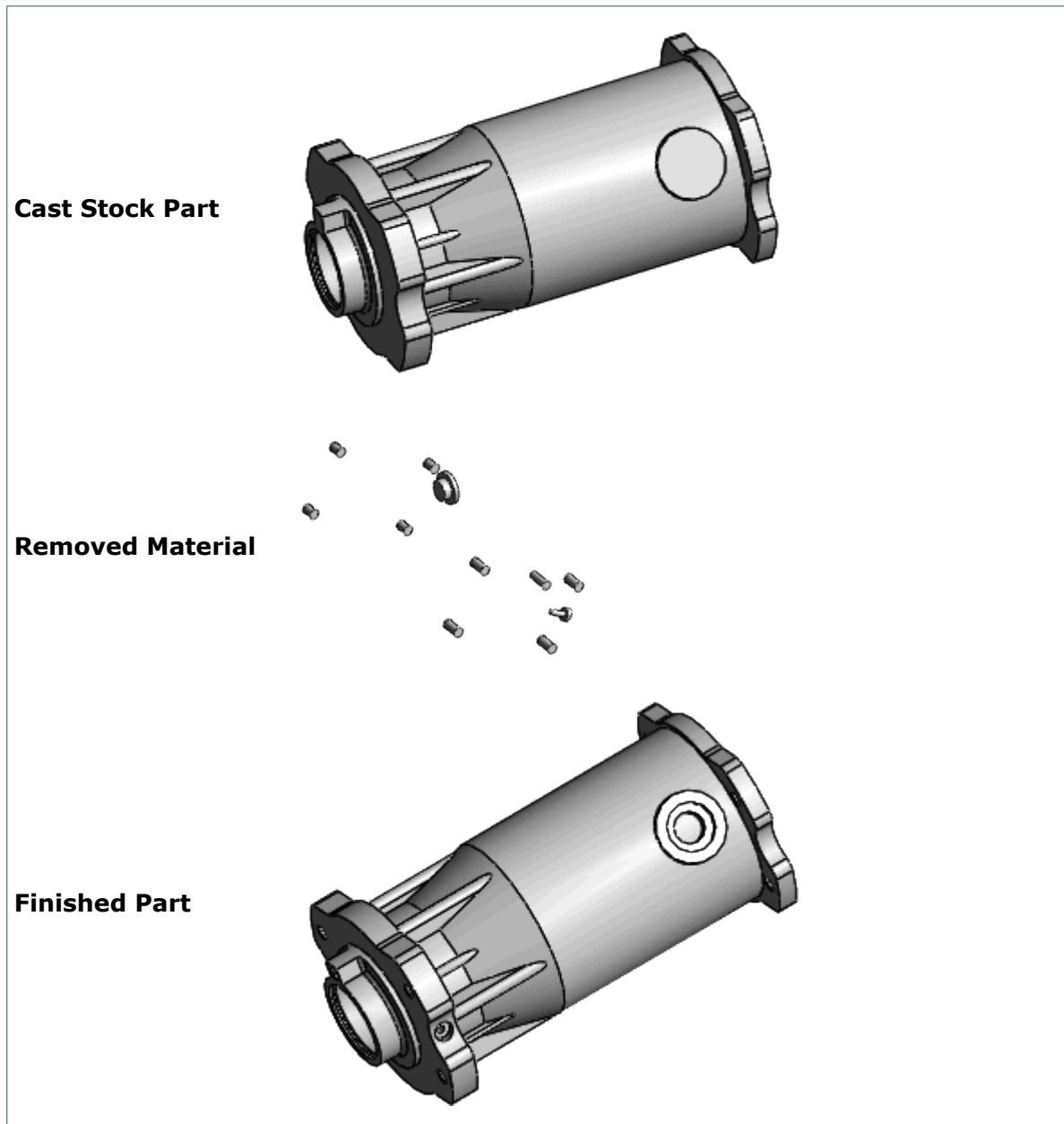
Finished Part

To estimate the cost of the finished part, you can combine cost estimates of these two parts:

- The cast aluminum stock part
- The material that is removed from the cast part during post-machining operations

In this example, you estimate the cost of removing material from the cast part.

Since finishing the cast part entails removing material, you can estimate the cost of finishing a cast part by estimating the cost of removing material from the cast part, either by assigning a milling or drilling operation for the removal, or by assigning a cost per volume of material removed. With either method, you first define a multibody part to represent the material to be removed from the cast part.

Opening the Parts

First, you open two parts: A finished cast part, and a part that represents the cast stock.

1. Open *install_dir*\samples\whatsnew\costing\cast_part_volume_removed\ CAST_CYLINDER.SLDPRT.

    This is a finished cast part with some drill and mill finishing operations.
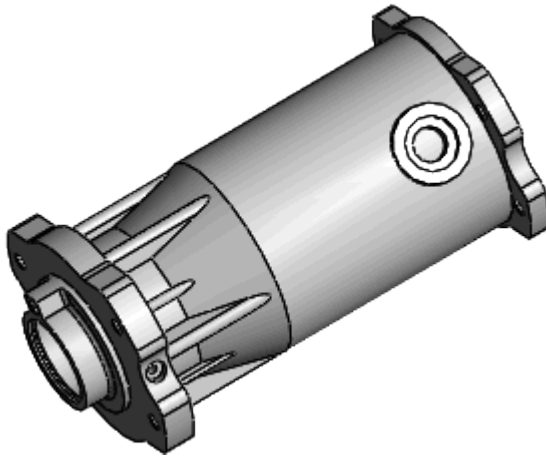
2. Save the part as my_cast_cylinder.sldprt.

3. To view the part used to represent the stock body for the cast part, open *install_dir*\ samples\whatsnew\costing\cast_part_volume_removed\ CAST_CYLINDER_STOCK.SLDPRT.

    This is the cast part with the drilled holes and milled portions suppressed.
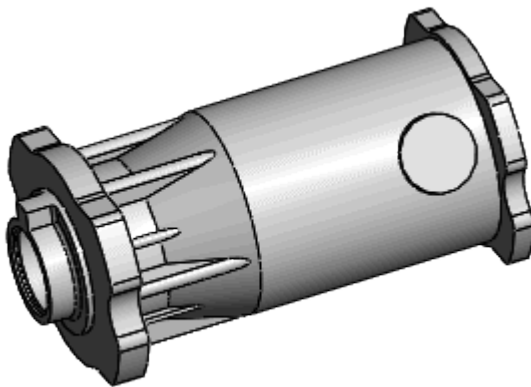
4. Close *install_dir*\samples\whatsnew\costing\cast_part_volume_removed\ CAST_CYLINDER_STOCK.SLDPRT.

Defining the Removed Material

Aside from setup costs, the cost of the material removed during finishing operations determines the cost of the finishing operations. Next, you define a multibody part for the material removed during post-machining operations.

One way to define a multibody part for the removed material is to subtract the stock body for the cast part from the finished part. What remains is the volume of material removed during the post-machining operations.

1. From the ConfigurationManager ![icon] for `my_cast_cylinder.sldprt`, right-click **Default [my_cast_cylinder]** and click **Add Derived Configuration**.
2. In the PropertyManager, for **Configuration name**, type `Costing Example`, and click
   ![checkmark] .
3. Add the stock part to the finished part:
   a) Click **Insert** > **Part**.
   b) Select *install_dir*`\samples\whatsnew\costing\cast_part_volume_removed\`
      `CAST_CYLINDER_STOCK.SLDPRT`, and click **Open**.
4. Drag the stock part to the finished part and click the bodies when they match up.
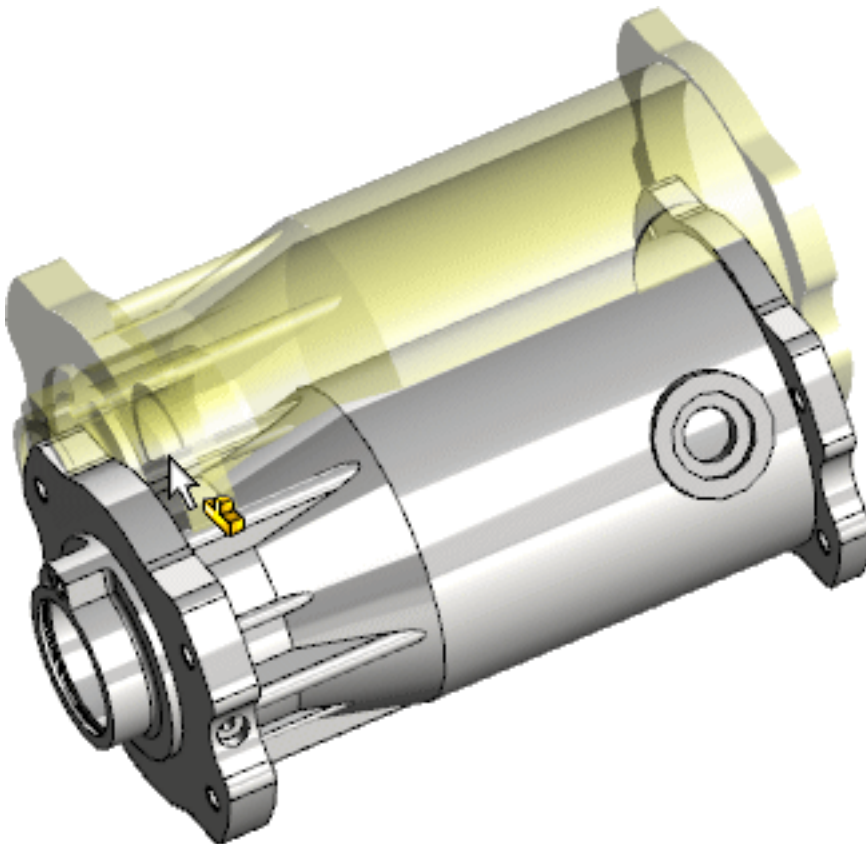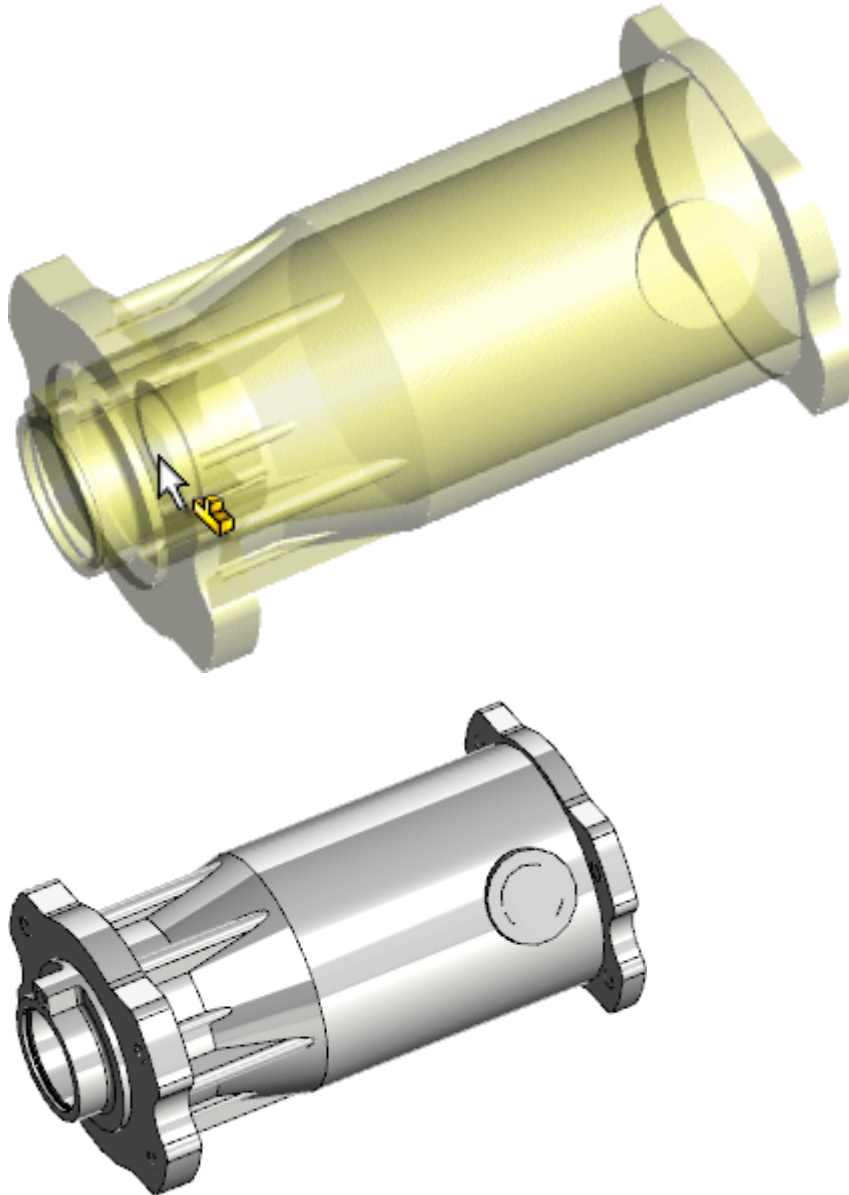
5. Click **Insert** > **Features** > **Combine**.
6. In the PropertyManager, for **Operation Type**, select **Subtract**.
7. Subtract the two bodies to define the part for the volume of removed material:
   a) In the FeatureManager design tree, expand **Solid Bodies (2)**.
   b) For **Main Body**, select **<CAST_CYLINDER_STOCK>**.

c) For **Bodies to Subtract**, select **Flange Holes**, and click ✔ .

d) In the Bodies to Keep dialog box, select **All bodies**, and click **OK**.

Calculating Removed Material Costs

Next, you calculate the cost of the removed material.

1. Click **Costing** 📇 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.

   In the Costing Task Pane, the software selects the default multibody template, which references both a sheet metal and a machining template.

2. Apply a fixed cost to the bolt-shaped body:

   a) In the Costing Task Pane, under **Body List**, select a body, such as **Combine1[2]**.

   > Your selection might differ, depending on how you set up the bodies.

   b) For **Type of body**, select **Custom/Removed Material**.

   c) Under **Operation Options**, select **Custom cost** and type 2.00.

3. Assign a custom drill operation to the largest body:
   a) Under **Body List**, select another body, such as **Combine1[1]**.
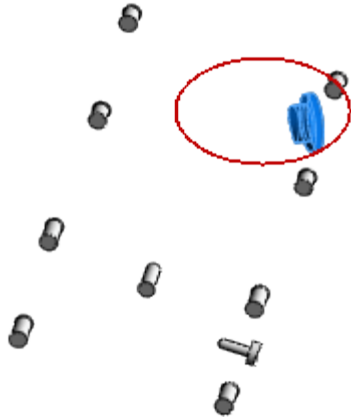
   b) For **Type of body**, select **Custom/Removed Material**.
   c) Under **Operation Options**, select **Custom operation** and select **HSS Drill**.
   d) Under **Material**, for **Class**, select **Aluminum Alloys**.
      The software derives operation costs for the material from the machining template.

4. Apply cost per volume removed to the remaining bodies:
   a) Under **Body List**, select all of the remaining bodies.
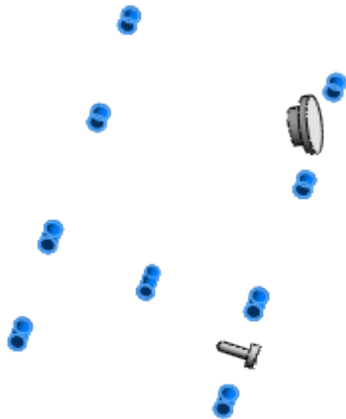
   b) For **Type of body**, select **Custom/Removed Material**.
   c) Under **Operation Options**, select **Custom cost per volume** and type `1.50`.

5. Click **Begin Cost Estimation**.
   The estimated cost of removing material to finish the part is displayed.

Modifying Body Costs

Next, you modify settings and update the multibody part cost estimate.

1. In the CostingManager, expand **Custom Bodies**.
2. To change the custom cost of removing the **Combine1[3]** body, right-click
   Combine1[3] and click **Apply Cost Override**.

3. Enter `0.11` for the custom cost.

> You can also change the cost in the CostingManager, by double-clicking the cost.

The software updates the cost estimate.

4. To modify the body costs, in the CostingManager, right-click **Combine1[1]**, and click **Change Body Settings**.

5. In the Costing Task Pane, under **Operation Options**, select **Custom cost per volume** and type `2.50`.

6. Click **Begin Cost Estimation**.

The software updates the cost estimate.

7. Close all parts without saving them.

Volume Features ★

Assigning Machining Operations to Volume Features ★

To assess the cost of a part that you create from a stock body, you can assign machining operations to volume features.

To assign machining operations to open parts that have volume features:

1. Click **Costing** 🪙 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.

2. In the Costing Task Pane, select the template, material, and stock body.

The software estimates the part cost.

3. In the CostingManager, expand the machining operation folder containing the volume feature: **Mill Operations**, **Hole Operations**, or **Turn Operations**.

4. To change the machining operations type, right-click the volume feature, and click a command to convert to another operation type:

Option	Description
Convert to Drill	Converts mill or turn operations to drill operations.
Convert to Turn	Converts mill or drill operations to turn operations.
Convert to Mill	Converts turn or drill operations to mill operations.

For example, if the volume feature appears under **Mill Operations**, right-click the volume feature, and click **Convert to Drill** or **Convert to Turn**.

5. To change the type of tooling operation, right-click the volume feature, and click **Processing** > **Tooling** and select a different type of tooling operation.

For example, for a flat-end milled volume feature, right-click the volume feature, and click **Processing** > **Tooling** > **Ball End Mill**.

Converting Slot Features to Volume Slot Features ⭐

Volume slot features are volume features that are created from standard slot features. You can convert standard slots into volume slots, and you can revert volume slots to standard slots.

By converting standard slot features to volume slots, you can use the custom volume feature options to improve cost estimates.

To convert a slot feature in a part to a volume feature for Costing:

1. Click **Costing** 🪙 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.
2. In the Costing Task Pane, select the template, material, and stock body.
   The software estimates the part cost.
3. In the CostingManager, expand **Mill Operations**.
   The application identifies existing slot features.
4. Right-click a slot feature and click **Convert to Volume Slot**.
   The cost is updated.

   > To revert a volume slot feature back to a slot, in the CostingManager, right-click the volume slot feature and click **Revert to Standard Slot**.

Saving Volume Features as Parts

Volume features represent the removed material in a part.

For some parts, to determine cost of the removed material, you must convert a set of volume features into a multibody part. Before you can treat a set of volume features as a multibody part, you must first save the volume features as parts.

To save a volume feature as a part:

1. Estimate the cost of a machined part:

   a) Click **Costing** 🪙 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.
   b) In the Costing Task Pane, select the template, material, and stock body.
2. In the CostingManager, expand the folders to display the volume feature.
3. Right-click the volume feature, and click **Insert into New Part**.
4. In the dialog box, enter a file name and click **Save**.

Volume Features for Cylindrical Stock Bodies

The software creates volume features for complex machining operations in cylindrical stock bodies. You can use a volume feature cost estimate to determine the cost of drilling, turning, and milling operations on the part.

Costing Volume Features Example

In this example, you learn about using setup operations and volume features in Costing.
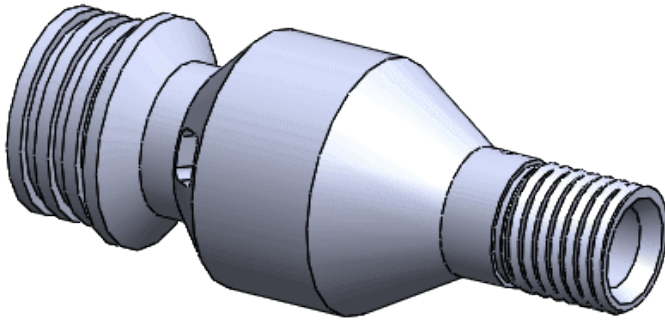
Opening the Part and Setting the Material

First, you open a turned part, and set the material properties.

1. Open *install_dir*\samples\whatsnew\costing\turned_part\turn_example.sldprt, and save the part as my_turn_example.sldprt.
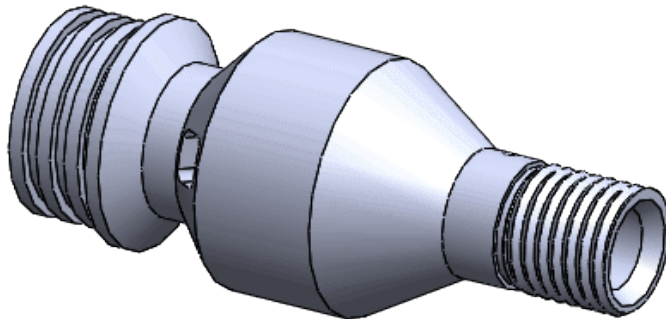
2. Click **Costing** 🪙 (Tools toolbar or the Evaluate tab on the CommandManager) or **Tools** > **Costing**.
3. In the Costing Task Pane, under **Machining Template**, select the default metric machining template, **machiningtemplate_default(metric)**.
4. In the Costing Task Pane, under **Material**, select the following:

   - For **Class**, select **Aluminum Alloys**.
   - For **Name**, select **6061 Alloy**.

5. Under **Stock Body**, for **Type**, select **Cylinder**.
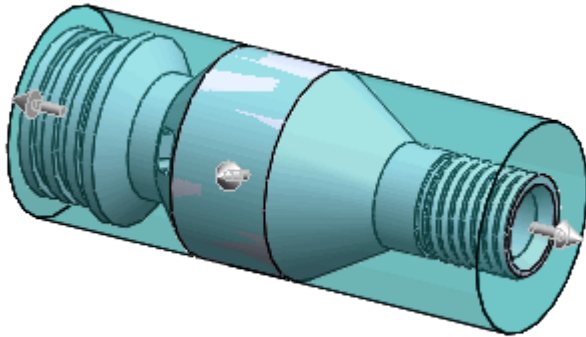
6. To set your material selections as the model material, click **Set Material**.
7. Under **Quantity**, select the following:

   • For **Total number of parts**, enter 80.
   • For **Lot size**, enter 80.

   The software updates the cost estimate in the Task Pane.

Checking Setup Operations and Creating Volume Features

Next, you check the setup operations folder, define the removed material as volume features, and add a setup operation.

1. In the CostingManager, expand the **Setup** folder, and expand the **Operations Setup** folder and its dependent folders.

2. In the Costing Task Pane, click **Costing Options** (upper left corner).

3. In the dialog box, for **Machining Options**, under **Removed material processing**, select **Use customizable volume feature recognition** and click **OK** twice.

   The software recognizes the removed material as volume features.

4. In the CostingManager, right-click **Operations Setup**, and click **Add Setup Operation**.

5. Right-click **Volume 2** and click **New Setup Operation**.

   Volume 2 is moved to a new setup folder.

6. Drag **Volume 4** - **Volume 10** to the new setup folder.

7. Expand the **Load and Unload Setup** folder to display the new operation folders.

Bypassing Templates for Cost Estimates

Next, you select options to obtain a cost estimate that is independent of a selected template, except for the supplied materials information.

1. In the Costing Task Pane, click **Costing Options** 📇 (upper left corner).
2. In the dialog box, for **Machining Options** select the options:
   a) Under **Volume Feature Calculation**, select **Cost per volume removed**.
   b) Type the following values: for **Roughing** type `0.05`, for **Semi-finishing**, type `0.01`, and for **Finishing**, type `0.01`

      In general, select values that are appropriate for the part costs, based on your experience.

   c) Under **Finish processing (Mill/Turn Operations)**, select **Override template settings for offset-based finishing** and type `.01` for **Semi-finishing** and for **Finishing**.

      The finishing and semifinishing values define a displacement distance for finishing and semifinishing operations.

> To apply these settings every time you use the Costing tool, select **Set as Costing default settings**.

3. Click **OK** twice.
4. In the CostingManager, expand the **Mill Operations** folder.
5. Right-click **Volume 3** and click **Add Operation** > **Finishing**.

   The software updates the cost estimate.

6. To revert back to template calculations for Volume 3, right-click and click **Volume 3** and click **Processing** > **Tooling** > **Flat End Mill**.

   The software updates the cost estimate, and evaluates the milling cost of Volume 3, based on the machining template.

Changing Machining Operations

Next, you change machining operations on selected volume features.

1. To calculate the removal of the material in Volume 2 by an HSS drilling operation, in the CostingManager, in the **Mill Operations** folder, right-click **Volume 2** and click **Convert to Drill** > **HSS Drilling**.

   The software updates the cost estimate, and moves **Volume 2** from the **Mill Operations** folder to the **Hole Operations** folder.

2. To calculate the removal of the material in Volume 3 by an OD turning operation, in the **Mill Operations** folder, right-click **Volume 3** and click **Convert to Turn** > **OD Turning**.
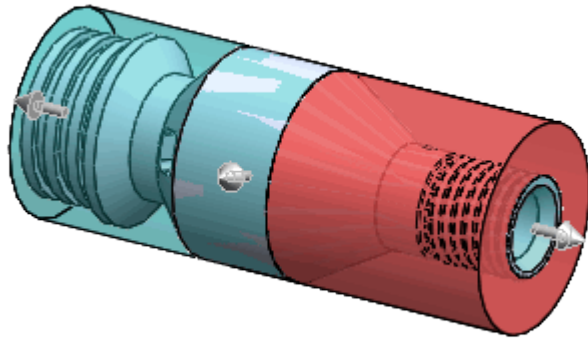
The software updates the cost estimate, and moves **Volume 3** from the **Mill Operations** folder to the **Turn Operations** folder.

3. Close the document without saving changes.

10
Drawings and Detailing

This chapter includes the following topics:

- **Balloons**
- **Bill of Materials (BOM)**
- **Dimensions**
- **Drawing Views**
- **Other Annotations**
- **Sheet Formats**
- **Automatically Splitting Tables Horizontally at a Row**

Balloons

Additional Magnetic Line Soft Snaps

Soft snaps at every 15 degrees are available to facilitate placing or changing the angle of magnetic lines.

Balloon Arrow Attached to Model's Vertex

When you select a balloon, and select and drag the tip of the balloon's arrow that is attached to a vertex of a model, the arrow moves, not the drawing view.

Leaders

You can set a balloon's leader to always attach to the balloon when quantity is specified.

To set a balloon's leader to always attach to the balloon:

In the Balloon PropertyManager, click **More Properties**, and select **Always Attached to Balloon** . To set a balloon's leader to break around quantity, select **Break Around** .

Quantity Placement on Balloons

In the Balloon PropertyManager, you can control the **Distance** between a balloon's border and its quantity.

Reattachable Balloons

You can reattach any balloon (including dangling balloons within a stack) to associate it with any component within the same drawing view.

To reattach a balloon:

1. Right-click the balloon and click **Reattach**.
2. Click a component within the same drawing view to attach the balloon.

Balloons within a stack show information from the new selection. Balloons with leaders automatically point to the new selection.

Bill of Materials (BOM)

Displaying Additional Properties in BOMs

You can display Part Name, Specification, and Standard in BOMs.

See **Additional Properties for Toolbox Components**.

Saving Sort Settings for BOMs

You can save sort settings for BOM tables with the **Save current sort settings** option in the Sort dialog box.

The sort settings are saved with the BOM so that you can reapply the saved sort settings without having to reselect them each time. The saved sort settings are also saved with the BOM template, which lets you propagate the sorting order across all drawings that use the same template.

Saving Sort Settings as a BOM Property

To save sort settings:

1. Right-click a BOM table in an assembly or drawing.
2. Set sort criteria for the table.
   For example, right-click a table column and click **Sort**.
3. Under **Sort scheme**, click **Save current sort settings**.
4. Click **OK**.
   The current sort settings are saved as a BOM property.

Saving Sort Settings to a Template

To save sort settings to a template:

1. Right-click the table and click **Save As**.
2. Navigate to the location where you want to save the template.
3. Type a **File name**, and for **Save as type**, select **Template (*.sldbomtbt)**.
4. Click **Save**.

Using Saved Sort Settings to Create a BOM

To use saved sort settings when you create a BOM:

1. In a drawing or assembly, click **Insert** > **Tables** > **Bill of Materials**.
2. Select a drawing view.
3. For **Table Template**, select the template for which you saved sort settings.

4. Complete the Bill of Materials PropertyManager and click ✔ .
5. Click in the drawing to place the BOM.
6. Right-click in the BOM table and click **Sort** > **Apply Saved Sort Scheme**.

Changing a Saved Sort Scheme

To change the sort scheme when a saved sort scheme exists:

1. Right-click a BOM table.
2. Click **Sort** > **Custom Sort**.
3. In the Sort dialog box, select new settings.
4. Do one of the following:

   - To replace the previously saved settings with your changes, leave **Save current sort settings** selected.
   - To remove the saved sort scheme from the BOM, clear **Save current sort settings**.

Dimensions

Aligning Angular Dimensions with Soft Snaps

You can align angular dimensions with soft snaps that match the offsets specified in **Document Properties**. You can also drag an angular dimension to align it with adjacent dimensions using soft snaps.

To specify the offset distances, click **Tools** > **Options** > **Document Properties** > **Dimensions**.

Angular Running Dimensions ★

Angular running dimensions are a set of dimensions measured from a zero-degree dimension in a drawing or sketch.

Angular running dimensions meet ISO-129-1: 2004 rules while providing additional styling options.
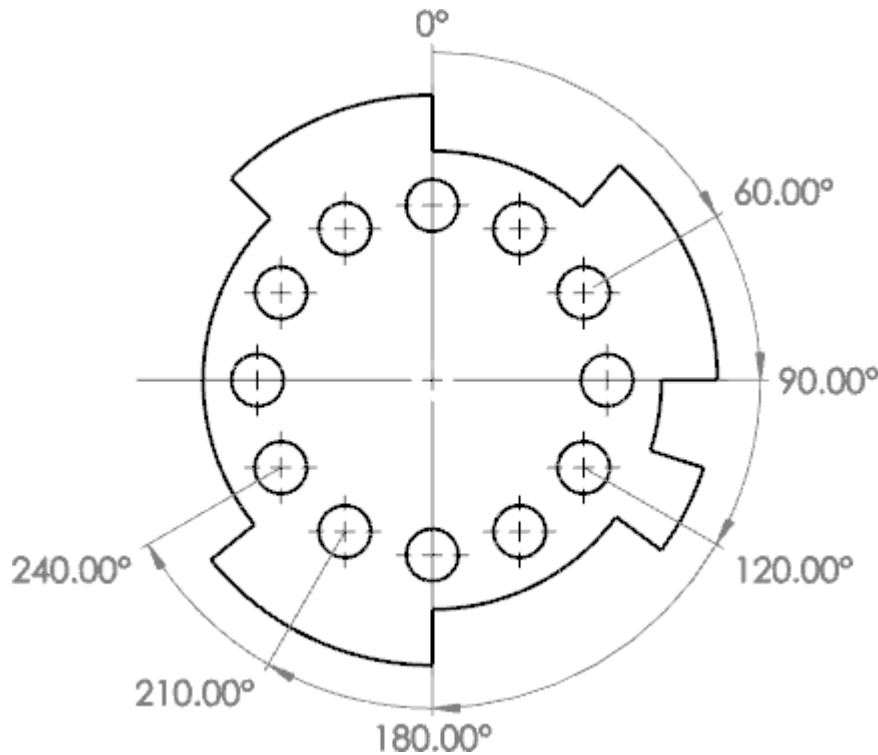
Use angular running dimensions for arc or circle models. In this example, you create and modify angular running dimensions for a flange.

Creating Angular Running Dimensions

You can create angular running dimensions for an arc or circle in a drawing or sketch.

In this example, you create angular running dimensions in a drawing.

To create angular running dimensions:

1. Open *install_dir*\samples\whatsnew\drawings\angular_running_dimension\
    flange.slddrw.

    A flange appears.
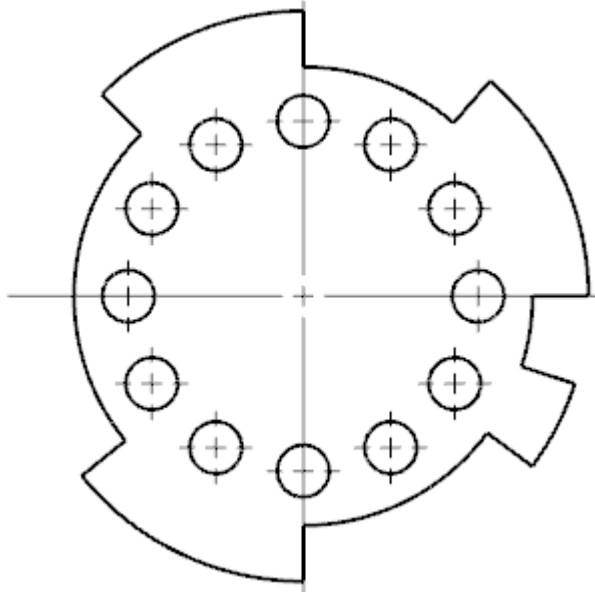
2. Save the flange model as `my_flange.slddrw`.

3. Close `install_dir`\samples\whatsnew\drawings\angular_running_dimension\
   `flange.slddrw`.

4. Click **Angular Running Dimension** (Dimensions/Relations toolbar) or **Tools** >
   Dimensions > **Angular Running Dimension**.

5. In the graphics area, select the center mark of the flange, from which all others will
   be measured, to be the base (the zero-degree dimension), and click again to place
   the dimension outside the model.
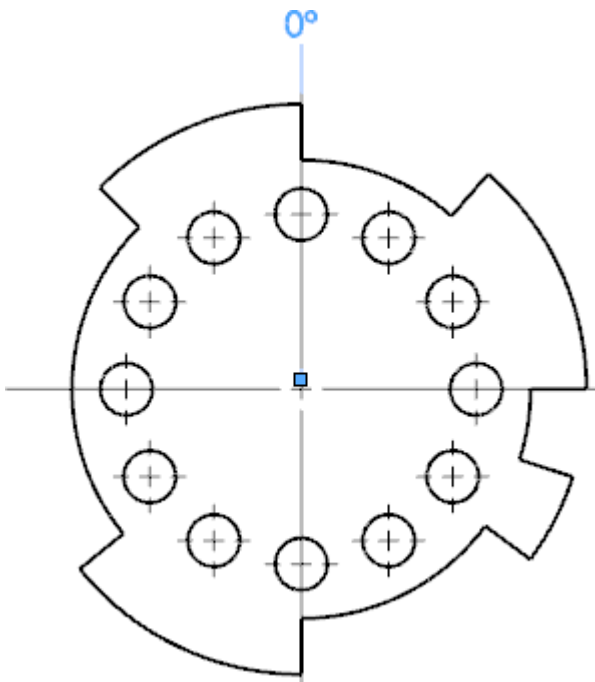
6. Select the vertices or center marks to dimension. As you select each item, the dimension is placed in the view, aligned to the zero-degree dimension.
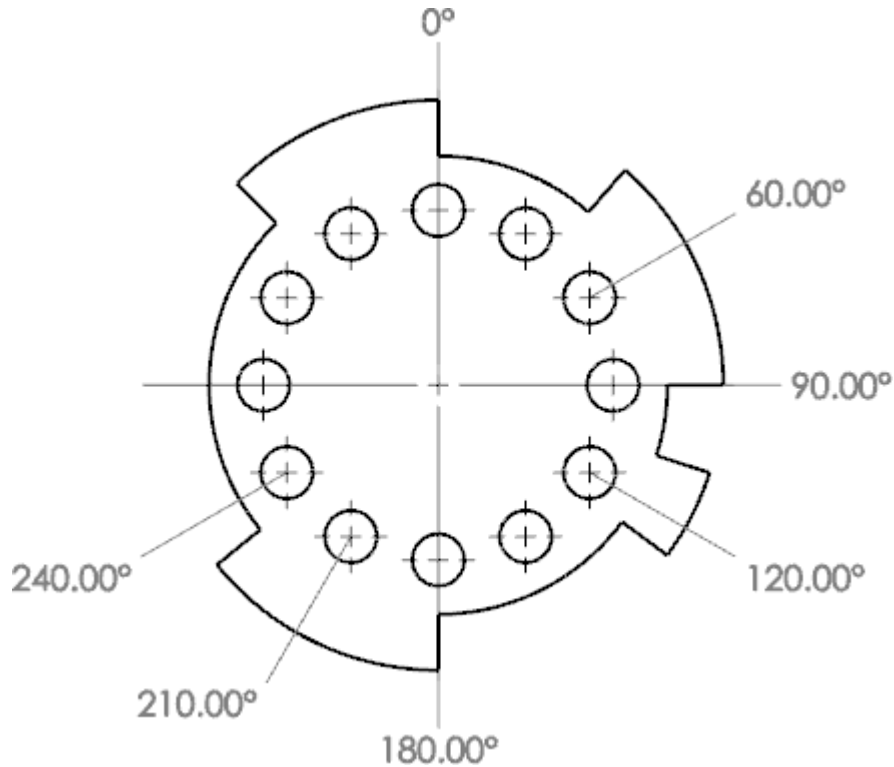
7. Click ✔ to exit the angular running dimension mode.

Adding Angular Dimensions Along the Same Running Dimension

You can add more angular dimensions along the same angular running dimension.

In this example, you add more angular running dimensions to the flange.

To add more dimensions along the same angular running dimension:

1. Open `my_flange.slddrw`, if it is not already open.
2. Right-click an angular running dimension that is on the flange and click **Add To Running Dimension**.

3. Select the vertices or center marks to dimension. As you select each item, the dimension is placed in the view, aligned to the zero-degree dimension.
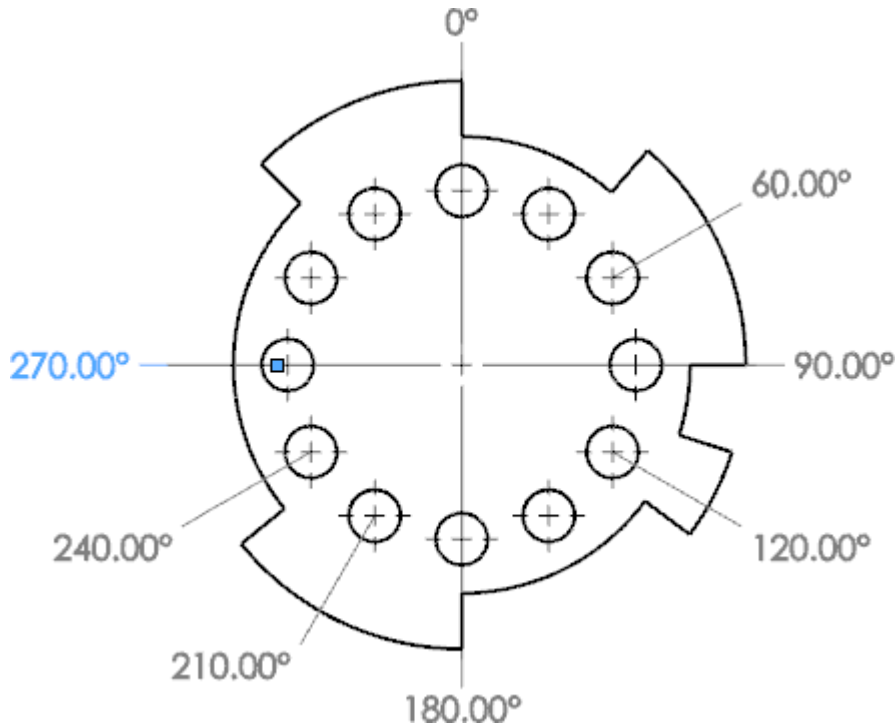
4. Click ✔ to exit the angular running dimension mode.

Modifying Angular Running Dimensions

You can modify existing angular running dimensions.

In this example, you modify the angular running dimensions on the flange.
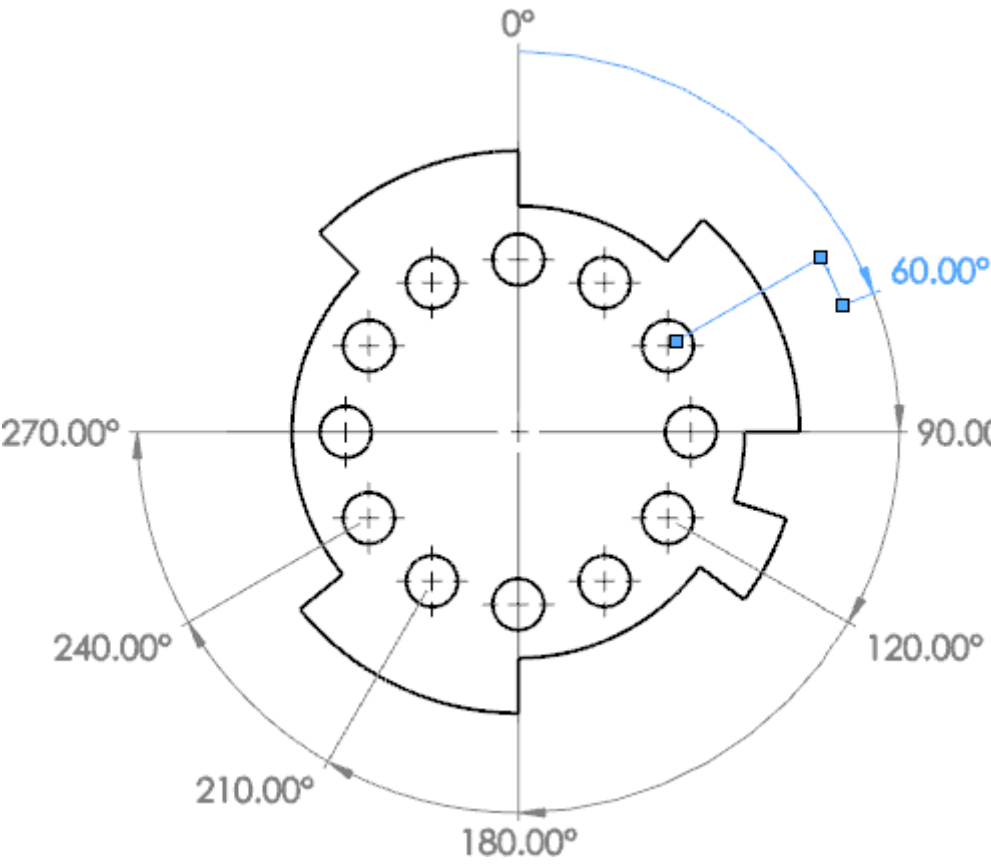
To modify angular running dimensions:

1. Open `my_flange.slddrw`, if it is not already open.
2. In the Dimension PropertyManager on the Leaders tab, make sure **Display as chain dimension** is selected.
3. In the graphics area, right-click an angular running dimension that is on the flange and click **Break Alignment**.
4. Drag the running dimension chain to move it.

   All the running dimensions on the chain move except for the dimension where the alignment is broken.
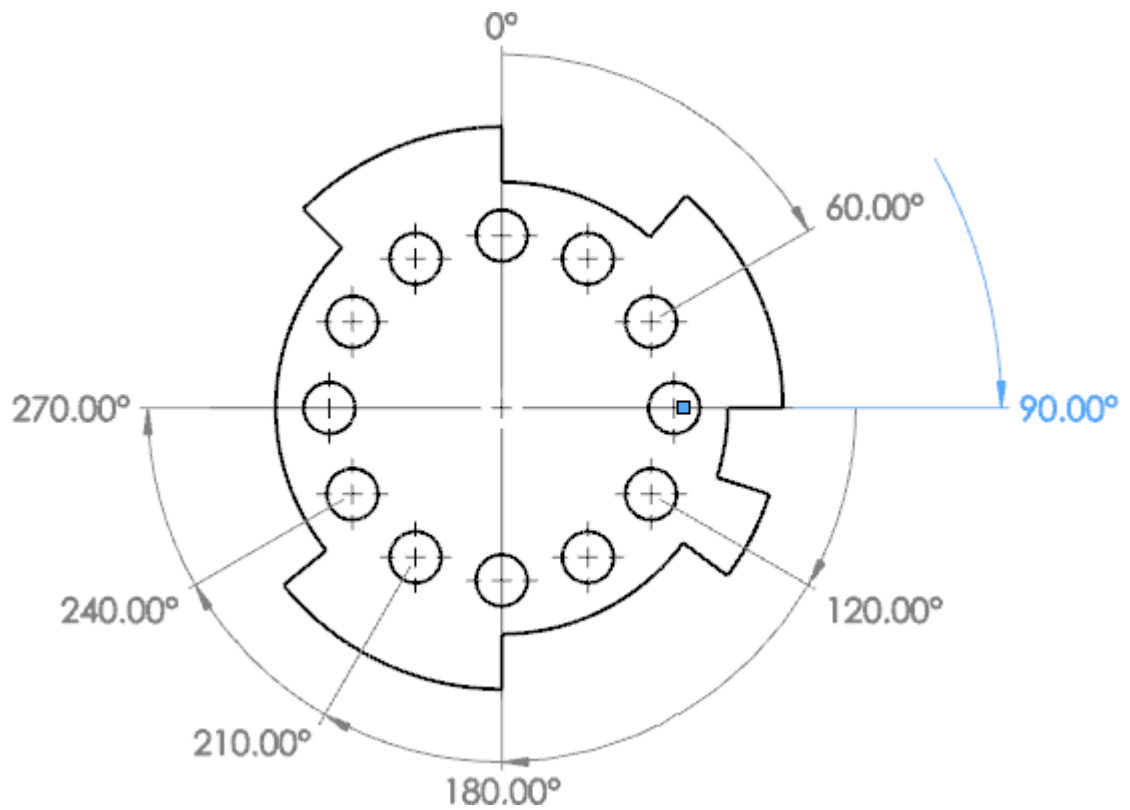
5. Right-click an angular running dimension that is on the flange and click **Display Options**, then choose from these options:

Option	Description
Align Running Dimension	Aligns all the running dimensions.

Option	Description
Jog	Bends the leader line of a dimension and allows you to reposition the dimension.

Option	Description
Re-Jog Running Dimension	Applies the automatic jogging to the running dimensions.

Option	Description
Show Parentheses	Adds parentheses around the selected dimension.
Show as Inspection	Shows the selected dimension as an inspection dimension.
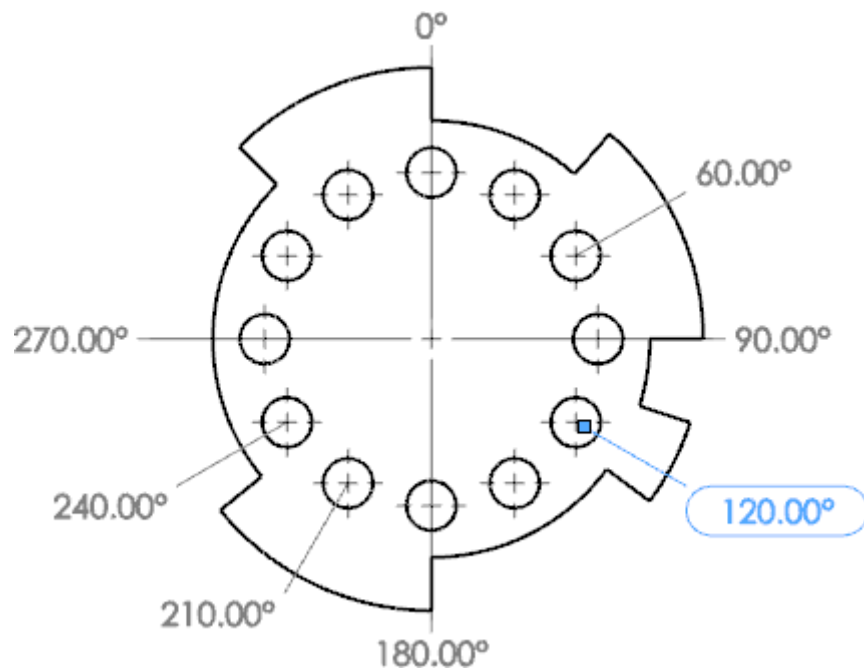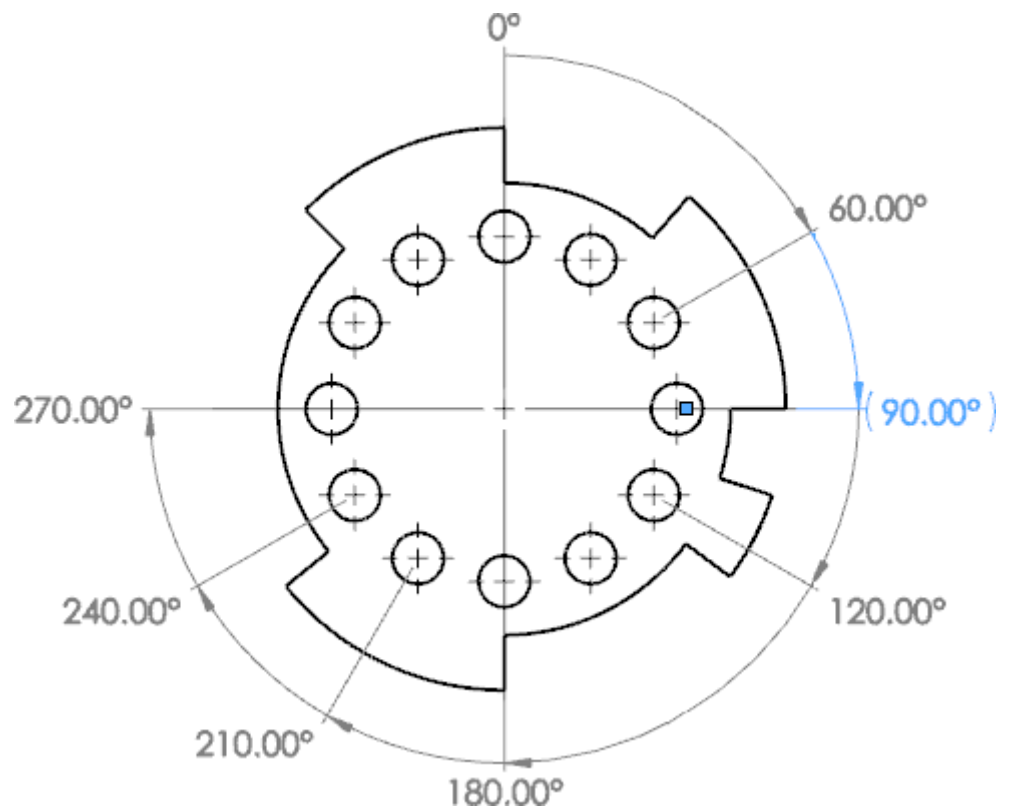
Displaying Angular Running Dimensions

You can change the display style of existing angular running dimensions.

In this example, you change the display style of the angular running dimensions on the flange.

To change the display of angular running dimensions:

1. Open `my_flange.slddrw`, if it is not already open.
2. In the graphics area, right-click an angular running dimension that is on the flange. In the Dimension PropertyManager on the Leaders tab, choose from these options:
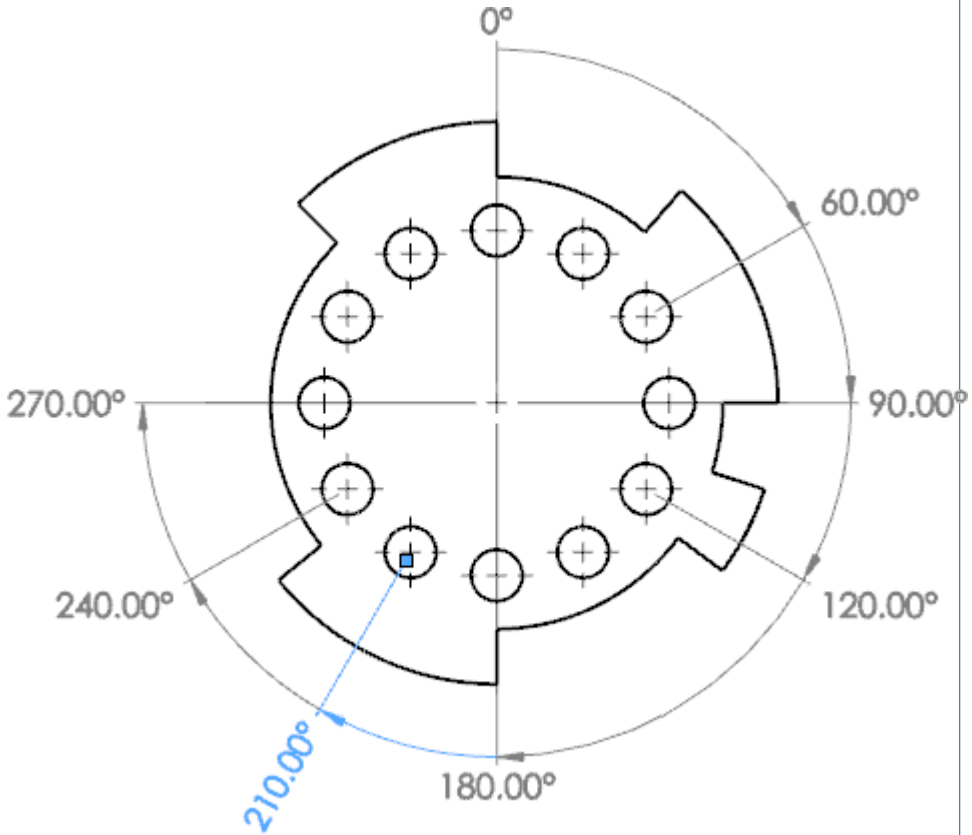
Option	Description
Display as chain dimension	Displays angular running dimensions as chain dimensions.
Run bidirectionally	Causes all angular dimensions to run from the base in the closest direction. No dimension is greater than 180° from the base.
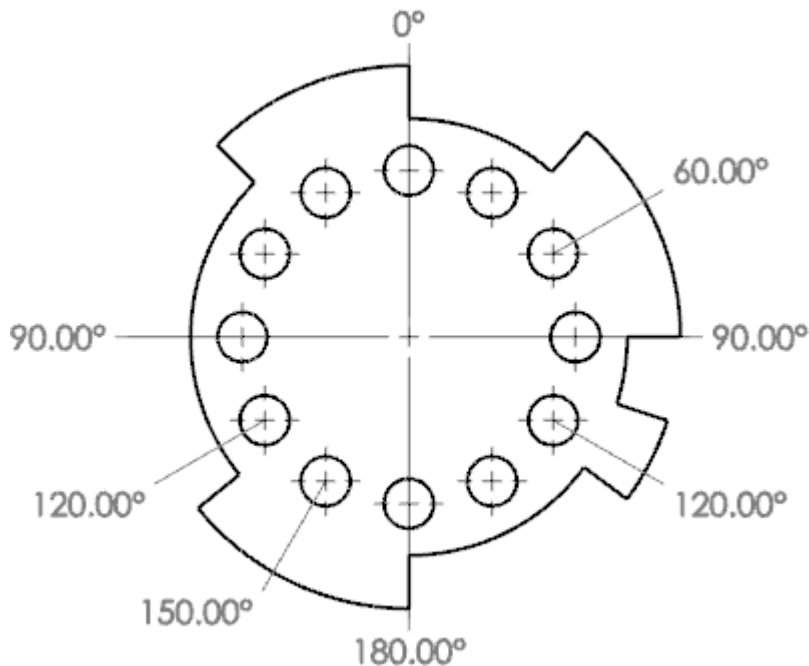
Option	Description
Extension line extends from center of set	Extends the extension line to the center of the dimension set.
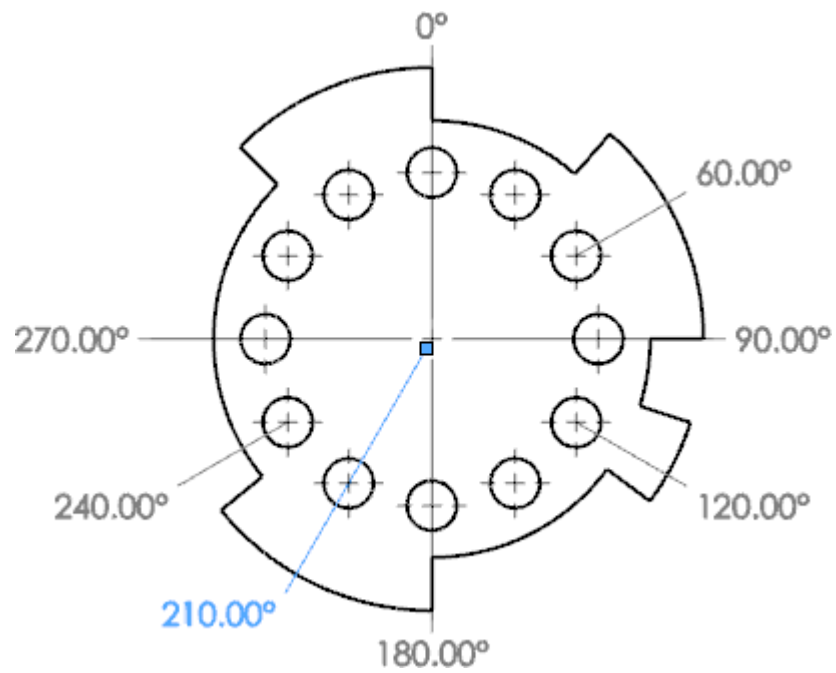
Option	Description
Custom Text Position	Changes text position for the selected dimension. For example, click to place the text inline.

3. Close the file without saving it.

Assigning Styles to Dimension Line Elements

You can assign line styles to dimension leaders, dimension lines, and extension lines using the Dimension PropertyManager.

Automatically Finding Virtual Sharps for Dimensions

While in the dimension tool, you can define virtual sharps between two entities and use the virtual sharps for dimensioning.

To find virtual sharps:

1. In the dimension tool, right-click the sketch entity or model geometry, and click **Find Intersection**.
2. Select the model geometry or sketch entity that intersects your original selection.
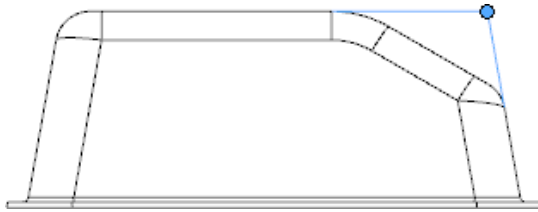
    The virtual sharp is found and applied to the active dimension tool.

3. Complete the dimension using the resulting virtual sharp.
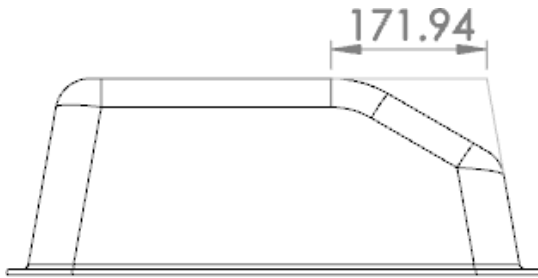
Dimensioning Two Points of the Same Arc

You can add a dimension where both ends are on the same arc, such as in the case of radii and holes where it is necessary to dimension both quadrants or the center and a quadrant.

To dimension two points of the same arc:

1. Click **Smart Dimension** ⬦ (Dimensions/Relations toolbar) or **Tools** > **Dimensions** > **Smart**.
2. **Shift** + select the first quadrant of the arc and select the center or the second quadrant.

> You do not need to hold down the Shift key once you select a quadrant of an arc to start a dimension.
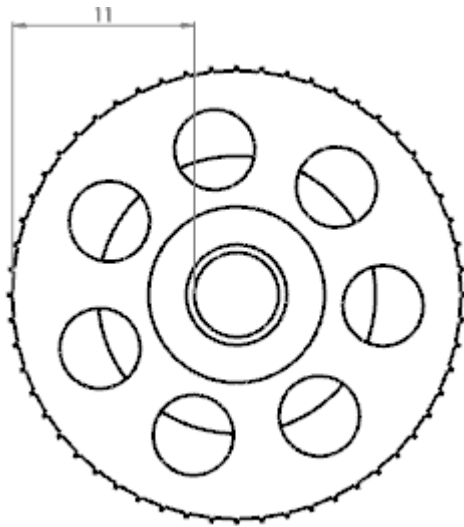
Highlighting Associated Geometry

In modeling and assembly environments, when you select a reference dimension, the associated model geometry highlights.

Placing Dimension Text

For ISO Drafting Standard dimensions with solid leader and aligned text, you can use the Dimension PropertyManager to place text above and below dimension lines, or split dual-dimension text above and below dimension lines.

You use the Dimension Text field in the PropertyManager to place text. You can split text only if the **Text Position** is set to **Solid Leader, Aligned Text** ⬦ in **Tools** > **Options** > **Document Properties** > **Dimensions**.
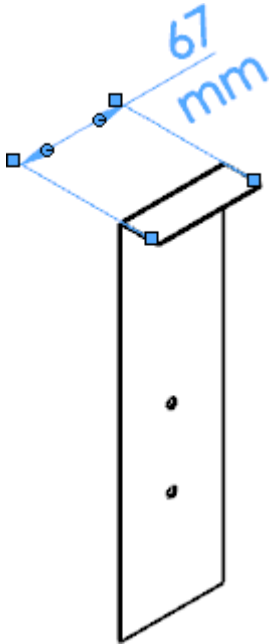
Setting Extension Line to Show Centerline

You can set individual extension lines to display as centerline style. This lets you identify when an extension line extends from a hole.

To set extension lines to display as centerline style:

Right-click the extension line and click **Set Extension Line as Centerline**.

Drawing Views

Attaching Existing Annotations to a Different Drawing View

You can assign existing annotation notes to the current drawing sheet. Within the same drawing sheet, you can assign existing annotation notes to any drawing view.

To assign an annotation note to a drawing sheet or a drawing view:

1. Right-click the annotation note.
2. Do one of the following:

Option	Description
Click Attachment > Attach to Sheet.	Assign an annotation note to a drawing sheet.
Click Attachment > Attach to View, and select the drawing view to which the annotation note shall be attached.	Assign an annotation note to a drawing view.

Creating Shaded Drawing Views With High Quality Edges

You can create shaded drawing views with high quality edges to prevent far side edges from displaying on the near side face of a model. This type of view is appropriate for thin-walled models with features, such as walls or ribs, that make contact with the back face of the model.

Also, when you use the high quality option for shaded drawing views, the edges print with better quality and you can hide the edges.

To create a high quality drawing view with shaded edges:

1. Open *install_dir*\samples\whatsnew\fundamentals\ThinWallPart.SLDPRT.
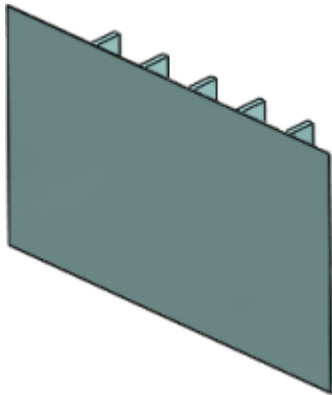
2. Click **File** > **Make Drawing from Part**.
   a) In the Sheet Format/Size dialog box, select **A (ANSI) Landscape** and click **OK**.
   b) Click **Open READ-Only**.
   c) Click **OK**.
   d) From the **View Palette**, drag the Isometric view into the drawing.

3. In the PropertyManager, under **Display Style**, click ⬛ **Shaded With Edges**.
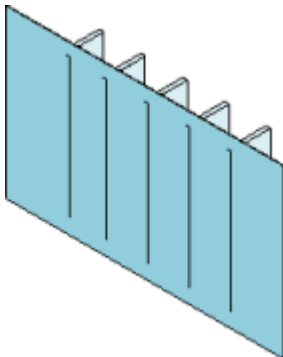
   Notice that the ribs contacting the back face are visible on the front face.

4. Click **High quality**.
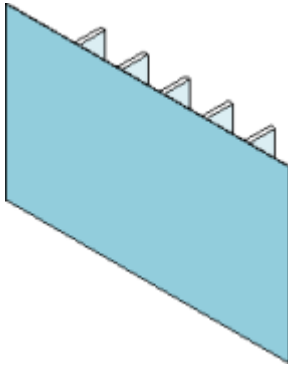
The ribs are no longer visible on the front face.

5. Click ✔ .

Editing Existing Section Views with Section View Assist

You can use Section View Assist, previously called the Section View User Interface, to add offsets to existing section views created with Section View Assist in SolidWorks 2013 and later.

> Section View Assist is not available for section views whose sketches have been directly edited.

To edit existing section views with Section View Assist:

1. Right-click on an existing section view or its cutting line and click **Edit Cutting Line**.

> For some complex section views created with SolidWorks 2013, you must select an insertion point before you can modify them.

2. On the Section view popup, click an offset.

3. Click ✔ .

Enhanced Options for View Labels

Auxiliary Views and Drawing Sheet

You can set auxiliary views and section views to be orthographically aligned to the drawing sheet.

Right-click the drawing view and click **Align Drawing View** > **Horizontal to Sheet Clockwise** or **Horizontal to Sheet Counterclockwise**.

Before being aligned to sheet	After being aligned to sheet clockwise
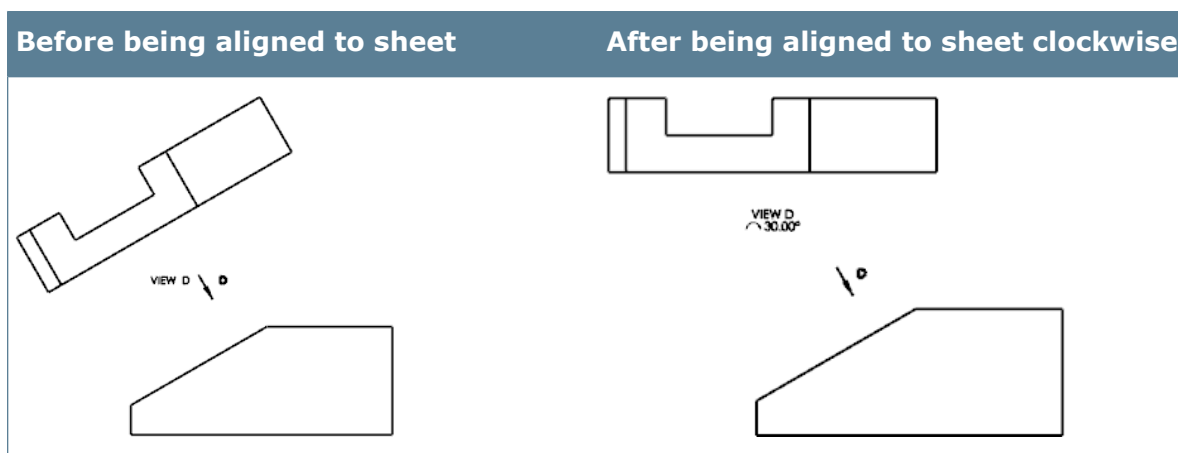
Tags in View Labels

You can use tags to add content to drawing view labels.

To edit a view label using tags, clear **Manual view label** in the Note PropertyManager, right-click the view label and click **Edit Text in Window**.

The tags within the label display their value as assigned from Document Properties. You can delete the tags and use the tag buttons or type in your own text.

View Label Font Control

View label font control apply to all view types.

Labels for detail, auxiliary, other and orthographic views have the same font control as section view does. You can also assign fonts to the individual label elements.

Enhancements to Annotation Views

Annotation View Flat to Screen

An annotation view called **Notes Area** view is always flat to screen.

In compliance with ASME Y14.41-2003 and other Model Based Definition standards, the

Notes Area annotation view is available on all parts and assemblies. All contents on Notes Area will be displayed flat to screen and static in relation to each other. When the model is rotated, all elements on the Annotation View will remain flat to screen at their position relative to each other.

General Tables in Models and Assemblies

You can insert general tables in models and assemblies.

To insert general tables, click **Insert** > **Tables** > **General Table**.

Tables on Annotation Views

You can assign tables to Annotation Views.

Tables are not forced to be flat to screen. Display of tables are controlled by the annotation view to which they are assigned. Note: Tables created in documents prior to this release

are automatically placed on the Notes Area annotation view to continue the previous behavior.

To assign tables to annotation views, right-click the table, click **Change Annotation View** and click a view.

Out-of-Date Drawing Views ⭐

You can add annotations and dimensions to out-of-date drawing views and attach these annotations and dimensions to the model geometry within the out-of-date views. In addition, out-of-date drawing views are no longer obscured.
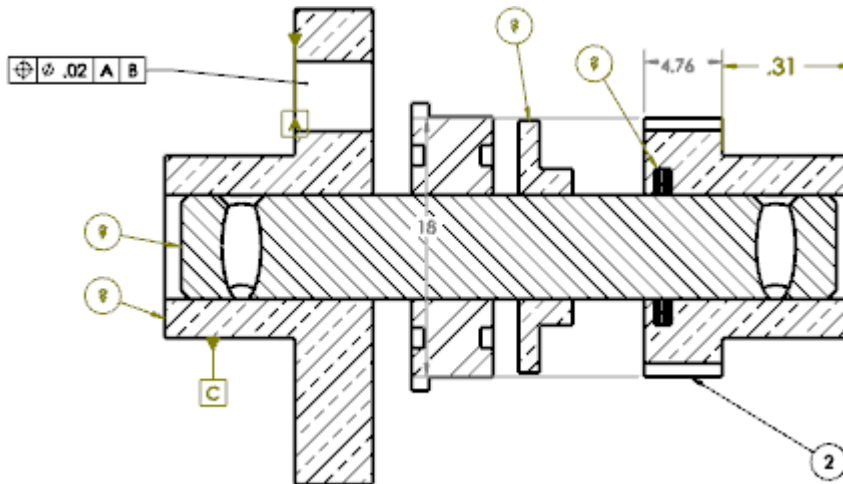
This feature saves you a substantial amount of time when you make small changes to large, out-of-date drawings.

Replace Model for Drawing Views ⭐

Use the Replace Model tool to change file references for individual drawing views while the drawing is open.

You can use the Replace Model tool to change file references between parts and parts, assemblies and assemblies, and between parts and assemblies. If you create drawings that have iterative changes or similar parts that require similar dimensioning schemes, this tool allows you to reuse a drawing in the new context.

In this example, you change file references between simple sheet metal parts and assemblies.

Replacing Part Reference with Assembly Reference

You can change file references from a part to an assembly.

In this example, you replace a sheet metal part with a sheet metal assembly in a drawing view. The sheet metal assembly was created by adding one PEM nut to a hole in the sheet metal part.

| Sheet Metal Part | PEM Nut | Sheet Metal Assembly |
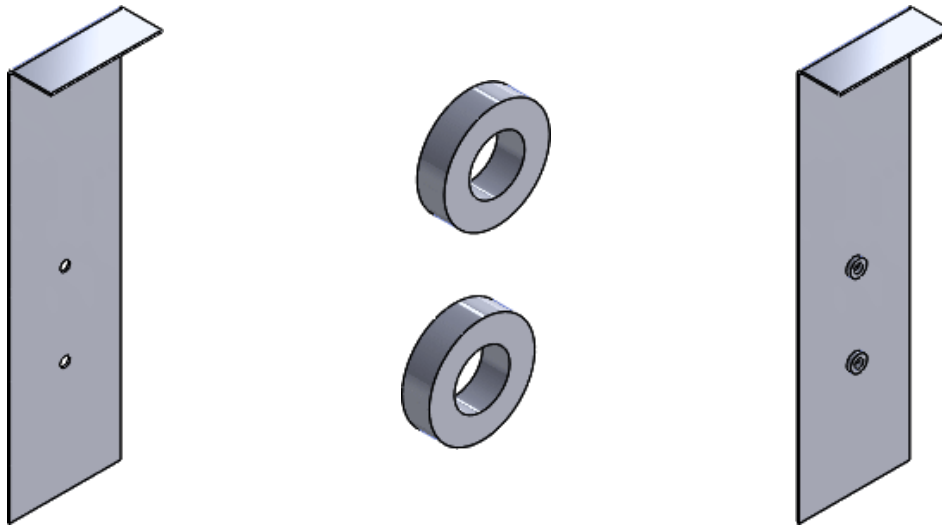
1. Open *install_dir*\samples\whatsnew\drawings\replace_model_in_view\
   sheet_metal.SLDDRW.

   The drawing of a sheet metal part appears.

2. Click **Replace Model** (Drawing toolbar) or **Tools** > **Replace Model**.
3. In the PropertyManager, under **Selected Views**, click **All Views**.
4. Under **New Model**, browse to select *install_dir*\samples\whatsnew\drawings\
   replace_model_in_view\sheet_metal_1_nut.SLDASM and click **Open**.

5. Click ✔ .

   The drawing is updated to show one nut added to part, but is otherwise identical to
   the drawing of the sheet metal part.
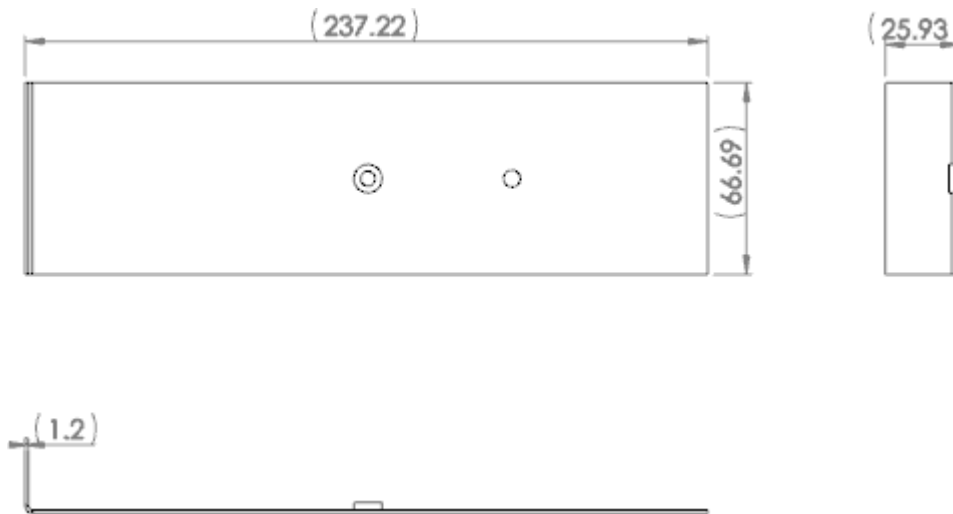
6. Save the drawing file as `sheet_metal_1_nut.SLDDRW`.

Replacing Assembly Reference with Assembly Reference

You can change file references from an assembly to an assembly.

In this example, you replace a sheet metal assembly with a new sheet metal assembly in a drawing view. The new sheet metal assembly was created by adding a second PEM nut to the second hole in the sheet metal part.

Sheet Metal Assembly   PEM Nut   New Sheet Metal Assembly
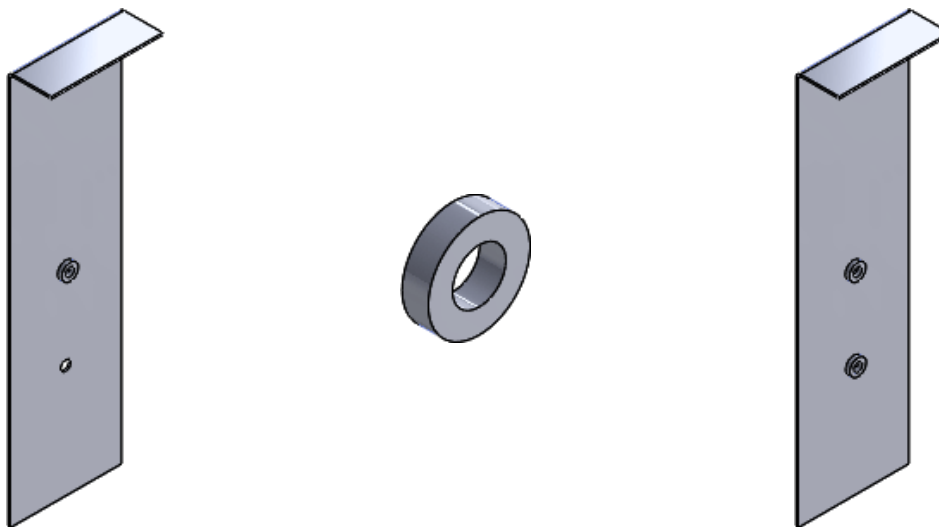
1. Open the drawing file that you saved in the previous module, `sheet_metal_1_nut.SLDDRW`.

   The drawing of the sheet metal assembly with one PEM nut appears.

2. Click **Replace Model** 🗂 (Drawing toolbar) or **Tools** > **Replace Model**.
3. In the PropertyManager, under **Selected Views**, click **All Views**.
4. Under **New Model**, browse to select `install_dir`\samples\whatsnew\drawings\ replace_model_in_view\sheet_metal.SLDASM and click **Open**.

5. Click ✔ .

   The drawing is updated to show a second nut added to assembly, but is otherwise identical to the drawing of the first sheet metal assembly.
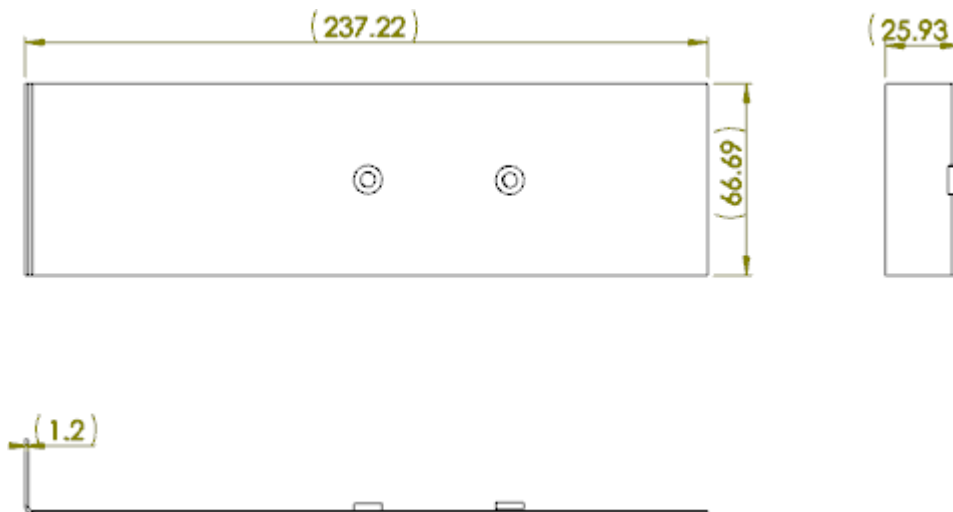
6. Save the drawing file as sheet_metal_2_nut.SLDDRW.

Replacing Assembly Reference with Part Reference

You can change file references from an assembly to a part.

In this example, you replace a sheet metal assembly with a sheet metal part in a drawing view. The sheet metal part was created by removing PEM nuts from the sheet metal assembly.

Sheet Metal Assembly   PEM Nuts   Sheet Metal Part
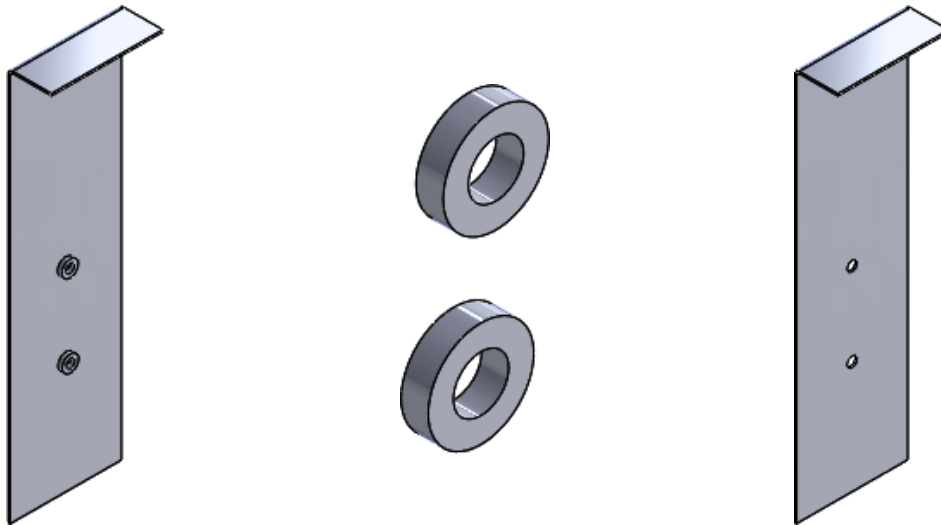
1. Open the drawing file that you saved in the previous module,
   `sheet_metal_2_nut.SLDDRW`.

   The drawing of the sheet metal assembly with two PEM nuts appears.
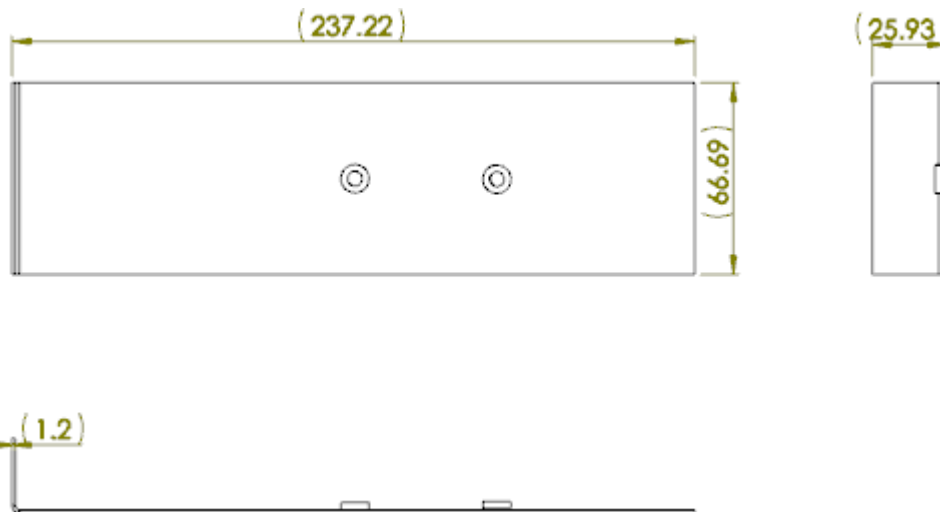
2. Click **Replace Model** (Drawing toolbar) or **Tools** > **Replace Model**.
3. In the PropertyManager, under **Selected Views**, click **All Views**.
4. Under **New Model**, browse to select *install_dir*\samples\whatsnew\drawings\
   replace_model_in_view\sheet_metal_1.SLDPRT and click **Open**.

5. Click ✔ .

   The drawing is updated to show a sheet metal part with PEM nuts removed, but is
   otherwise identical to the drawing of the sheet metal assembly.
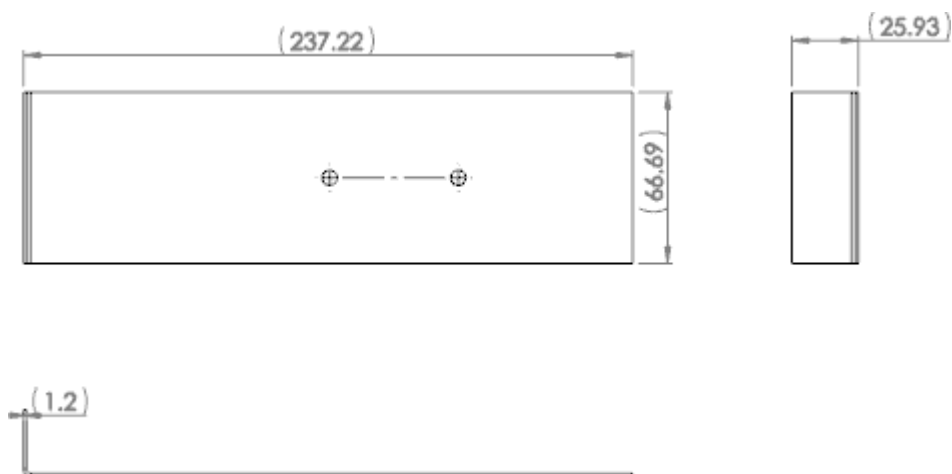
6. Close the drawing without saving it.

Section View of a Surface ⭐

You can create a section view of surface bodies from a parent view of a part. Additionally, you can apply annotations such as notes and dimensions to the cross section of the surface.

To create a section view of surface bodies:

In a drawing view, click **Section View** 🔄 (Drawing toolbar) or **Insert** > **Drawing View** > **Section**. In the PropertyManager, select **Display surface bodies**. This setting persists across sessions.
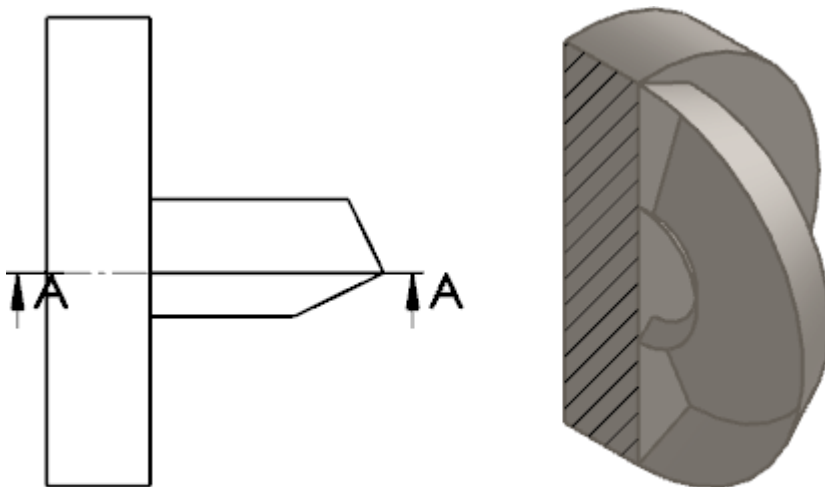
SECTION A-A

You can create a section view of surface bodies for legacy drawings if the drawings' parent views have surface bodies.

View Palette Shows All Views

View Palette ![icon] shows drawing views placed on drawings.

After placing a drawing view on a drawing, the view persists in the View Palette and is identified with a drawing icon. Additionally, the View Palette contains dimetric and trimetric views.

Other Annotations

Adding Center Marks to Hole Wizard Slots

You can use the Center Mark tool to add center marks for slots created with the Hole Wizard.

You control how center marks are added for Hole Wizard slots just like you do for slots created from sketches.

1. Click **Options** ![icon] (Standard toolbar) or **Tools** > **Options**.
2. On the Document Properties tab, under **Drafting Standard**, click **Centerlines/Center Marks**.
3. Specify the drawing document properties for **Center marks**, **Slot center marks**, and the **Center mark layer**.
4. Click **Detailing**, then, under **Auto insert on view creation**, select **Center marks - slots -part** and **Center marks -slots -assembly**.
5. Click **Tables** > **Hole**.
6. Set the **Scheme** and **Tag angle/offset from profile center** to use, and click **Show hole centers**.
7. Click **OK.**
8. Click **Center Mark** ![icon] (Annotations toolbar) or **Insert** > **Annotations** > **Center Mark**.
9. In the PropertyManager, select the edge of the slot for which you are inserting the center mark.
10. Under **Auto Insert**, select **For All Slots** and select the **Options** and **Slot center marks** to use.

    The location of the center mark is determined by the values used to define the slot in the Hole Wizard.

11. Click ![icon] .
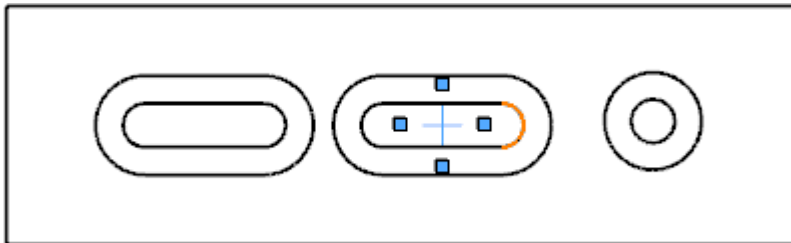
Attach Datum Targets to Specific Points

You can dimension and inference datum targets to specific points.

You can use vertex inferences in both drawing and sketch environments to place datum targets.
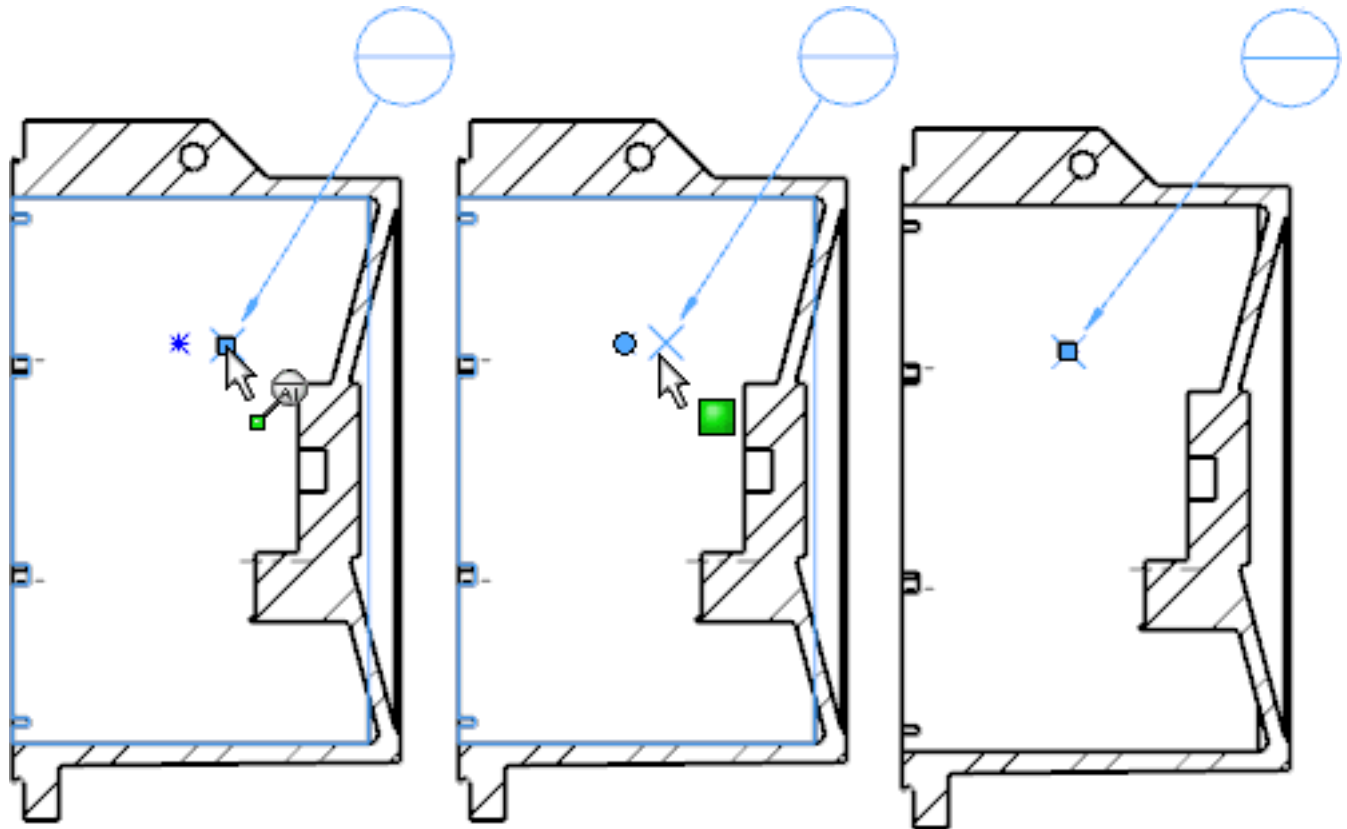
While sketching in parts and assemblies, you can use reference dimensions and model dimensions to control the location of the datum target points. These dimensions are driven in the model environment and driving in the sketch environment.
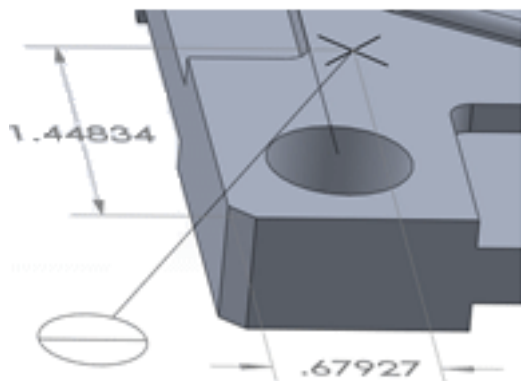
When you select a datum target in the graphics area, the geometry it is attached to is highlighted.

Previously, this functionality was only supported for drawings.

Displaying Annotation Notes in Uppercase

The **All uppercase** option displays note text and custom property values in uppercase letters in drawings, regardless of the case in the note edit field.

You can set the option for individual annotation notes and balloons in the **Text Format** section of the Note PropertyManager.

You can also change the default behavior for the current document by displaying the Options dialog box and selecting **All uppercase** on the Drafting Standards page of the Document Properties tab.

Controlling Case for Individual Notes and Balloons

Use the Note PropertyManager to control how the case is displayed for individual notes and balloons in drawings, models, and assemblies.

To set the text of a note to display in uppercase:	1. Click **Insert** > **Annotation** > **Note**. 2. Expand **Text Format** and click **All uppercase**. All alphabetic text in the document is shown in uppercase. The text appears in uppercase but the actual text value is not converted. If you edit the text value in the Edit in Window dialog box or the Custom page of the Properties dialog box, the text appears as you originally entered it.
To set a balloon containing text to display in uppercase:	1. Double-click the balloon text. 2. In the Note PropertyManager, expand **Text Format** and click **All uppercase**.
To toggle the All uppercase setting on or off:	Select a note or balloon and click **Shift + F3**.

Setting the Default Case for a Document

Use the Document Properties page of the Options dialog box to set the default case to use in notes and balloons in the current document.

1. Click **Options** 🖳 (Standard toolbar) or **Tools** > **Options**.
2. On the Drafting Standard page of the Document Properties tab, under **Uppercase**, click **All uppercase**.
3. To specify strings that should not be automatically capitalized, enter them in the **Exclusion list** field, separated by semi-colons (;).
4. Click **OK**.

GD&T Feature Control Frame Compliance

The Geometric Tolerance (GD&T) dialog box supports changes specified in the ASME Y14.5-2009 standards.

Using the Geometric Tolerance dialog box:

- You can combine multiple adjacent rows with the same geometric symbols into a single composite frame.
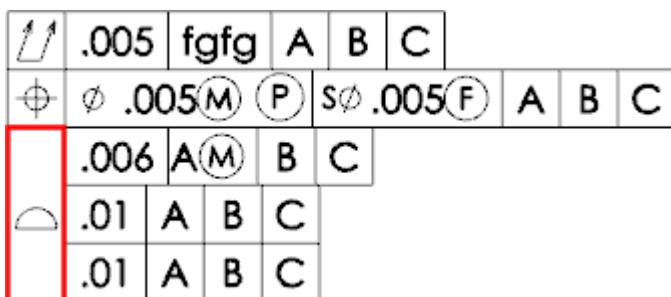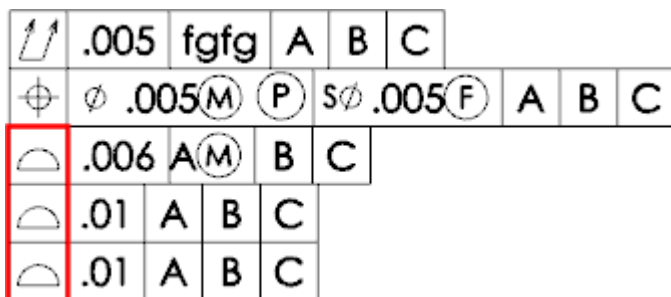
- You can insert the symbol for **Translation** at the end of a selected field.

If you add other symbols, the **Translation** symbol appears after them in the field.

JIS Weld Symbols

Six additional JIS weld symbols are available in `GTOL.SYM` and appear in on the JIS Weld Symbols page of the Symbol Library dialog box.

Edge

Keyhole

Scarf

Stud

Penetration

Backing

When you use the JIS drafting standard for weld symbols, the second fillet option is available for the following symbols:

- **Single Bevel Butt**
- **Single J Butt**
- **Single Bevel Butt with Root**
- **Square Butt**

The effect is the same as it is for the ISO versions of the **Bevel Butt Single** symbol.

Slot Callouts

You can use the Hole Callout tool to create callouts for Hole Wizard slots.

The callout format definition file, *install_dir*\SolidWorks Corp\SolidWorks\lang\ *language*\calloutformat.txt, has been modified to contain new callouts for each slot combination. New sections have been added for countersunk, plain, and counterbore slots.

The slot callouts are at the end of each standard so you can more easily find them if you want to modify them, or combine them with an already modified calloutformat.txt file.

You can re-attach the callout leader to the slot at any point along the arc sections of the slot. The leader always points to the center of the arc. You can control the angle using the **Radial/Diameter leader snap angle**. Click **Tools** > **Options** > **Document Properties** > **Dimension** to access this option.

The callout displays all slot-related dimensions that are marked for drawing, plus the THRU-ALL note or depth symbol value if the slot is through or blind.
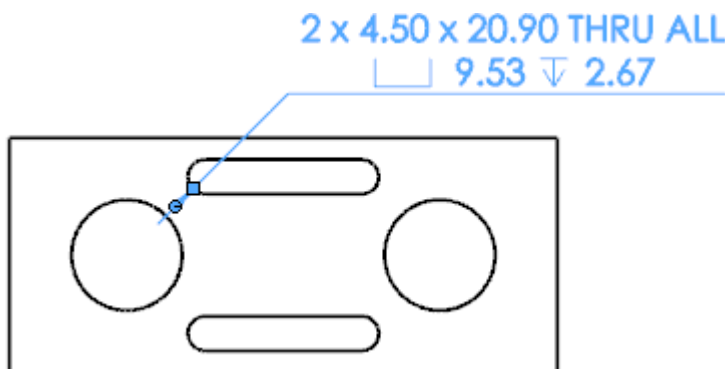
> The slot length is determined by the tangent to tangent length of the main part of the slot, not the counterbore or countersink tangents.

Symbol Library Dialog Box

The Symbol Library dialog box has been overhauled for ease of use. Symbols in the dialog box are categorized by type. You can multiselect symbols and, using the Windows Character Map, you can select additional symbols to insert.

The Symbol Library is available to add symbols to:

- Dimension text
- Weld symbols
- Geometric tolerances
- Datum Features
- Annotation notes
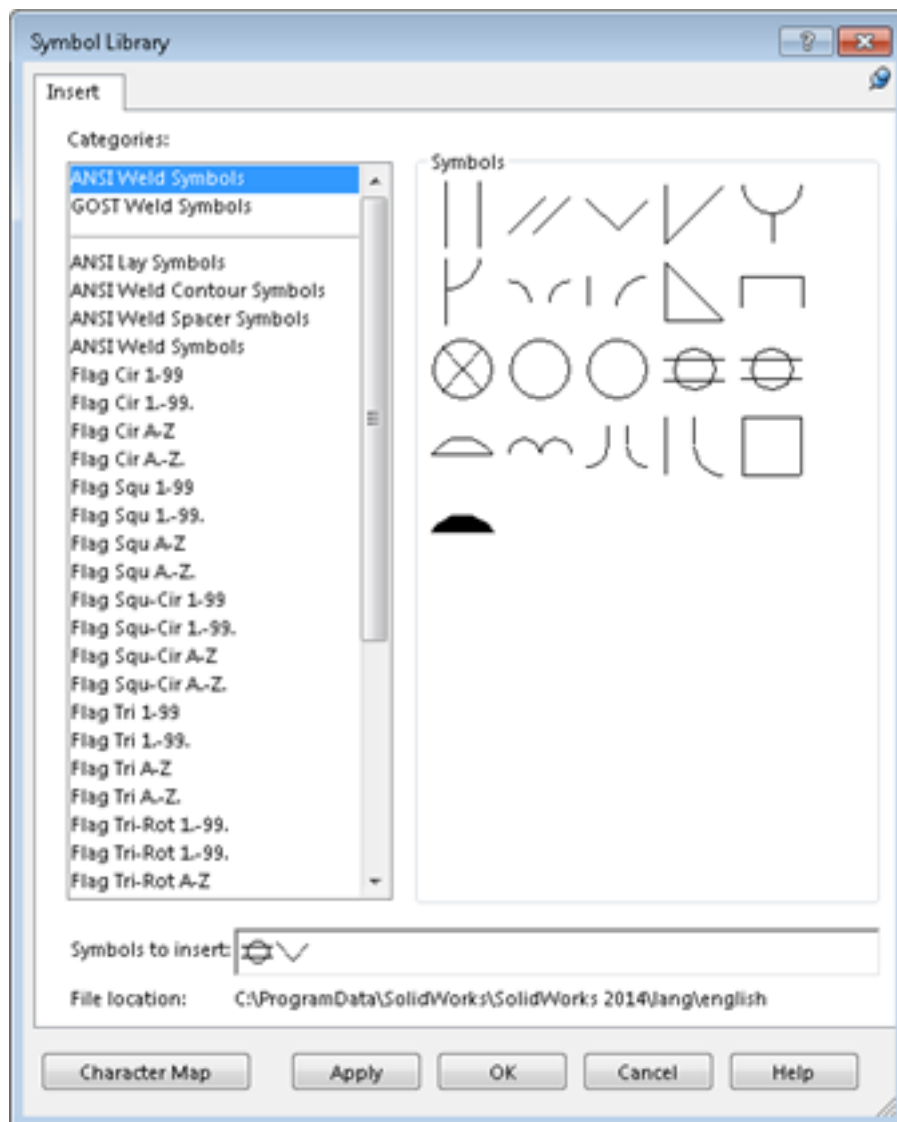
To use the Symbol Library **to insert symbols into dimension text:**

1. Click in a drawing dimension to open the Dimension PropertyManager.
2. On the Value tab, at the bottom of the **Dimension Text** section, click **More**.

   The Symbols Library dialog box opens. The top section of the **Categories** list on the left identifies the categories that contain the symbols that are displayed on the right.

3. Click ⊞ at the top of the dialog box to pin it open.

   > You cannot pin the Symbol Library dialog box when you launch it with an annotation note preview active or from the Weld Symbol dialog box.

4. To add additional symbols to the **Symbols** display, click another category in the list. Tooltips provide the symbols' names and tags.
5. Click a symbol to add to the drawing dimension.

   It appears in the **Symbols to insert** field.

   > You can also add a symbol directly to the drawing dimension by double-clicking it in the **Symbols** display.

6. Do one of the following:

   - To add the symbol to the dimension immediately, click **Apply**.

     The dialog box remains open so you can select additional symbols.

     > **Apply** is only available when the Symbols Library dialog box is pinned.

   - Click additional symbols to add them to the **Symbols to insert** field.

7. To add symbols from the Microsoft Character Map:
   a) Click **Character Map**.
   b) In the Character Map, click a symbol and click **Select** to add it to the **Characters to copy** field.
   c) When you have added the characters you want, click **Copy**.
   d) Close the Character Map.
   e) Paste the characters into the **Symbols to insert** field.
8. Click **OK**.

Symbol Library File Location

You can save the symbol library file, GTOL.SYM, in any searchable folder.

If you are administering a network with many users in distributed locations, you can place a single GTOL.SYM file in a location that all users can access.

Users then specify the location using the File Locations tab of the System Options dialog box.

To specify a new location for the symbol library file:

1. Click **Options** ▦ (Standard toolbar) or **Tools** > **Options**.
2. On the System Options tab, click **File Locations**.

3. Under **Show folder for**, select **Symbol Library File**.
4. Select the current location of the symbol library file and click **Delete**.
5. To specify the new location, click **Add**.
6. In the Browse to dialog box, navigate to the location of GTOL.SYM and click **OK**.

If the GTOL.SYM file in the new location has different symbol definitions, they are applied to your drawings.
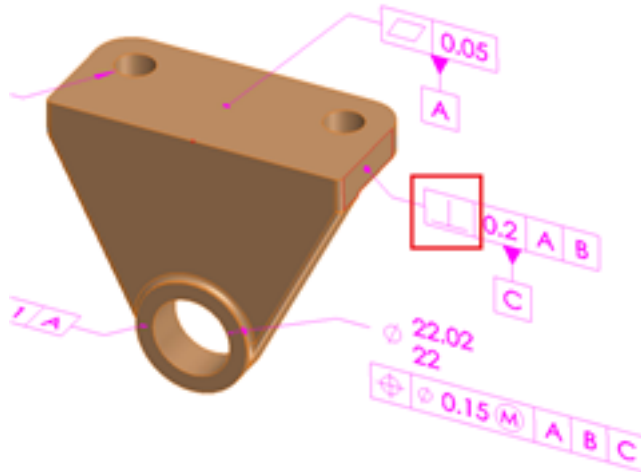
Symbol in original Symbol Library file location
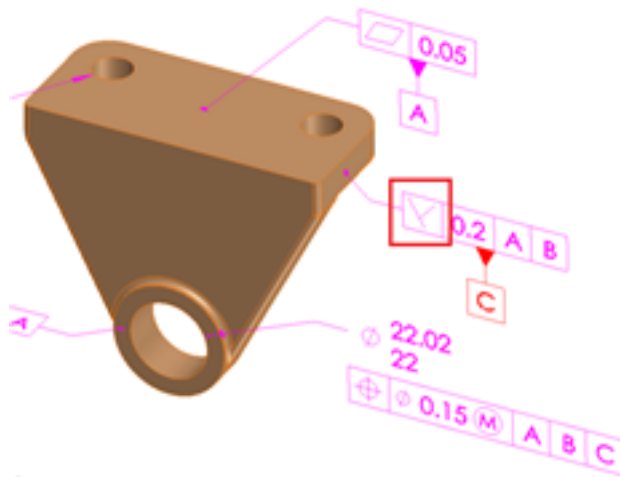
Symbol in updated Symbol Library file location

Glass Sheet Crosshatch

Glass sheet crosshatch is available and may be applied to new section views.

Straight Leader Attachment

An annotation note that is surrounded by a triangle border allows straight leaders and multi-jog leaders to attach to the corners of the border.

Sheet Formats ★

You can use a second sheet format in drawing templates.

The **Drawing Sheets** document property lets you specify a default sheet format for when you add new sheets to drawing documents. This property lets you automatically have one sheet format for the first sheet and a separate sheet format for all additional sheets.

To specify a different sheet format for a new sheet, click **Tools** > **Options** > **Document Properties** > **Drawing Sheets**, select **Use different sheet format**, and browse to select a sheet format file (file ending in .slddrt).

Automatically Splitting Tables Horizontally at a Row

You can use the **Horizontal Auto Split** command to make the SolidWorks software automatically adjust table rows when you add or remove items from tables.

Supported tables:

- Hole Tables
- Bill of Materials
- General Tables

The automatic splitting of tables applies to the entire table, including portions that appear on other drawing sheets.

You can specify that the new table segments be placed directly below the last table split or directly to the right of the last table split:

Before split:

ITEM NO	Number	DESCRIPTION	QTY.
1	SA-19-927334	BASE Sub Assembly	1
2	SA-97-249-45	Slider Assembly	1
3	PRT-VN-237843	Long Hex Bolt	2
4	PRT-VN-09-19292	Bolt	2
5	ASM21-292332	Sub Holder Assembly	1
6	fromparent+ASM-21-292332	fromparent+Sub Hoolder Assembly	1

After split, placed below:

ITEM NO	Number	DESCRIPTION	QTY.
1	SA-19-927334	BASE Sub Assembly	1
2	SA-97-249-45	Slider Assembly	1
3	PRT-VN-237843	Long Hex Bolt	2

ITEM NO	Number	DESCRIPTION	QTY.
4	PRT-VN-09-19292	Bolt	2
5	ASM21-292332	Sub Holder Assembly	1
6	fromparent+ASM-21-292332	fromparent+Sub Hoolder Assembly	1

After split, placed to the right:

ITEM NO	Number	DESCRIPTION	QTY.	ITEM NO	Number	DESCRIPTION	QTY.
1	SA-19-927334	BASE Sub Assembly	1	4	PRT-VN-09-19292	Bolt	2
2	SA-97-249-45	Slider Assembly	1	5	ASM21-292332	Sub Holder Assembly	1
3	PRT-VN-237843	Long Hex Bolt	2	6	fromparent+ASM-21-292332	fromparent+Sub Hoolder Assembly	1

To set up automatic splitting of a table:

1. Right-click the table and click **Split** > **Horizontal Auto Split**.
2. In the dialog box, enter a number in the **Maximum number of rows** field.
3. Under **Apply**, select one of the following:

   - To apply the **Maximum number of rows** value to the table once, click **This time only**.
   - To continuously maintain the automatic splitting of the table, click **Continuously**

     As items are added and removed from the table, it is updated to remain in compliance with the **Maximum number of rows** value.

4. Under **Placement of new split tables**, select one of the following:

   - To place the split portions of the table to the right, click **Horizontal alignment**.
   - To place the split portions of the table below, click **Vertical alignment**.

5. Click **Apply**.

When **Continuously** is in effect for a table, to stop automatic splitting, right-click the table and click **Stop Auto Splitting**.

Current splits are kept but are no longer adjusted automatically as you make changes to the table.

11
eDrawings

eDrawings® Professional is available in SolidWorks Professional and SolidWorks Premium.

This chapter includes the following topics:

- **Viewing 3D ContentCentral on iPad**
- **Viewing DimXpert Dimensions from SolidWorks Part Files**
- **Viewing Rotational Exploded Steps**

Viewing 3D ContentCentral on iPad

You can use eDrawings to view 3D ContentCentral® on an iPad. 3D ContentCentral is a free service for locating, configuring, downloading, and requesting 2D and 3D parts and assemblies, 2D blocks, library features, and macros.

Start eDrawings on your iPad, and click **http://www.3dcontentcentral.com/**.

Viewing DimXpert Dimensions from SolidWorks Part Files

In eDrawings, you can view DimXpert dimensions from files that are saved in SolidWorks 2014 or later.

Viewing Rotational Exploded Steps

In the eDrawings software, you can view rotational exploded steps saved in assembly files created with SolidWorks 2014 or later.

You can also view exploded steps that rotate a component with or without linear translation in the eDrawings software.

12
SolidWorks Electrical

SolidWorks Electrical Professional and SolidWorks Electrical Premium are available as separately purchased products that you can use in SolidWorks Standard, SolidWorks Professional, or SolidWorks Premium.

This chapter includes the following topics:

- **Integration with Enterprise PDM**
- **Replace Microsoft Access with SQLite**
- **Report Templates for Electrical Calculations**
- **Support for Mixed Schema**
- **Additional Features**

Integration with Enterprise PDM

SolidWorks Electrical is integrated with SolidWorks Enterprise PDM.

You can select an instance, manage the export and import of a project with Enterprise PDM, and view the Enterprise PDM status in the Project Manager.

Replace Microsoft Access with SQLite

The underlying database management system has been upgraded from Microsoft Access to SQLite.

This change significantly improves file archival methods.

SQLite is fully compatible with 64 bit operating systems and runs on any version of Microsoft Office that is installed on the computer.

Report Templates for Electrical Calculations

Using Reporting in SolidWorks Electrical, report templates are available to show common calculations, like the maximum amp capacity of a wire or cable and the voltage drop across a length of cable or cable from harness.

You can use the report templates for harnesses, and cables.

Support for Mixed Schema

SolidWorks Electrical supports a mixed mode for drawing called mixed schema that allows you to use schematic tools and line diagram tools in a single environment.

Previously, SolidWorks Electrical only maintained separate drawing types for line diagrams and for schematics.

Additional Features

The following features are available in this release:

Feature	Description
Sorting and renumbering terminals in the Terminal Strips Editor	You can use the Terminal Strips Editor to sort and renumber selected terminals according to positioning, marking or connection criteria.
Manual page break in terminal strips	In the Terminal Strips Manager, you can divide the terminal strip by manually adding a page break.
Location and function outline	When placing a location or function outline, the objects inside the outline can be automatically associated with the location or function.
Translation of text for projects	A new command allows you to search and replace text in the interface used to translate text for projects.
New data for material terminals of components	You can manage the maximum number of connections on a material terminal of a component, and the use of the terminal.
Editing component material terminals	You can edit component material terminals from the symbol context menu.
New object used to hide the drawing	You can use a new entity type called Wipeout to hide entities that are located behind other entities.
New interface for Reports Manager	The Reports Manager interface has been significantly improved.
Reports, TXT export	The TXT export of project data supports the `.csv` format in addition to the TXT format.
Reports, TXT and Excel export	You can generate and add data files to the project.
Reports templates selector	You can use filters in the reports templates sector to search for a template.
User data	An index shows the order number of user data. The index displays in dialog boxes for electrical objects properties.

Feature	Description
Passive symbol	A new passive symbol is available. A passive symbol is not considered as an electrical symbol. It does not have a mark, circuits or connection points. It is stored in a library for insertion into a drawing.
Drawing contextual menu	You can use more commands, such as addition, editing and swapping, to manage the title block.
2D footprint thumbnails	You can insert a mask at the same time as a 2D thumbnail to hide a rail.
Unarchiving	During unarchiving, you can use a command in a contextual menu to specify which objects to keep and which to replace.
Component search	A search function has been added to a side panel of the components contextual menu.
Cross references configuration	In the contextual menu of the cross references thumbnails, you can access the cross reference drawing configuration.
PDF export	A new option generates PDF files with the tree structure for the project in the form of bookmarks.
Filters in the Project Manager	You can use new criteria in the Project Manager to reduce the list of projects.
Copy and paste special	When using the Paste Special command, a dialog box opens allowing you to manage marks.
Equipotential conflicts	You can resolve equipotential conflicts manually with a new command.
Importing DWG files	When importing DWG files into the project, if the DWG file has definitions of symbols or title blocks, you can launch the wizard to place title blocks and symbols in the library.
Wire properties or equipotential editor	A dialog box displays the properties of the wire, such as marks and technical data, or all the wires of an equipotential.
Wire and equipotential label	You can use a new type of symbol, composed mainly of attributes, to display data specific to the wires and equipotentials.

Feature	Description
User data on the wires	The wires display translatable and non-translatable user data.
Insertion of a location outline	New options are available when inserting a location outline.

13
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:

- **Automated Cache Management**
- **Consolidated Delayed in State Notifications**
- **Dynamic Notification Recipients Filter**
- **Extracting Hidden SolidWorks BOMs**
- **Flexible "Always work with latest version of files" Option**
- **Flexible Retrieval of Referenced Files During Check Out**
- **Increased Workflow Performance**
- **Navigating Tree Structures**
- **User Defined Custom Columns**
- **Version Comparison of Referenced Files**
- **Version Free Variables and Workflow Transitions**
- **Visibility of Child Quick Information Warnings in Parent Assemblies**

Automated Cache Management ★

Two options in the User and Group properties dialog boxes let administrators edit user profiles to set caching behavior. These options automate refreshing and clearing the local cache on designated folders.

For example, if you store SolidWorks Design Library folders in the vault, to ensure that users always have the latest version of these files, you can specify that when users log in, the cache is automatically refreshed for these folders.

The Cache Options page of the user and group properties dialog boxes contains the following options:

Clear cache during logout	Automatically clears the local cache upon logout. This removes unneeded files, reduces the cache size, and increases security on machines used by multiple users.
	When users logs out, Enterprise PDM clears the cache of any folders with this setting using an automated version of the **Clear Local Cache** command.
	The cache is also cleared when users select to work offline or when users shut down their computers without logging out.

| **Refresh cache during login**	Ensures that users have the latest version of items like CAD file templates and standard libraries each time they log in.
	When users log in, for every folder with this setting, Enterprise PDM performs an automated version of the **Get Latest** command. Enterprise PDM updates older versions of files in the user's cache and uploads the latest version of files that are not in the user's cache.

Administrators can set cache options for individual users or for groups. When users are part of a group for which cache options have been set, the group settings are reflected in the users' properties.

Consolidated Delayed in State Notifications

To reduce the number of individual notifications users receive, Enterprise PDM groups all the delayed in state notifications that occur during the specified time interval and sends a single email containing the details of the files that meet the delayed in state criteria.

Administrators or users can configure folder notifications to alert users when files in the folder have been in a workflow state for a specified amount of time. These notifications are referred to as "delayed in state" notifications.

Previously, separate emails were sent for each file that met the delayed in state criteria.
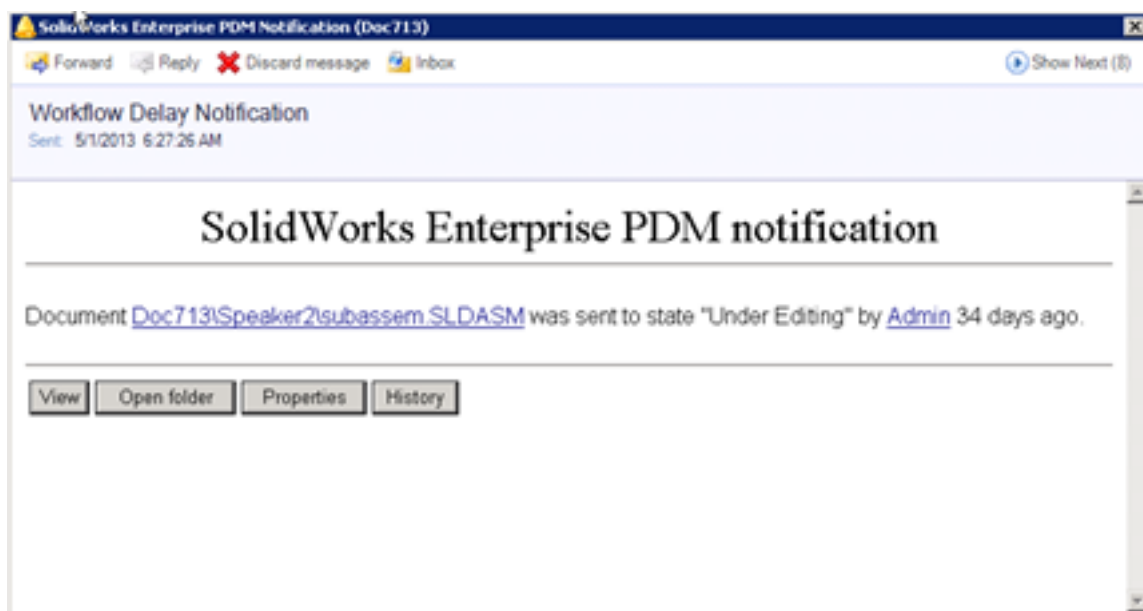
To get details of all the files that were identified as delayed in state, users had to open each notification by clicking **Show Next**, or by opening each notification in the Inbox.
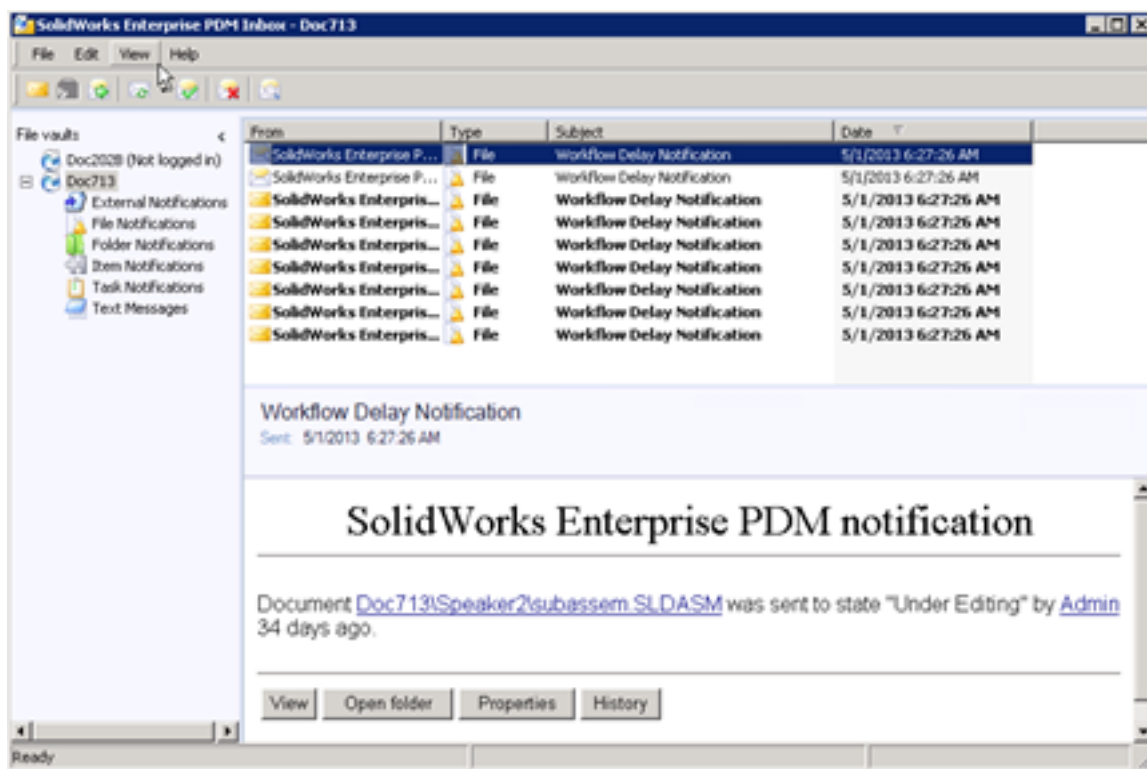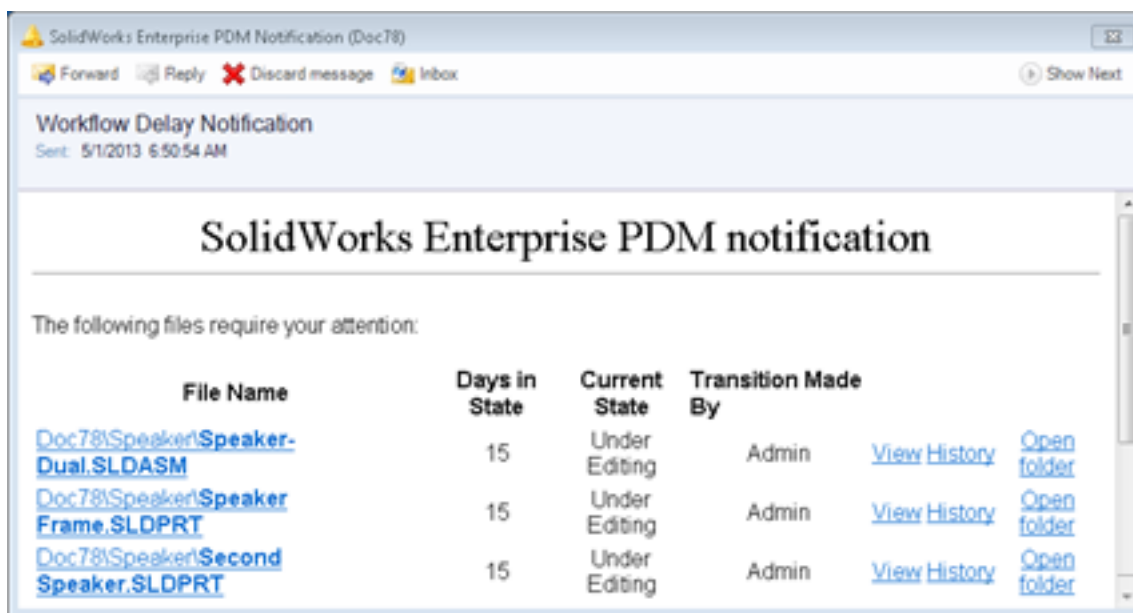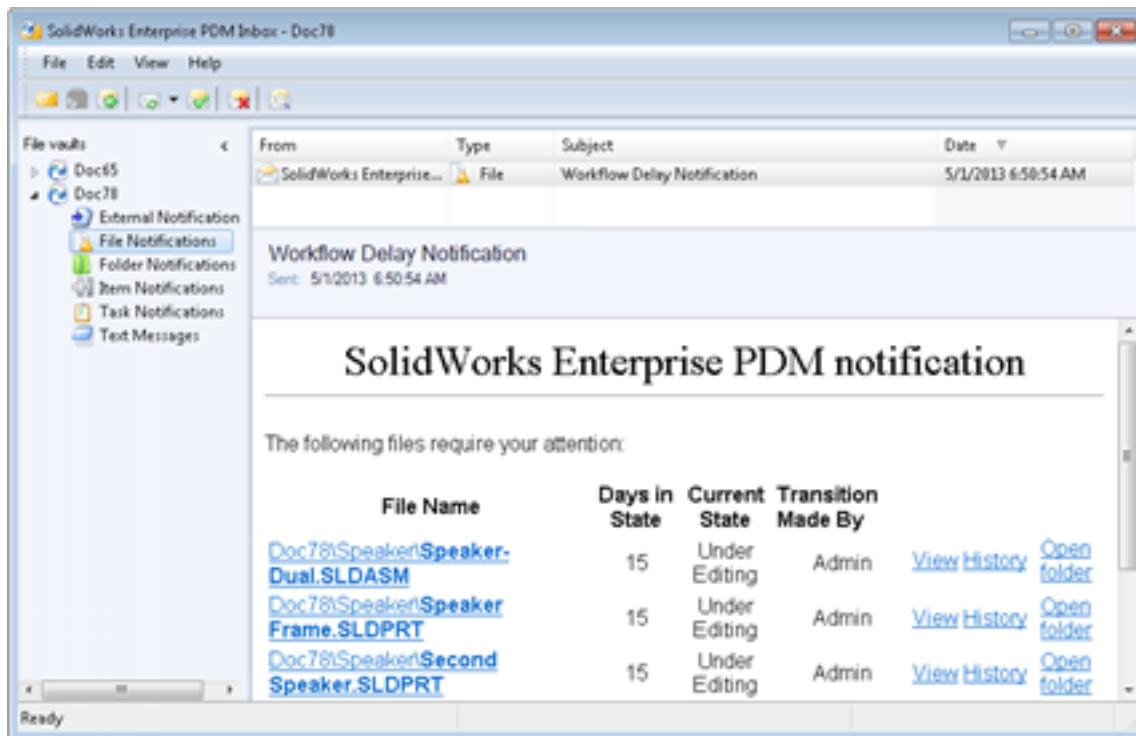

Now a single notification lists all files that are delayed.

The Inbox contains that single message.

The notification contains the following information for each file:

File Name	Click the link to open the file in the appropriate editor.
Days in State	Length of time the file has been in the current state.
Current State	Current workflow state.
Transition Made By	User who performed the most recent state change.
View	Opens SolidWorks files in eDrawings. Other files are opened in the SolidWorks Enterprise PDM File Viewer.
History	Opens the History dialog box for the file associated with the notification.
Open Folder	Opens the folder containing the file in a new Explorer window.

Dynamic Notification Recipients Filter ⭐

A filter for dynamic notification recipients in the Do Transition dialog box facilitates selection for users when a company has many users designated for dynamic notification.

This filter is similar to the one in the Workflow editor for adding Role members and notification recipients.

As users type alpha numeric characters into the **Filter** field, the results in the recipients list update dynamically. When users select names from the filter results, those names remain as they type another filter string.
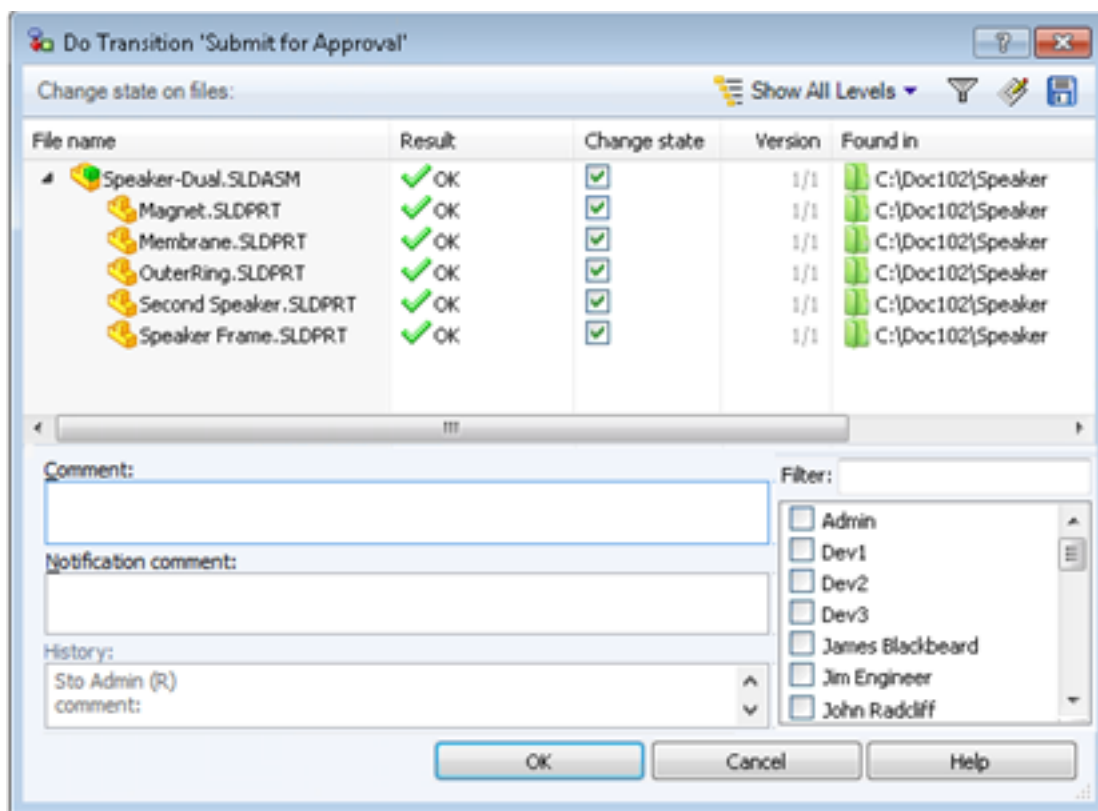
First filter string applied	User selected	Second filter string applied
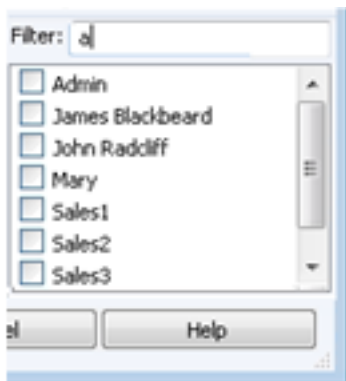
Extracting Hidden SolidWorks BOMs

In the SolidWorks application, you can hide a Bill of Materials on a drawing or assembly to prevent it from distracting you. Meanwhile, you may want to access the SolidWorks BOM in Enterprise PDM to export ballooning and other information to other applications like an MRP/ERP system.
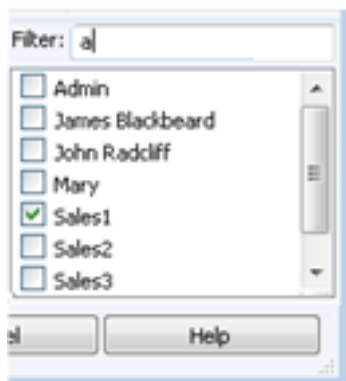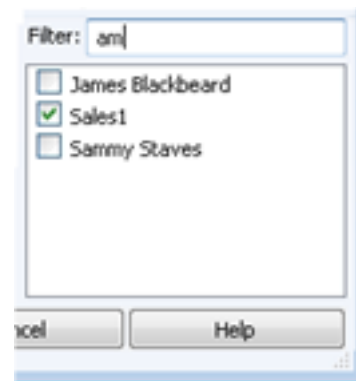
To enable Enterprise PDM to always extract hidden BOMs:

- Click **Display** > **Options** > **Extract hidden BOM from SolidWorks files on Check-In**.

   When you check in SolidWorks files with hidden BOMs, the software extracts the BOMs to Enterprise PDM as SolidWorks BOMs.

Flexible "Always work with latest version of files" Option

Administrators can give users who work in the SolidWorks add-in access to earlier versions of files even when the **Always work with latest version of files** setting is enabled.

The **Enable the get version command in SolidWorks** option lets users access specific versions of files in the SolidWorks add-in by:

- Expanding the **Get** flyout in the SolidWorks Enterprise PDM task pane toolbar
- Right-clicking a file in the task pane and clicking **Get**
- Right-clicking a file in the FeatureManager and clicking **Get**
- Clicking **Enterprise PDM** > **Get**
- Right-clicking a component in the graphics area and clicking **Get**

This option appears on the Reference Dialog tab of the Settings dialog box when you select **Always work with latest version of files**.

Flexible Retrieval of Referenced Files During Check Out

Administrators can configure user settings to specify how the Check Out dialog box handles referenced files.

Users typically want to work with the latest version of files. However, some users need to work with assemblies that use older versions of references that they have cached, rather than the latest version of the files in the vault.

The revised Reference Dialog page in the user and group Settings dialog box lets administrators specify whether or not reference files will be automatically selected for **Get latest** in the Check Out dialog box. It contains this option, which is the default:

Auto select reference files to get latest when checking out.

To maintain the cached version of reference files during check out, you must clear this option.

If you enable this option, you can set these conditions:

Always work with the latest version of files.	Retrieves the latest available version of files to the local vault view when viewing or opening them.
	This option was previously available on the Explorer tab of the Settings dialog box, under **Miscellaneous**.
Enable the Get Version command in SolidWorks.	Only available when you select **Always work with the latest version of files**.
	Enables the **Get Version** command in SolidWorks to retrieve earlier versions of open files.
Check out files silently without showing the check out dialog box.	Prevents the display of the Check Out dialog box, when you check out files with references.
	The software checks out only the selected files; users must separately check out each referenced file.
Try to check out all referenced files when the referencing file is checked out.	When users check out files with references to other files, the referenced files are also marked for check out (if it is possible).
	If you also activate the option to check out files silently, checking out parent files automatically checks out all referenced files.

Increased Workflow Performance

The time required for changing workflow states in high latency environments has been significantly reduced.

The time reduction varies depending on the number of actions you set in the transition and the amount of latency. The greatest improvement occurs in systems with the highest latency.

Navigating Tree Structures ★

The reference tree structures maintained by Enterprise PDM are often large and contain many levels. New controls in dialog boxes and Preview tabs make it easier to navigate these structures.

The following navigational aids have been added in Enterprise PDM 2014.

In the Check In, Check Out, Copy Tree, and Change State dialog boxes:

- A **Top Level Only** / **Show All Levels** control lets you show just the highest level of the tree structure or the entire tree structure.
- Expand ▷ and collapse ◢ arrows within the tree structure let you expand or collapse individual nodes of the tree.
- **Next Warning** ⚠ and **Previous Warning** ⚠ arrows in the dialog box toolbars take you directly to warnings.
- The option to turn on a second set of parent node check boxes at each subassembly level lets you select or clear all the check boxes for that node.

On the Contains and Where Used tabs:

- Expand ▷ and collapse ◢ arrows within the tree structure let you expand or collapse individual nodes of the tree.

User Defined Custom Columns ★

To facilitate work in dialog boxes and Preview tabs, you can add columns based on Enterprise PDM variables. You can also rearrange the dialog box display by dragging the column heads.

For example, if your company organizes documents by part number rather than by file name, you can add a column that shows the Part Number.

You can add custom columns to the Check In, Check Out, Undo Checkout, and Copy Tree dialog boxes and to the Contains and Where Used Preview tabs.

To add a custom column to the Check Out **dialog box:**

1. In Windows Explorer, select the files to check out and click **Actions** > **Check Out**.
2. In the dialog box, right-click anywhere in the table headings.
   A list of standard headings appears. Check marks indicate which headings are currently displayed.
3. At the bottom of the list, click **More**.
4. In the Choose Columns dialog box, scroll to the variable you want to use and select it.

5. Click **OK**.

   A new column that uses the variable as its heading appears on the right side of the dialog box.

6. Click and drag the column to the desired location.

Version Comparison of Referenced Files ⭐

In the Enterprise PDM Add-in for SolidWorks, you can set the Task pane display information to include a **Reference version** column. This column shows the version of a file that is referenced by an assembly compared to the latest version of the same file in the database.

Additionally, the 🔶 quick info icon appears when an assembly references a file version that is not the latest version in the database.

When you are working with an assembly that contains many subassemblies, this information can help you decide whether to check out a subassembly and rebuild it so that it references the latest design version.

Version Free Variables and Workflow Transitions

When you include version free variables in the data card of a file, you can use a workflow transition to update the content of the variables without creating a new version of the file.

Administrators must first configure the variables to be version free and then create a workflow transition that includes actions that set the value of the variables.
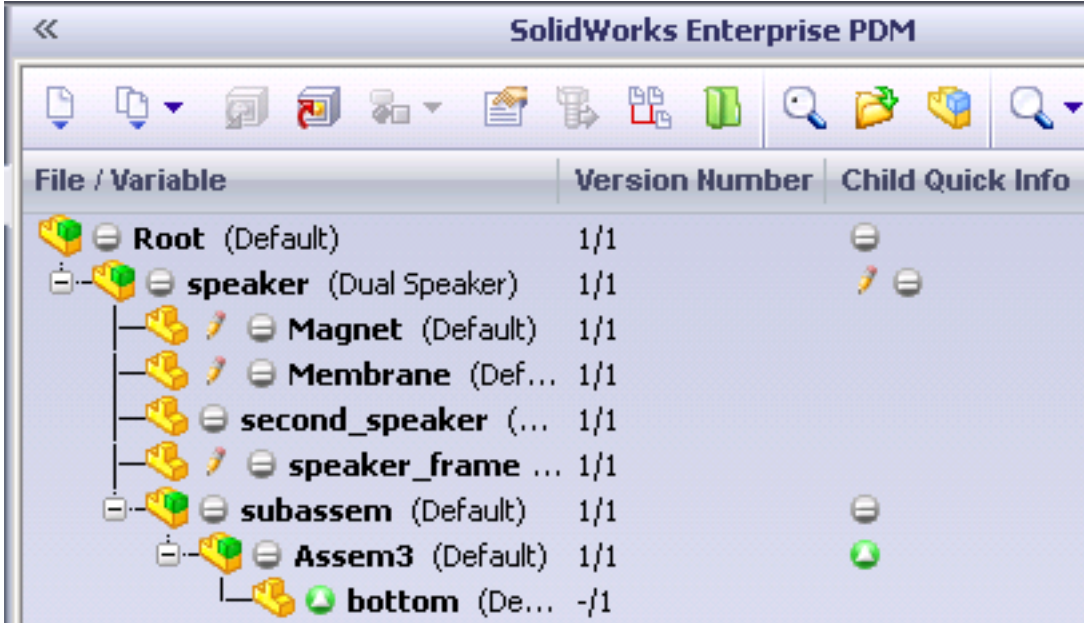
When users run the workflow transition by changing the state of a file, the variables' value update but the file is not versioned unless there are other set variable actions for non-version free variables.

Visibility of Child Quick Information Warnings in Parent Assemblies

Child Quick Information warnings in the Enterprise PDM SolidWorks add-in indicate the status of the in-session SolidWorks file and its references. These icons now appear at the top assembly level for all lower level files.
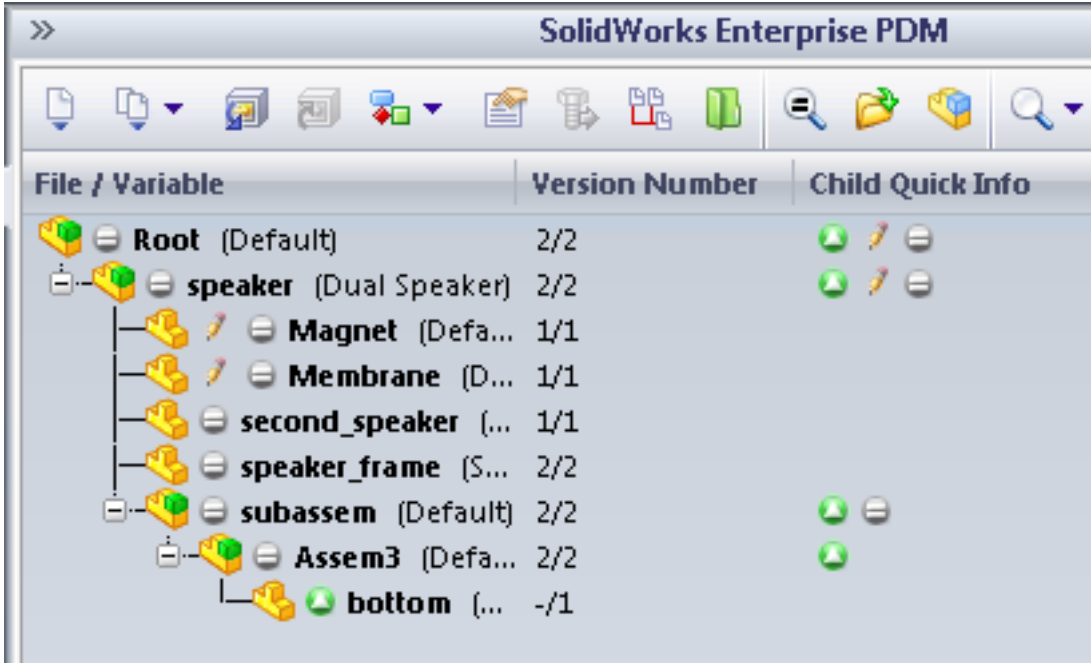
Previously, the icons only appeared up one level. You needed to completely expand an assembly with more than two levels to view the status of lower level files.

SolidWorks Enterprise PDM 2013

SolidWorks Enterprise PDM 2014

14
SolidWorks Flow Simulation

SolidWorks Flow Simulation is a Computational Fluid Dynamic (CFD) 3D solution available as a separately purchased product that can be used with SolidWorks Standard, SolidWorks Professional, or SolidWorks Premium.

This chapter includes the following topics:

- **CircuitWorks and SolidWorks Flow Simulation**
- **eDrawings Support**
- **Mesh Control**
- **Mesh Parallelization**
- **Parametric Study Mode**
- **Usability Improvements**

CircuitWorks and SolidWorks Flow Simulation

With CircuitWorks, an add-in to SolidWorks Premium, you can create 3D models from the file formats written by most electrical computer-aided design (ECAD) systems. You can collaborate to design printed circuit boards (PCBs) that fit and function in SolidWorks assemblies.
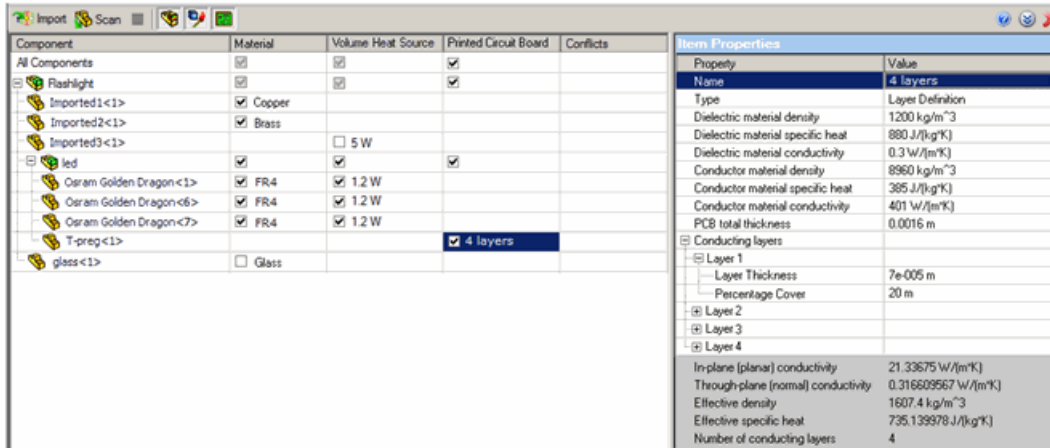
You can import the following CircuitWorks ECAD file PCB or component thermal properties to SolidWorks Flow Simulation:

- Dielectric and conductor density
- Specific heat
- Conductivity for PCBs
- Conductivity for volumetric heat sources from components

When you conduct thermal analysis with SolidWorks Flow Simulation, the SolidWorks model references the imported customer thermal properties.

Two options in the Flow Simulation tree facilitate setting up the Computational Fluid Dynamics (CFD) analysis for cooling simulation of electronic components:

- Right-click **Heat Sources** and select **Import volume source from model**. Select the heat sources to import in **Item properties**.
- Right-click **Printed Circuit Boards** and select **Import Printed Circuit Boards from model**. Select the PCBs to import in **Item properties**.

eDrawings Support

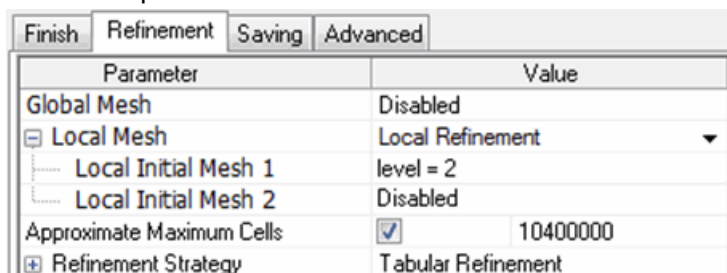You can export multiple SolidWorks Flow Simulation plots in a single eDrawings file.
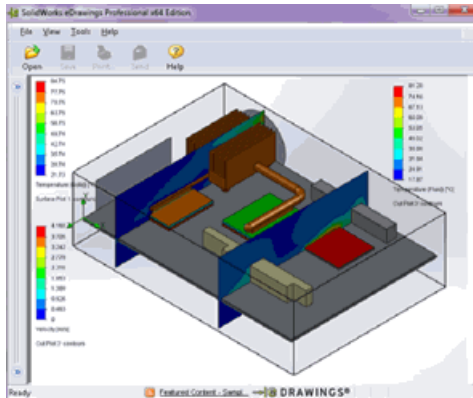
Mesh Control

Solution-adaptive meshing is a procedure for adapting the computational mesh to the solution during the calculation. SolidWorks Flow Simulation extends the solution-adaptive refinement procedure to local regions of the computational domain for more accurate and meshed-optimized CFD simulations.

Mesh Parallelization

SolidWorks Flow Simulation speeds up the meshing task by using multiple cores.

The solver and the mesher can now leverage multi-core processing capability.

Select the number of cores to use for meshing in the Run and Batch Run dialog boxes.

Parametric Study Mode

Parametric Study Mode lets you run multiple fluid flow tests automatically. You can evaluate the intuitive results to choose the best design.

You can:

- Save studies in the top project tree using the parametric study feature for better visibility of the CFD simulations and easier investigation of the design scenarios.

- Save selected Design Points into the project tree to save only desired scenarios. Optionally, you can save the full results of the Design Points.

Usability Improvements

- The **Leak Tracker** tool is available at the bottom of the graphical screen for better visibility. You can locate holes or displacement gaps in the model by visualizing the route from an internal face to an external face of the component.
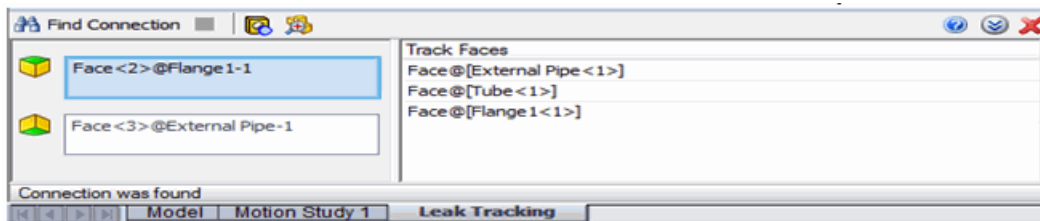
- You can modify boundary and heat source conditions from a single PropertyManager, which accelerates the setup of the CFD analysis for large projects.

- You can define goals for surface area and volume with the **Equation Goal** tool. You can add surface area and volume as reference geometries into the **Equation Goal**. The tool is located in the model **View**, at the bottom of the graphical area, improving its visibility.

- You can specify ambient velocity via aerodynamic angles (angle of attack and angle of sideslip) instead of a 3D vector. This improvement makes it easier to perform a Parametric study.

15
Parts and Features

This chapter includes the following topics:

Conic Fillets ★

You can use the **Fillet** tool to create symmetrical conic shaped fillets for parts, assemblies, and surfaces.
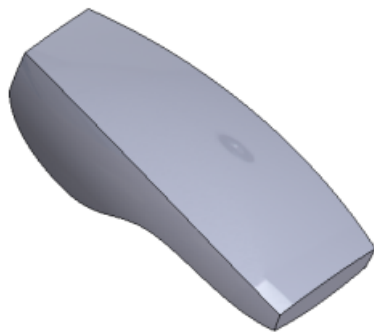
You can apply conic shapes to **Constant Size**, **Variable Size**, and **Face** fillets.

Creating Constant Size Conic Fillets

In this example, the conic shape you create is a hyperbola.

To create this type of conic fillet:

1. Open *install_dir*\samples\whatsnew\parts\remote_control.sldprt.

2. Click **Fillet/Round** (Features toolbar) or **Insert** > **Features** > **Fillet/Round**.
3. In the PropertyManager, under **Fillet Type**, select **Constant size**.
4. Select the top face of the model.

   Face<1> is added to **Items to Fillet**.

Tangent propagation and **Full preview** are selected.

5. Under **Fillet Parameters**, type 5mm for **Radius**.

6. Under **Fillet Parameters**:

    a) Set **Radius** ⬈ to 5mm.
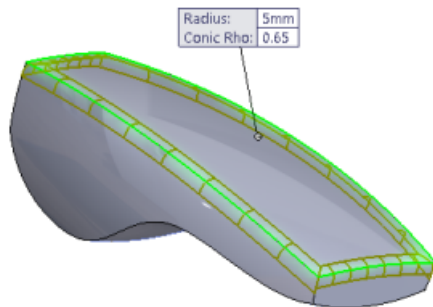
    b) For **Profile**, select **Conic Rho**.

    c) Set **Rho Radius** ρ to 0.65 degrees.

    Rotate the model to view the curvature of the fillets. Use the callouts in the graphics area to change values for **Radius** and **Conic Rho**. The shape of the fillets adjust dynamically.

7. Click ✔.
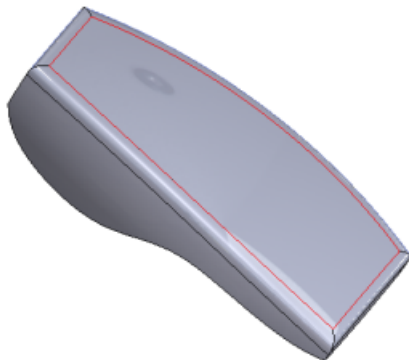
Extruded Cuts

You can create extruded cuts in both directions with one click.

In the Cut-Extrude PropertyManager, select **Through All - Both Directions**. Direction 2 automatically updates with the same value.

Previously, you had to specify **Through All** for each direction separately.

Holes

Creating Slots from the Hole Wizard

You can create counterbored, countersunk, and straight slots from the Hole Wizard.

To create a Hole Wizard slot for a part:

1. Select a face and click **Hole Wizard** (Features toolbar) or **Insert** > **Features** > **Hole** > **Wizard**.
2. In the PropertyManager, on the Type tab, for **Hole Type**, select a slot type:

Option	Description
Counterbore Slot	
Countersink Slot	
Straight Slot	

3. Under **Hole Specifications**, for **Slot length** , enter a value.

4. Select other options and click .

You can also create Hole Wizard or hole series slots in assemblies.

Displaying Decimal Values for Drill Hole Sizes

When specifying straight hole or slot sizes, you can display decimal hole sizes for letter, number, or fractional drill sizes.

You can display decimal sizes for straight holes or slots in the Hole Wizard Type PropertyManager, or the Hole Series (First Part) PropertyManager.

To open the Hole Wizard Type PropertyManager from a part, click **Hole Wizard** (Features toolbar) or **Insert** > **Features** > **Hole** > **Wizard**.

To open the Hole Wizard Type PropertyManager from an assembly, click **Hole Wizard** (Features toolbar) or **Insert** > **Assembly Feature** > **Hole** > **Wizard**.

To open the Hole Series (First Part) PropertyManager, from an assembly, click **Hole Series** (Features toolbar) or **Insert** > **Assembly Feature** > **Hole** > **Hole Series**.

To display decimal values for hole sizes from either PropertyManager:

1. Select a straight hole or slot:

    *   In the Hole Wizard Type PropertyManager, for **Hole Type**, select **Hole** or **Slot** .

    *   In the Hole Series (First Part) PropertyManager, for **Start Hole Specification**, select **Hole** or **Slot** .

2. For **Standard**, select one of the following:

    *   **ANSI Inch**
    *   **DME**
    *   **PCS**
    *   **Progressive**
    *   **Superior.**

3. For **Type**, select one of the following:

    *   **All Drill Sizes**
    *   **Letter Drill Sizes**
    *   **Fractional Drill Sizes**
    *   **Number Drill Sizes**

4. Set other options and click ✔ .

Mass Properties

Volumetric Units

Volumetric units, such as gallons, liters, cubic yards, and cubic decimeters, are supported by Mass Properties.

More Options for Deleting Features ★

When you delete a parent feature that has child or absorbed features, you can select which features to delete.

The Advanced Confirm Delete dialog box lets you selectively delete child and absorbed features. The features that you retain might display warnings or error messages in the FeatureManager design tree. You must manually resolve any errors.

Deleting Absorbed and Child Features

To delete absorbed and child features in a model:

1. Open *install_dir*\samples\whatsnew\parts\arm.sldprt.
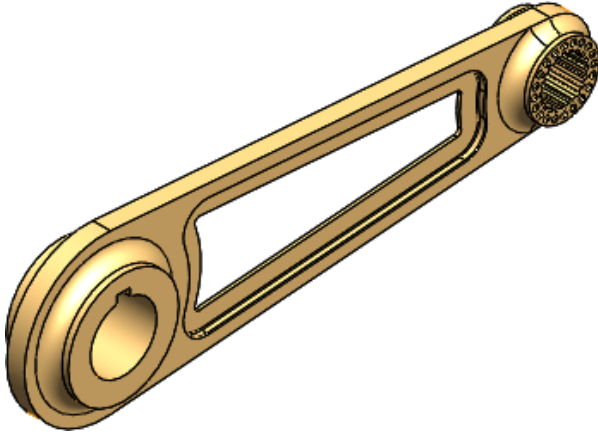
2. In the FeatureManager design tree, right-click **Cut-Extrude2** and select **Delete** ✕.

   The Confirm Delete dialog box appears in **Basic** mode and lists all dependent items related to **Cut-Extrude2**. You cannot delete items individually in Basic mode.

3. Click **Advanced**.

   The dialog updates from **Basic** to **Advanced** mode. This mode lets you select which items to keep and which to delete.

4. Click to clear **Fillet2**.

   Mirror1, **Sketch7**, and **Cut-Extrude4** are selected for deletion.

5. Click **Yes**.

   A warning displays for **Fillet2**. You can ignore the error or repair the model before continuing with your work.

Patterns

Highlight Seed Feature

To identify a seed feature in a pattern more easily, click the pattern in the FeatureManager design tree. The seed highlights in a different color from the pattern instances.

Pattern instances are highlighted in **Selected Item 1** color, and seeds in **Selected Item 2** color. You can change these colors by clicking **Tools** > **Options** > **Colors**.

Display Tooltips with Pattern Information

You can display a tooltip about a pattern.

When you set this system option, information displays when you hover over the pattern name in the FeatureManager design tree.

Information about the pattern includes:

- Pattern name
- Pattern type
- All seeds used to create the pattern
- Spacing and number of instances for Direction 1 and Direction 2
- Instances skipped
- Instances varied

Click **Tools** > **Options**. On the System Options tab, click **Display/Selection** and select or clear **Display pattern information tooltips**.

Multibody Support for Pattern and Mirror Features

When pattern or mirror features split single bodies into multiple bodies, you can select which bodies to keep using the Bodies to Keep dialog box.

You can also specify which configurations to keep up to date.

Previously, the SolidWorks software randomly selected bodies to keep.

Reference Geometry

Inferenced Reference Points

The Point PropertyManager has additional items for creating reference points.

You can create reference points at the intersections of the following entities:

- An axis and a plane
- An axis and a surface, both planar and non-planar
- Two axes

You can use the **On Point** option to add reference points on a sketch point and on the end of a sketch section.

Plane Alignment

When creating a plane using a line and a plane or a face, the SolidWorks software aligns the X-axis of the new plane with the line.

This ability delivers a more predictable orientation when sketching on new planes, and corrects inconsistencies in previous versions.

Reference Plane Normal

The **Flip Normal** option in the Plane PropertyManager lets you flip the normal vector of a manually created reference plane.

An indicator arrow displays the direction of the normal vector in the graphics area.

This option is not available for default reference planes.

Weldments

Propagate Cut List Properties

The workflow has improved for propagating cut list properties to a new part. You can use the **Insert Into New Part** command to copy cut list properties to the file level or the cut list level.

The **Save Bodies**, **Insert Part**, and **Insert Mirror Part** commands have this same capability, which helps with designing multibody parts.

Right-click the **Cut List Properties** folder in the FeatureManager design tree and click **Insert Into New Part**.

Structural Members as Library Features

You can configure multiple weldment profiles of structural members as library features.

You can add different configurations of a structural member and save them as one profile in a library feature.

For example, instead of having 50 separate library feature files for square tubing sizes, you can have one library feature file with 50 configurations that you can control by a design table.

When creating or editing structural members, you can select the configured library features in the Structural Member PropertyManager.

16
SolidWorks Plastics

SolidWorks Plastics Professional and SolidWorks Plastics Premium are available as separately purchased products that you can use in SolidWorks Standard, SolidWorks Professional, or SolidWorks Premium.

This chapter includes the following topics:

- **Getting Started Wizard**
- **Help Updates for Dynamic Advisor**
- **Injection Location**
- **Machine Details**
- **Mold Layout Wizard**
- **Runner Design Wizard Gate Descriptions**

Getting Started Wizard

The Getting Started Wizard guides you through the SolidWorks Plastics workflow. It guides you through the process of creating a meshed model, selecting a plastics material, specifying process parameters such as mold and melt temperature plus injection time and pressure, and running an analysis.

The Getting Started Wizard also helps you view and interpret the results, and access the Dynamic Adviser for problem-solving advice and creating reports.

To access the Getting Started Wizard, click **SolidWorks Plastics** > **Getting Started Wizard**.

> The Getting Started Wizard is available in SolidWorks Plastics Professional and SolidWorks Plastics Premium.

Help Updates for Dynamic Advisor

The Dynamic Advisor help has been modified to contain the most up-to-date information.

Injection Location

Gate Selection has been renamed to **Injection Location**.

To more accurately reflect the specific action you perform with this command, **Gate Selection** has been renamed to **Injection Location**. You can specify an injection location at a location other than a gate, for example, at the beginning of a sprue. The functionality does not necessarily involve or create actual gate geometry.

Machine Details

You can access machine details in the Fill Settings PropertyManager.

Modifications in the Fill Settings PropertyManager:

- **Flow Settings** has been renamed to **Fill Settings**.
- The PropertyManager includes the **Maximum Injection Pressure** option.
- You are no longer required to select or specify an injection molding machine during the analysis setup workflow. Instead, in the PropertyManager, you can specify the maximum injection pressure limit.
- You can access the machine database under **Advanced**.

Mold Layout Wizard

The Mold Layout Wizard guides you through creating and analyzing single-cavity, multi-cavity, and family mold layouts. The Mold Layout Wizard helps you create feed system components, such as sprues, runners, and gates. It also shows you how to automatically balance family mold runner systems.

To access the Mold Layout Wizard, click **SolidWorks Plastics** > **Mold Layout Wizard**

The Mold Layout Wizard is available in SolidWorks Plastics Premium only.

Runner Design Wizard Gate Descriptions

The following gates have been renamed to reflect generally accepted industry terminology.

| Previous Gate Name	New Gate Name
Side	Edge Gate
Submarine	Submarine Gate
Horn	Banana Gate

17
Routing

Available in SolidWorks Premium.

This chapter includes the following topics:

- **Display Colored Connector Pins**
- **Edit Connector Tables**
- **Electrical Flattening Enhancements**
- **Enhancements to Routing Library Manager**
- **O-let Improvements**
- **Slope Improvements**
- **Spool Enhancements**
- **Support for Assembly Connectors in From-To Lists**
- **Support for Crimps and Terminals in Connector Library**

Display Colored Connector Pins

To help you connect wires to pins, wire pins for flattened routes are colored when they display in connector blocks. The pin colors match the color of the wires used in the 3D Route. These pin colors are also listed in the connector tables.
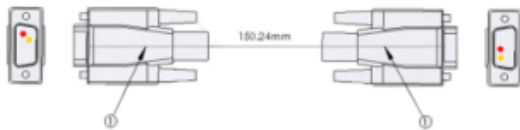
Edit Connector Tables

A BOM-based connector table is available using the `.sldbomtbt` file extension. BOM-based connector tables preserve edits through model updates.

You can also save your connector tables as templates. Column selection, column order, and customized headers are stored in templates for re-use. You can still create earlier versions of connector tables using the `.sldtbt` templates.

BOM-based connector tables can also display additional details about the pin connections if they are present in the model. For example, terminals, seals, and plugs are available in **Column Properties**.

Previously, updates to the model would undo any text added to the connector tables.

Electrical Flattening Enhancements

Enhancements for electrical flattening include the ability to:

- Route with loops. Loop segments display in dotted lines for annotation and manufacturing flattened routes.
- Support coverings and clips.
- Modify the connector locations and orientations of flat routes. By rotating connectors, you can place the desired connector face on top in flattened drawings.
- Adjust multi-pin connector fan-out length.
- Adjust the formboard size for manufacturing flattened routes.
- Select items to display for flattened route drawings. The items include balloons, clips, cable lengths, connector tables, coverings, wire balloons, circuit summary and electrical BOMs.

Enhancements to Routing Library Manager

The Routing Library Manager supports assembly fittings for piping and tubing, and assembly connectors for electrical routing. You can use the Routing Component Wizard to ensure that your assembly fittings and connectors have the necessary requirements for use in route assemblies.

Previously, the Routing Component Wizard supported only parts.

O-let Improvements

Using a sketch dimension, you can set a dimension to an o-let stub to locate it with a linear dimension or orient it with an angular dimension.

Existing o-lets which are mated in place will continue to solve until you delete and re-insert them.

Slope Improvements

Improvements for slopes include the ability to drag fittings and to add multiple route segments to existing sloped routes.

You can also add weld gaps to segments contained in the slope.
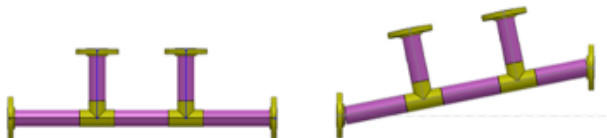
To add a slope to a pipe route, open a routing assembly and click **Piping** > **Edit Route**
. Right-click the sketch entity at the location where you want to add the slope and select **Add Slope** .

Spool Enhancements

You have more flexibility when creating spools. You can define spools using the following components:

- non-continuous or disjoint route segments
- components mated to end fittings
- hangers and supports
- pipes with penetration points
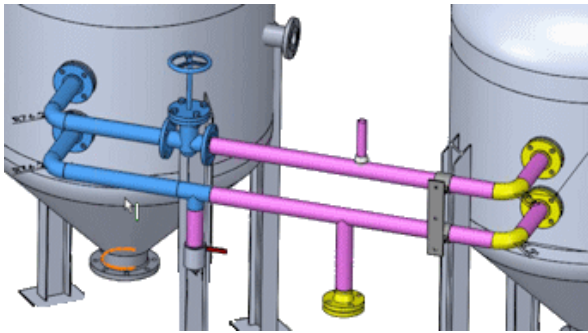
Previously, you could only create spools with route segments and fittings that were contiguous.

To define spools in a routing assembly, **Routing** > **Piping** > **Define Spools** .

Support for Assembly Connectors in From-To Lists

You can use assembly connectors in From-To Lists. The details about the assembly connectors display in connector tables and BOMs.

Support for Crimps and Terminals in Connector Library

You can pre-define terminals, seals, and miscellaneous components using the `components.xml` and `.xls` files.

This additional information is automatically added to wire and cable connections while creating electrical routes.

You can also define terminals, seals, and miscellaneous components for each wire and cable connection in the Edit Wires PropertyManager.

The details of these components display in BOMs.

A sample library of these components is also provided with the Routing Library and is named `InterconnectAndAccessoriesLibrary.xml`.

18
Sheet Metal

This chapter includes the following topics:

- **Adding Sheet Metal Gussets**
- **Bent Lofted Bends**
- **Corner Reliefs and Bend Transitions**
- **Swept Flanges for Conical and Cylindrical Bodies**
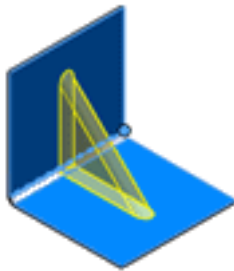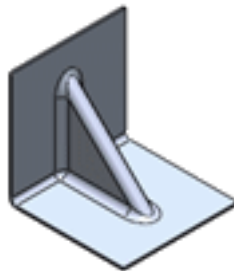
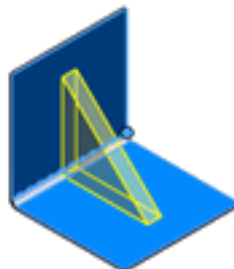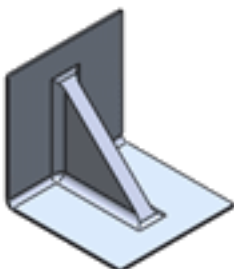## Adding Sheet Metal Gussets

You can create sheet metal gussets with specific indents that go across bends.

To add sheet metal gussets:

1. Click **Sheet Metal Gusset** (Sheet Metal toolbar) or **Insert** > **Sheet Metal** > **Sheet Metal Gusset**.
2. In the Sheet Metal Gusset PropertyManager, under **Position**, select:

   - The bend face or two planar faces that are adjacent to a bend.
   - The edge to be used as a reference.

     This can be any linear entity in order to specify the alignment of the gusset.

   - The point to be used for dimensioning.

     If you pick the edge you automatically get an endpoint as a reference. However, you can also choose another entity, such as a sketch point or an endpoint of a line.

     To move the gusset in relation to the selected point, check **Offset** and specify the offset amount.

3. Under **Profile**:

   a) Do one of the following:

   - For a gusset that is symmetrical relative to the bend face, select **Indent depth** and specify a depth.
   - For an asymmetrical gusset, select **Profile dimensions**.

     Either specify the depth (**d1**) of one gusset leg and specify an angle (**a1**) or specify the depth of each gusset leg.

To reverse the lengths of the legs, click **Flip dimension sides**.

b) Select a gusset type:

Option	Button	Preview	Gusset
Rounded rib			
Flat rib			

c) If you choose **Flat rib**, optionally, specify an **Edge Fillet**.

4. Under **Dimensions**:
    a) Enter a value for **Indent width** to specify the width of the gusset.
    b) Enter a value for **Indent thickness** to specify the gusset wall thickness.
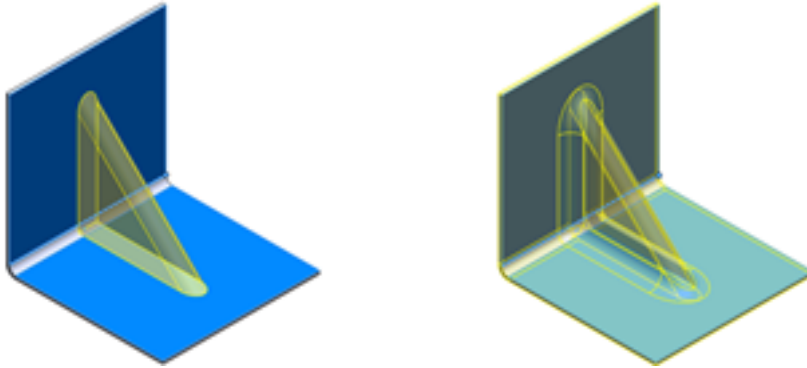
    c) To specify an angle for the gusset sides, click and enter a value for **Side face draft**.

    d) Optionally, change the values for **Inner corner fillet** and **Outer corner fillet**. You can also disable the buttons to remove fillets.

5. Under **Preview**, select one of the following:

**Partial preview**   **Full preview**

6. Click ✔ .

Bent Lofted Bends ★

You can create a bent lofted bend to create physical bends, rather than formed geometry and approximated bend lines in a flat pattern. Bent lofted bends form a realistic transition between two parallel profiles to facilitate instructions for press brake manufacturing.
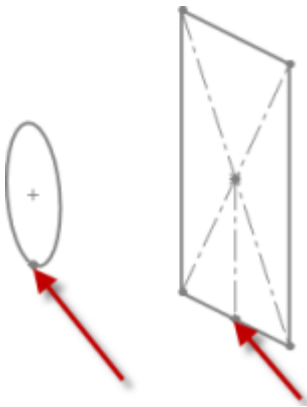
To create a bent lofted bend:

1. Open *install_dir*\samples\whatsnew\sheetmetal\lofted_bends.SLDPRT.
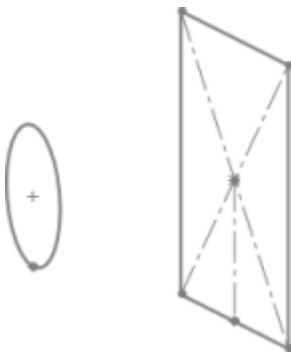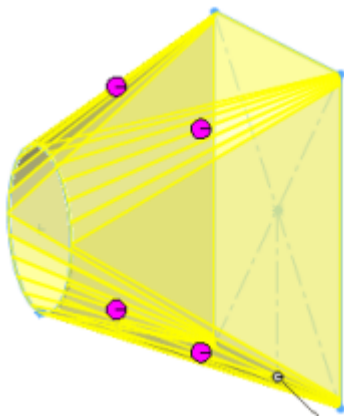

2. Click **Lofted Bends** 🔩 (Sheet Metal toolbar) or **Insert** > **Sheet Metal** > **Lofted Bends**.

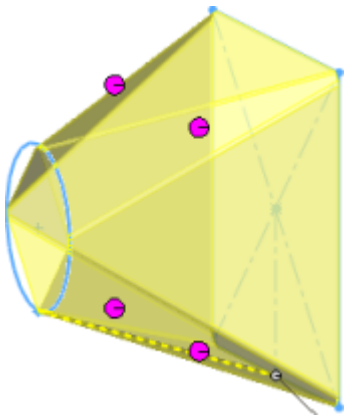3. Under **Manufacturing Method**, select **Bent**.

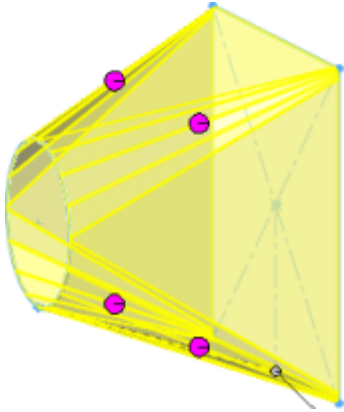4.   Select both profiles at the open points shown.

The preview appears:

5.   Under **Faceting Options**, select **Number of bends** to specify the number of bends for each segment between a vertex and the related portion of the arc.

6. Under **Facet Value**, type `5` for the **Facet Value**.

7. Select **Refer to endpoint** to calculate the transition between segments at the sharp corner.

8. Under **Sheet Metal Parameters**, select **Override default parameters**, in **Thickness**, type `1.0`, and in **Bend Radius**, type `1.0`.

9. Under **Bend Allowance**, select **K-Factor**, and type `0.5`.

10. Under **Auto Relief**, select **Rectangular**, select **Use relief ratio**, and in **Ratio**, type `0.5`.

11. Click ✔ .
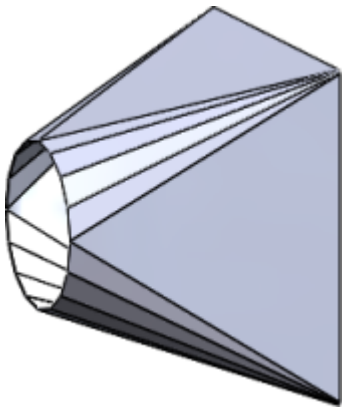
Corner Reliefs and Bend Transitions

You can apply corner treatments to a folded sheet metal body that will persist in the flattened state. You can view how the corner treatment impacts the folded sheet metal body and how it carries over to the flat pattern.

You can use any of the following corner treatment types:

Rectangular

Circular			
Tear			
Obround			
Constant Width			
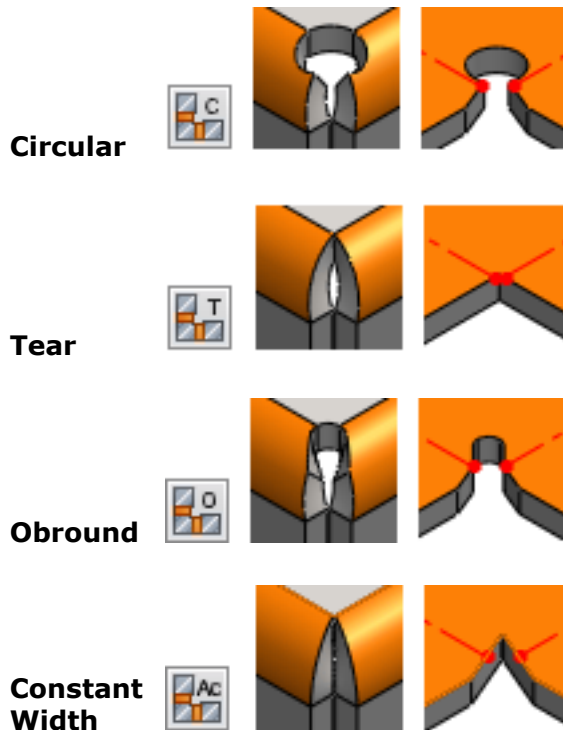
You can create configuration-specific corner reliefs and suppress or unsuppress them within configurations. For example, for configuration A, you can define a corner relief by selecting all corners and applying a circular relief. In configuration B, you can select all corners and apply a rectangular relief that is suppressed in configuration A and vice versa.

Adding a Corner Relief

You can add a corner relief to a single sheet metal body.

1. Click **Corner Relief** (Sheet Metal toolbar) or **Insert** > **Sheet Metal** > **Corner Relief**.
2. In the Corner Relief PropertyManager, under **Scope**, select the sheet metal body to which to apply the corner relief.
3. Under **Corners**, click **Collect all corners** to list all corners in the sheet metal body.

   If you list multiple corners, you can specify multiple corner relief treatments.

   When you select a corner in the **Corners** section, the faces that define the corner are shown in the **Define Corner** section and the corner is highlighted in the graphics area.
4. To add an individual corner instead of collecting all corners, in the **Define Corner** section, select the faces that define the corner and click **New Corner**.

   The corner is added to the list in the **Corners** section.
5. With a corner selected, under **Relief Options**, select the type of relief to apply.

   The relief type determines other options you can specify:

Available options	Rectangular	Circular	Tear	Obround	Constant Width
Centered on bend lines	Yes	Yes	NA	Yes	NA
Slot Length	Yes	Yes	NA	Yes	NA
Ratio to thickness	Yes	Yes	Yes	Yes	Yes
Tangent to bend	Yes	Yes	NA	NA	NA
Add filleted corners	Yes	NA	NA	NA	NA
Corner Fillets	Yes	NA	NA	NA	NA
Slot Width	NA	Yes	NA	Yes	NA
Narrow corner	Yes	Yes	NA	Yes	NA

Previews of your choices appear in the graphics area.

6. Click ✔ .

7. Click **Flatten** 🔲 (Sheet Metal toolbar) to see how the corner relief appears in the flattened sheet metal part.

Swept Flanges for Conical and Cylindrical Bodies

The **Cylindrical/Conical Bodies** section of the Swept Flange PropertyManager lets you select a linear sketch entity that propagates to the flat pattern as a fixed entity.

> The **Cylindrical/Conical Bodies** option is only available when the selected path along which the profile sweeps is a sketch.

With the **Swept Flange** tool, you can use composite sketch contours that are swept along a circular path. When you select a linear sketch entity to propagate, the software can correctly flatten cylindrical or conical shapes that include features such as rolled ribs or flanges that have been added as seams.

To specify a cylindrical flat pattern, you select a sketch entity that is horizontal or vertical with respect to the sweep path. To specify a conical flat pattern, you select a slanted sketch entity.

Without the **Cylindrical/Conical Bodies** option, when you flatten a conical body, the flattened shape is a rectangle.
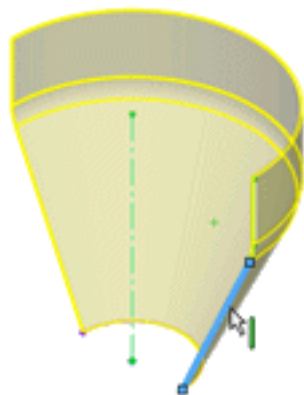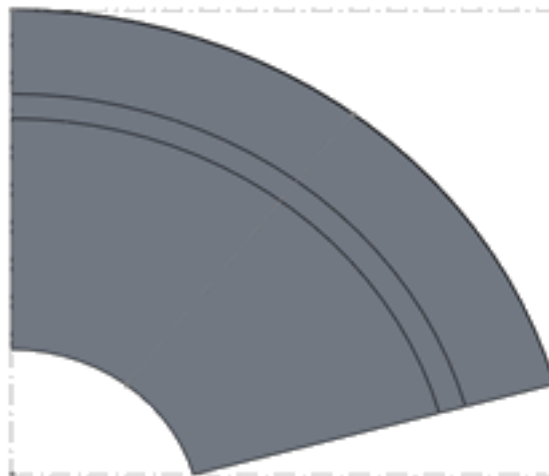
Conical shape

Flattened without using the **Cylindrical/Conical Bodies** option

When you use the **Cylindrical/Conical Bodies** option to select a slanted sketch entity, the flattened pattern is conical.


Slanted sketch entity selected

Resulting conical flattened pattern

## Creating a Conical Body with a Swept Flange

When you add a swept flange to a conical sheet metal body, you can use the **Cylindrical/Conical Bodies** section to ensure that the resulting body flattens correctly.

The **Cylindrical/Conical Bodies** option is only available when the selected path along which the profile sweeps is a sketch.

1. Click **Swept Flange** (Sheet Metal toolbar) or **Insert** > **Sheet Metal** > **Swept Flange**.

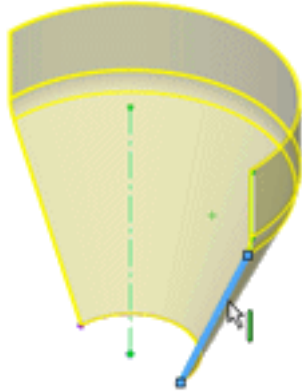2. Select the profile and path for the swept flange feature.
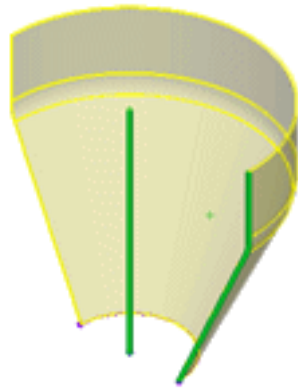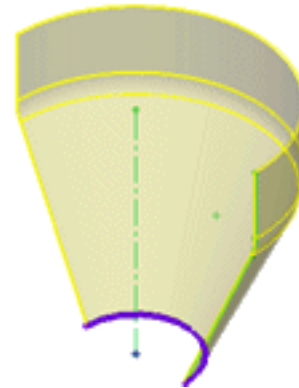
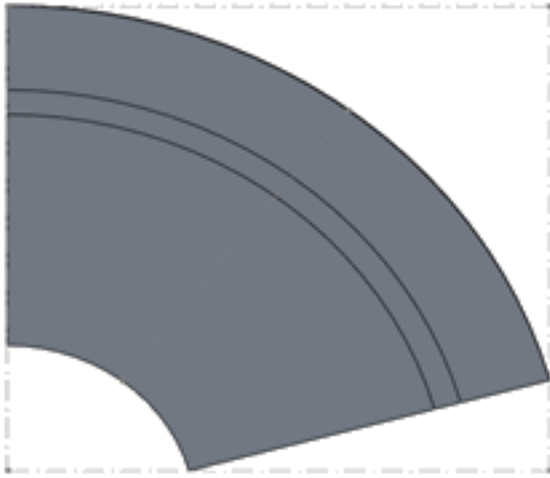

Profile   **Path**

3. Expand **Cylindrical/Conical Bodies** and click the check box to enable the option.

4. Select a linear sketch segment that is part of the profile to act as a fixed entity when you unfold the part.

5. Click ✔ .

6. Click **Flatten** (Sheet Metal toolbar) to see the resulting flat pattern.

19
SolidWorks Simulation

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.

The linear static analysis capability is available in SolidWorks Premium.

This chapter includes the following topics:

- **Connectors**
- **Contact**
- **Materials**
- **Performance**
- **Post-Processing**
- **Importing Results from SolidWorks Plastics**

Connectors

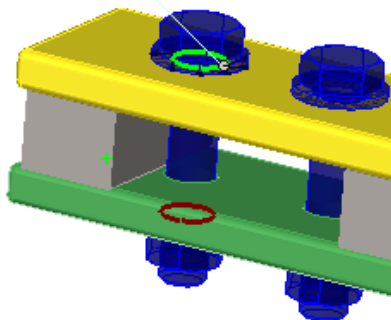### Automatic Conversion of Toolbox Fasteners to Bolts ★

You can automatically convert Toolbox fasteners in models to Simulation bolt connectors.

Available for SolidWorks Simulation Professional and above.

To run the automatic detection tool, in the Study PropertyManager, under **Options**, select **Convert Toolbox fasteners to bolt connectors (may take time)**.

This tool is available for linear static, nonlinear static, and nonlinear dynamic studies.

During the conversion process, all information related to the location, geometric features, and material of the Toolbox fasteners is mapped internally to the formulation of the corresponding bolt connectors.

Head Diameter (mm)	24
Nut Diameter (mm)	18
Nominal Shank Diameter (mm)	12
Axial load (N)	9605.78896309

In the Connector (Bolt) PropertyManager these fields are populated during conversion:

Type	All geometric entities are populated for these type of connectors:
	Standard or Counterbore with Nut
	Countersink with Nut
	Standard or Counterbore Screw
	Countersink Screw
Material	If the Toolbox fastener has material assigned, then the same material is used for the bolt connector definition. Otherwise, the default material Alloy Steel is used.

Strength data	**Tensile stress area**	ISO Bolt threads: $A_t = 0.7854 * [d - (0.9382 / n)]^2$, where d is given in mm, and n is given in threads/mm.
		ANSI threads: $A_t = 0.7854 * [d - (0.9743 / n)]^2$, where d is given in inches, and n is given in threads/inch.
		D = nominal diameter of bolt (from the Toolbox Fastener data), and n is number of threads per inch (mm) (from the Toolbox Fastener data)
		Reference: Machinery's Handbook published by Industrial Press.
	Bolt Strength	Equals the Yield Strength of the fastener's material.

Pre-load F_I = 0.75 x A_t x 0.20 Yield Strength

Reference: Machinery's Handbook published by Industrial Press, 27th Edition – pg. 1495.

You can view the Simulation bolt connectors that were created under **Connectors** 🔩 .

Bolt connectors share the naming convention of their Toolbox counterparts that have the same size and length. Bolt connectors of the same size are grouped into a subfolder. The Toolbox components excluded from the analysis are listed in **Auto-Converted Toolbox Fasteners** under **Parts**.

- Review and verify the properties of the converted connectors before you proceed with the analysis.
- You still need to define a contact condition (no-penetration in most cases) to prevent interference between bodies fastened by the Toolbox fasteners.
- Toolbox items such as gears and pins are not automatically converted to Simulation connectors.

Listing Connector Output Forces

For spring, spot weld, and link connectors, you can list the output forces after you run a static or nonlinear analysis.

Right-click **Results** 📁 and click **List Connector Force**.

The **List Pin/Bearing Force** option is removed. Listing of output forces for all connectors is available from the Result Force/Connector force PropertyManager.

You can list the results for all connectors in the current study, or filter the results for selected connectors only. Available selections are:

- **All connectors**
- **All pins**
- **All bolts**
- **All springs**

- **All spot welds**
- **All links**
- **All bearings**
- Individual connectors

When you select a row in the table, the corresponding connector highlights in the graphics area and a callout lists all forces.

Press **Shift** to select multiple, consecutive rows. Press **Ctrl** to select multiple, individual rows.

To export all connectors' forces to a Microsoft Excel spreadsheet, click **Save to file** 📑.

Contact

Contact Pressure Plots in 2D Simplification

You can view contact pressure plots in linear and nonlinear 2D Simplification studies.

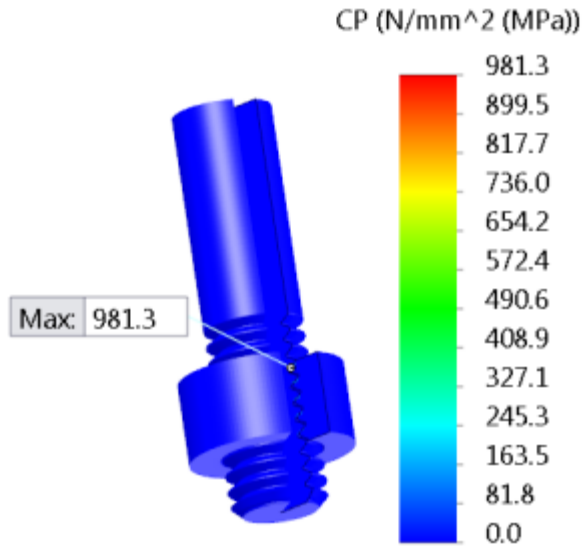Available for SolidWorks Simulation Professional and above.

In the Stress Plot PropertyManager, under **Component** 🌈 , select **CP: Contact Pressure**.

Contact pressure is developed in areas where two bodies come into contact during the analysis and no penetration contact is defined. To enhance the visibility of areas that come into contact and their view is obstructed, an option to show the contact pressure plot only on selected entities is available.

In the Stress Plot PropertyManager, under **Advanced Options**, select **Show plot only on selected bodies**. Select the faces to show the contact pressures.

You can view the contact pressures in 2D plots as vectors, and also in 3D plots. In the Stress Plot PropertyManager, under **Display**, select **Show as 3D plot** or **Show as vector plot**.

For 3D plots, the results are extruded across the section depth as defined in the properties of the study (plane stress, plane strain), or revolved about the axis of symmetry (axi-symmetric studies).

CP (N/mm^2 (MPa))

	981.3
	899.5
	817.7
	736.0
	654.2
	572.4
	490.6
	408.9
	327.1
	245.3
	163.5
	81.8
	0.0

Max: 981.3

Contact Visualization Plot ★

With a new contact visualization plot, you can inspect all the contacts created in the model.

Available for SolidWorks Simulation Professional and above.

A contact visualization plot allows you to detect easily the different types of contact in your model applied with global, component, or contact set definitions.

Right-click **Connections** ▯ and click **Contact Visualization Plot** 🔧.

The regions of the model where contact is defined are rendered in a color unique to each contact type. The types of contact supported are:

Contact Type	Color Rendering
Bonded	🟥 Red
No Penetration	🟪 Purple
Allow penetration (Free contact)	🟩 Green
Shrink Fit	🟧 Orange
Virtual Wall	🟨 Yellow
Thermal Contact Resistance	🟪 Purple
Insulated	🟩 Green

In the Contact Visualization Plot PropertyManager click **Calculate** to view the geometry-based contact pairs. The geometry-based contacts include all geometric entities (edges, faces or bodies) that are initially touching (before starting the analysis) and have a contact definition assigned automatically or manually.

To view at the mesh element level the areas of the model detected by the solver (before starting the analysis) with an assigned contact definition, select **Include solver generated contacts (mesh required)**.

Under **Results**, select **Show solver-based contacts**. The Contact Visualization Plot renders the mesh elements that belong to the geometric entities of the model with assigned contact definitions.
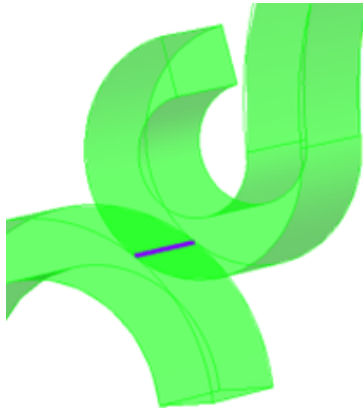
Plot of geometry-based no-penetration contact. Rendering of the touching edge between the two components.
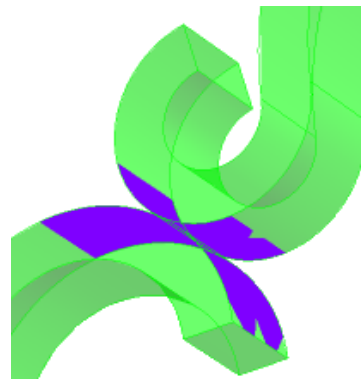
Plot of solver-based no penetration contact. Rendering of mesh elements that participate in contact formulation.

Materials

SolidWorks Simulation Professional and SolidWorks Simulation Premium users with active SolidWorks Simulation subscription services can access a much larger library of materials via the online SolidWorks materials portal.

Available for SolidWorks Simulation Professional and above.

An extended material database is provided through a partnership with Materiality LLC.

You can search for materials based on the type of simulation study you want to perform, or the material model you want to use. Materials downloaded from the online SolidWorks materials portal are automatically added to the materials list available in the SolidWorks Simulation Material dialog box.

Performance

Performance improvements include the reduced solution time and better convergence of results for certain nonlinear studies, the reduced time for initial loading of Simulation studies, and the availability of a new direct sparse solver for large problems.

Enhancements for nonlinear studies include:

- Better convergence for node-to-surface no-penetration contact formulation with overall performance improvement for up to 30%.
- Improved accuracy of results for shells with plasticity material models when using small displacement formulation.

Initial Loading of Simulation Studies

The time for performing initial loading of models with Simulation studies is reduced. The performance improvement is more prominent for models with multiple Simulation studies.

A new option **Load all Simulation studies when opening a model (requires more time to open a model)** is available under **System Options**.

Select this option to save all the information for Simulation studies to the memory upon opening a model. It might take longer time to open models with multiple Simulation studies.

If you clear this option, only the basic information for each Simulation study is saved to the memory upon opening a model. When you activate a study, the rest of the Simulation study data is saved to the memory.

Large Problem Direct Sparse

A direct solver is available that leverages multicore processing capability and improves solution speed for static and nonlinear studies.

In the Static or Nonlinear dialog box, under **Solver**, select **Large Problem Direct Sparse**.

By leveraging enhanced memory-allocation algorithms, the solver can handle simulation problems that exceed the physical memory of your computer.

If you initially select the **Direct Sparse** solver and due to limited memory resources it has reached an out-of-core solution, a warning message alerts you to switch to the **Large Problem Direct Sparse**.

Post-Processing

User interface enhancements:

- Common access to Edit Definition, Chart Options, and Settings PropertyManagers.
- Ability to edit the min and max values on plot legends. Click the min or max value of the plot legend and type a new value.
- Quick access to the **Fringe Options** of the plot. Right-click anywhere on the plot legend boundary and click **Fringe Options**.
- Ability to quickly change the display state of mesh on result plots. Right-click anywhere on the plot legend boundary and select or clear **Show Mesh**.

Mirrored Results About Planes of Symmetry

For models where you apply planar symmetry or cyclic symmetry restraints to leverage the geometric and loading symmetry conditions, you can view results on the full model. The display of results on the full model, instead of only a section of the model, gives you better insight on the model behavior and helps you identify potential modeling errors.

In the PropertyManager of the desired result quantity, under **Advanced Options**, select **Display symmetric results**.

The results are mirrored about the planes of symmetry for models where you analyze half, a quarter, or one-eighth of the actual geometry. For circular symmetry, results are repeated cyclically about the defined axis of revolution.

> For circular symmetry, the angle between the selected faces must be evenly divisible into 360. The axis of symmetry must lie at the intersection of the two selected planes where you apply the cyclic symmetry.

The new option for displaying results on the full model is available for stress, displacement, strain, design insight, and factor of safety plots and for all study types that support the planar and cyclic symmetry restraints.

You can save the plots that display the results on the full model to eDrawings files. The load symbols, mesh, and restraint symbols applied to the model are visible only to the geometry sections that are analyzed, and they are not mirrored about the planes of symmetry.

One quarter of the pressure vessel is analyzed by applying symmetry restraints to a set of orthogonal faces

Results shown on the full model geometry

## Results Comparison Across Configurations

With the **Compare Results** tool, you can compare up to four result plots from Simulation studies that are associated with different configurations of the same model.

This cross-configuration comparison mode gives you full control over the display of results by using existing configuration functionality.

To access this PropertyManager, click **Compare Results** (Simulation CommandManager).

Under **Options**, select **All studies in all configurations (slower)** to compare up to four result plots from studies associated with different configurations.

For plots of the same type, you can select the plot which controls the settings.
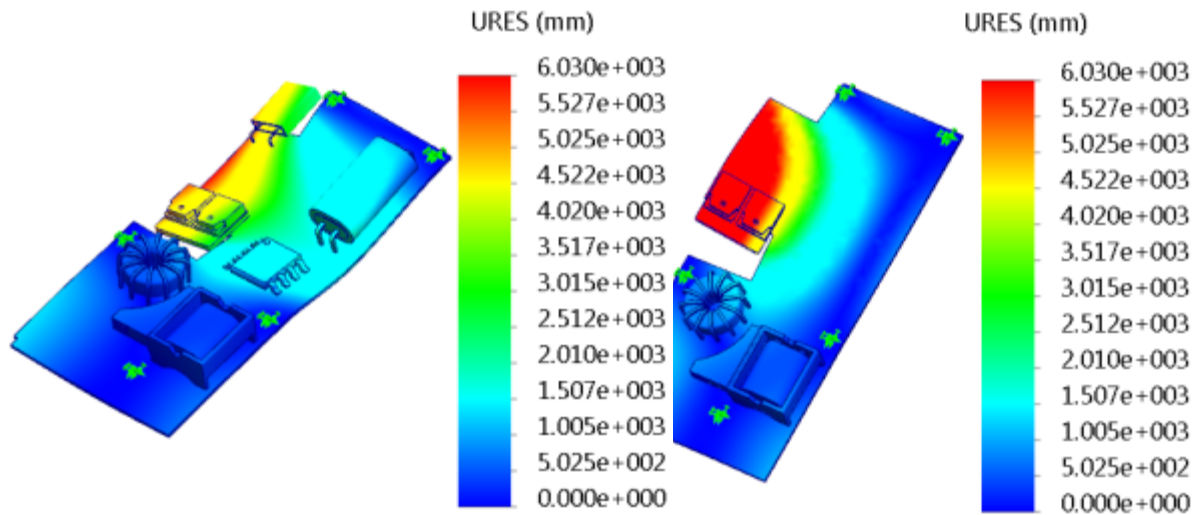
Default Configuration - First Mode Shape   Second Configuration - First Mode Shape
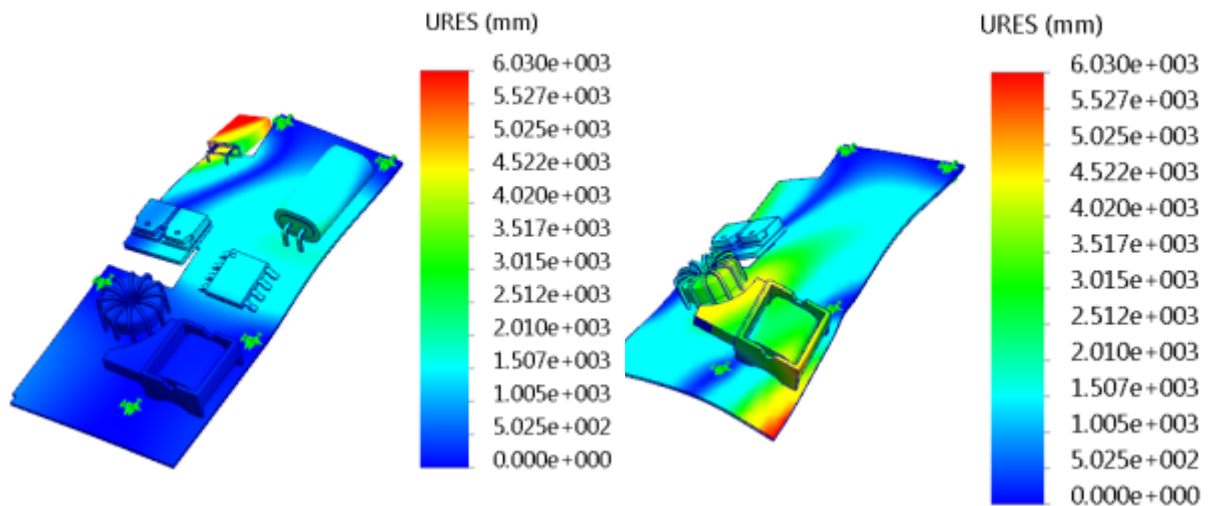
Default Configuration - Second Mode Shape Second Configuration - Second Mode Shape

Importing Results from SolidWorks Plastics ★

For plastic parts, you can import temperatures and in-mold residual stresses from SolidWorks Plastics Premium in a nonlinear static study.

Available for SolidWorks Simulation Premium.

By simulating the cooling down process from temperatures sustained during the injection molding process to room temperature, you can calculate the final residual stresses and displacements of the plastic part.

> You can only import data from SolidWorks Plastics Premium for a single part that meshes with solid elements.

After you run the packing process (Flow + Pack) in SolidWorks Plastics Premium, you can export the in-mold residual stresses and temperature data to a file (`*.POE`) in a binary format that can be read by Simulation. The binary file stores:

- Node coordinates and element connectivity of the original mesh
- Nodal values for temperature and in-mold residual stresses (three normal and three shear stress components)
- Material properties

You import the data stored in the binary file (`*.POE`) in a Simulation nonlinear static study to calculate the final displacements of the part.

Simulation applies a time curve from time t = 0 sec to t = 1 sec to the nodal temperatures to cool uniformly the part to room temperature (298 Kelvin). Shrinkage of the part may occur during the cooling process.

Simulation adds the in-mold residual stress components (at pseudo time t = 0 sec) and the quenching thermal stress components (at time t = 1 sec) to calculate the final results.

> You cannot control how long it will take to cool the part, since time in nonlinear static studies is pseudo time.

Importing Injection Molding Data from SolidWorks Plastics

In this example, you import from SolidWorks Plastics the residual stresses and temperatures generated after the packing phase in a Simulation nonlinear study. You calculate the final stresses and displacements of a plastic cup after it cools down to room temperature.

To open the model, go to:

install_dir\samples\whatsnew\plastics\cup.sldprt.
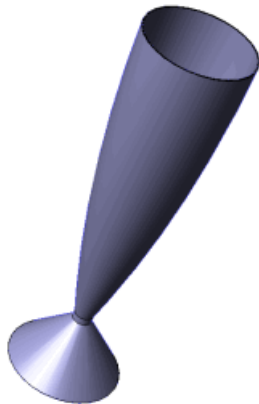
Review the **Ready Study** features:

- Imported in-mold stresses
- Material definition stored in the **Custom Materials** folder
- Mesh properties

Importing Injection Molding Data

You create a nonlinear study and import the in-mold residual stresses and temperatures at the end of the packing phase from SolidWorks Plastics.

1. Create a nonlinear static study with a name `In-mold stresses`.
2. Right-click the new study folder and click **Properties**.
3. In the **In-mold stresses** tab:
   a) Select **Import in-mold stresses from SolidWorks Plastics**.
   b) For **File name**, browse to location *install_dir*\samples\whatsnew\plastics and select `inmold_stresses.POE`.

   The file `*.POE` contains stress and temperature nodal data calculated after the packing process in SolidWorks Plastics. The file is in binary format and you cannot edit it.

   The option **Include material from SolidWorks Plastics** is selected by default, because the material has been assigned for the part in SolidWorks Plastics. To apply different material properties, clear this option and apply a material from the SolidWorks Material database.

4. Click **OK**.

   A new icon 🔵 **In-mold stresses** appears under **External Loads** 📥.

   The nonlinear solver applies the imported in-mold stresses and nodal temperatures from the end of packing phase at time t = 0.

   To simulate the cooling process, it associates internally the in-mold temperature data with a time curve decreasing from the initial values at time t = 0 to a room temperature (298 Kelvin) at pseudo time t = 1.0.

Meshing and Running the study

You mesh the part and run the nonlinear study.

1. Mesh the model using the **Standard** mesh and the default meshing settings.
2. Run the nonlinear study.

   The solver maps internally the nodal and element connectivity information stored in the `*.POE` file to the new mesh created in Simulation.

   > You do not have to add fixtures to run the analysis. The nonlinear solver internally applies soft springs to stabilize the part and resolve stiffness singularities.

## Viewing Results after Cooling

After the analysis completes, you can plot the final residual stresses and displacements of the part after it is completely cooled.

The final shape of the part (at pseudo time t = 1 sec) considers the effect of both the in-mold stresses (at pseudo time t = 0) and quenching thermal stresses.

To view the final shape of the part:

1. Under **Results,** open the **Displacement Plot** PropertyManager.
2. Set **Time** to `1` sec and select **Automatic** under **Deformed Shape**.

3. In the **Settings** tab, select **Superimpose model on the deformed shape**.
4. Click **OK**.
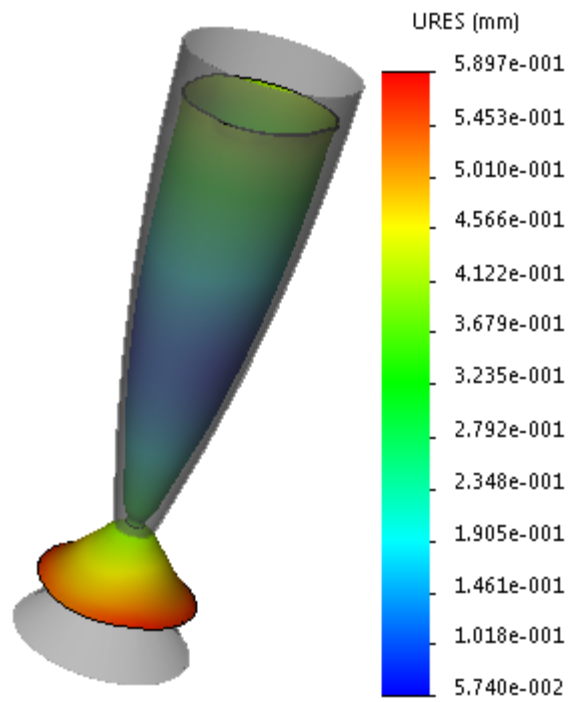
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Sketching

This chapter includes the following topics:

- **Automatic Tangency for Conics**
- **Maintaining Proportions in a Sketch Profile**
- **Path Length Dimension**
- **Replace Sketch Entities**
- **Sketch Equations**
- **Sketch Picture Scale Tool**
- **Splines**
- **Using Stick Fonts**

Automatic Tangency for Conics

Auto Tangency accelerates the process of sketching a conic curve.

When you select this option in the Conic PropertyManager, you set the first and second endpoints for the conic curve. The Conic tool applies tangent relations at each endpoint of the curve and selects the top vertex of the curve. This option is helpful when you reference existing geometry.

If you clear this option, you must define the top vertex of the curve.

Maintaining Proportions in a Sketch Profile

When you specify the first dimension in a sketch for parts or assemblies, all entities in the sketch profile automatically scale uniformly based on the dimension.

This functionality helps maintain the proportions of a sketch profile and only applies when you add the very first dimension to a sketch. Angular and scalar dimensions are not supported.

Path Length Dimension

You can set a path length dimension to a chain of sketch entities. You can set the dimension to be a driving dimension, so that when you drag the entities, the path length resizes consistently.

The path length dimension can help constrain belt and chain assemblies or pulley systems.
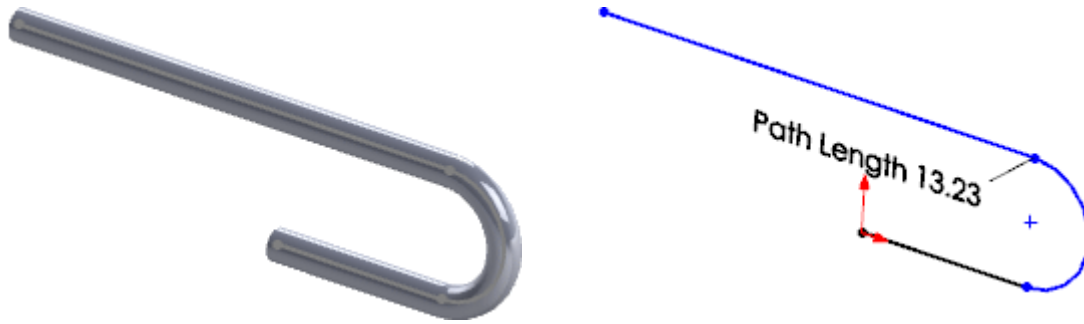
You can reference this dimension in custom properties, configurations, equations, or tables, including design tables and BOMs.

Click **Tools** > **Dimensions** > **Path Length** or right-click the sketch and click **Dimensions** > **Path Length** .

Replace Sketch Entities ★

The **Replace Sketch Entity** command lets you replace one sketch entity with another without breaking downstream references (unless the downstream reference requires a different type of geometry, such as mates). For example, if you replace a line with a spline, downstream features that reference faces created by that spline, will rebuild successfully.

When you delete one sketch entity and replace it, you often need to create a new set of references. If you use the **Replace Sketch Entity** command instead, the new entity inherits references from the old one and you don't need to recreate them. During the replacement, you can also change inherited references and add new ones.

You can replace only one entity at a time.

To access the **Replace Sketch Entity** command:

- Right-click a sketch entity and click **Replace Entity**.
- Right-click a sketch entity and click **Delete**. In the Sketcher Confirm Delete dialog box, click **Replace Entity**.

- Click **Tools** > **Sketch Tools** > **Replace** .

Example for Replace Sketch Entity

In this example, you replace an arc with a spline.
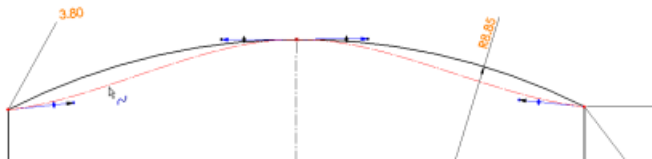
To replace a sketch entity:

1. Open *install_dir*\samples\whatsnew\sketching\replace_entity.sldprt.
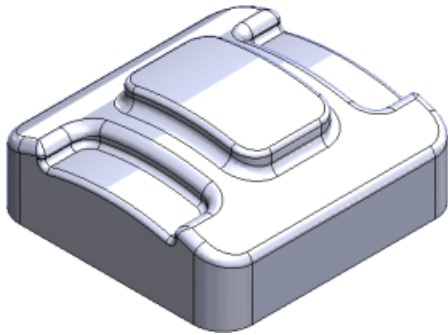
2. In the FeatureManager design tree, select **Boss-Extrude2** and right-click **Edit Sketch** .

3. In the graphics area, use the rotate and zoom tools to view the sketch as needed and select the spline near the top edge of the sketch.

4. In the pop-up toolbar, click **Construction Geometry** .

   This command converts Spline1 from construction geometry to a solid curve.

5. Select the arc and in the pop-up toolbar, click **Replace Entity** .
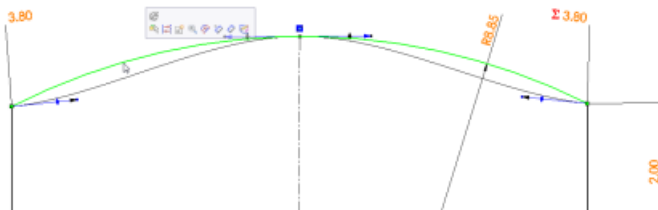
   The Replace PropertyManager lists **Arc2** as the entity to replace.

6. In the PropertyManager, click **Make Construction**.

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7. In the graphics area, select **Spline1**.
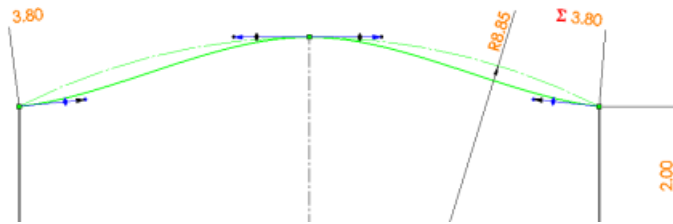
8. Right-click ✔ and close the sketch.

The model automatically rebuilds.

Spline1, the new sketch entity, maintains all the dimensions and relations that had been defined for Arc2.
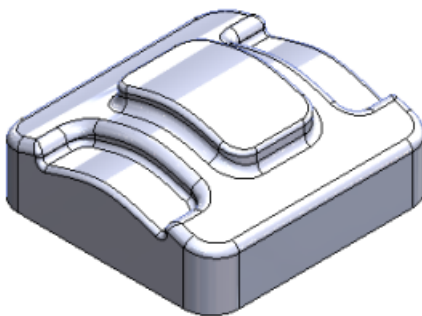
Sketch Equations

You can specify sketch dimensions using equations instead of discrete values. The equations solve automatically as you drag the sketch geometry.

To use equations to specify dimensions, double-click a dimension in a sketch and in the Modify dialog box, start the equation by typing = (equal sign). You can also enter a **Global Variable**, **Function**, or **File Property** from the flyout menu.

Sketch equations are listed in the Sketch View of the Equations, Global Variables, and Dimensions dialog box. Click **Tools** > **Equations** to display the dialog box.

You can also edit sketch equations in the Equations dialog box to create custom constraints.

Sketch Picture Scale Tool

The **Sketch Picture** tool provides a scale tool that lets you easily and accurately position bitmap images in sketch planes and scale them to fit.

Click **Tools** > **Sketch Tools** > **Sketch Picture** .

Splines

Setting a Fixed Length ⭐

You can set a dimension to fix the length of a spline. You can reference this dimension in custom properties, configurations, equations, and tables, including design tables and BOMs.

Previously, you could not dimension splines to a fixed length.

To set a fixed length:

1. Open *install_dir*\samples\whatsnew\sketching\fixed_length.sldprt.

2. Click **Smart Dimension** ⟋ (Dimensions/Relations toolbar) or **Tools** > **Dimensions** > **Smart Dimensions**.

3. Click anywhere on the spline to place the dimension and click ✔ to close the Dimension PropertyManager.

   A radial style dimension appears and is attached to the spline.


   LENGTH 20.89
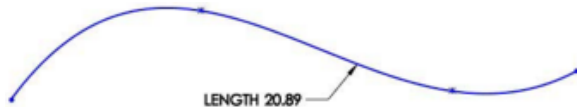
4. Drag an endpoint of the spline.
   The length remains the same when you drag endpoints or change the shape of the spline.

5. To change the spline's length, double-click the dimension and enter a new value in the Modify dialog box.
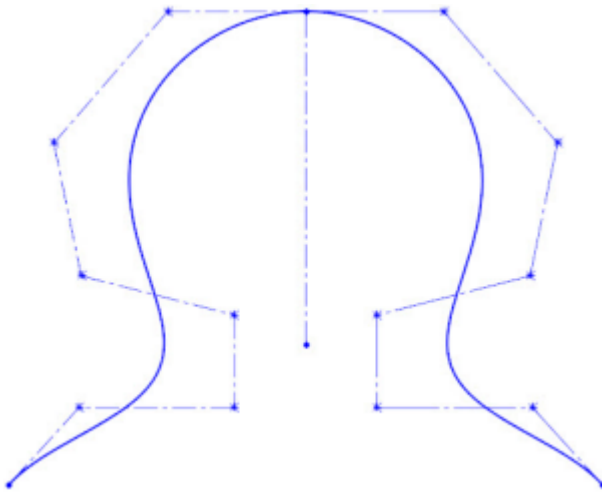
Style Splines ⭐

Use the **Style Spline** 〰 tool to sketch single-span Bézier curves. These curves let you create smooth and robust surfaces, and are available in 2D and 3D sketches.

The style spline consists of only one span. You shape its curvature by selecting and dragging control vertex points. Sketch entities connect the control vertices which form the control polygon of the curve.
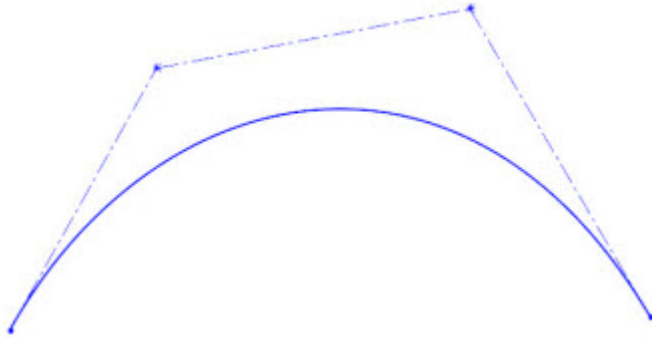
With style splines, you can easily control the degree and continuity of the curves. You can inference style splines for tangency or equal curvature. You can constrain the points and dimension the curves' sides. These curves also support mirroring and self-symmetry.

You can access the **Style Spline** tool from **Style Spline** ![icon] (Sketch toolbar) or **Tools** > **Sketch Entities** > **Style Spline**.

Sketching a Style Spline

You can sketch a style spline as a bridge curve between two existing entities.

To sketch the style spline:

1. Open a new sketch. Use the **Centerpoint Arc** ![icon] tool to sketch two arc entities.

2. Click **Style Spline** ![icon] (Sketch toolbar) or **Tools** > **Sketch Entities** > **Style Spline**.
3. In the graphics area, click the first endpoint of the arc.

   The first click creates the first control vertex point in the style spline.

4. Hover the pointer over the inference line and click to add a second control vertex point as shown.

   If you snap the second control vertex to the inference line, a tangent relation is created at that endpoint.

5. Continue moving the pointer to the right and hover over the next inference line. Click the pointer when the equal curvature icon appears.

   If you snap the third control vertex to the inference line, an equal curvature relation is created at that endpoint.
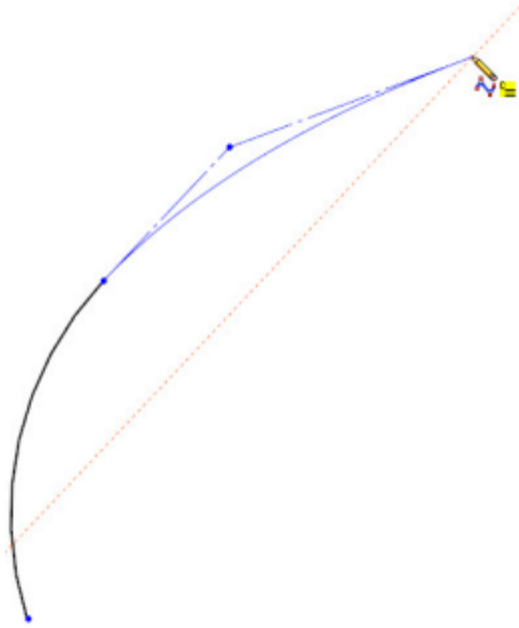
6. To complete the style spline, continue adding more control vertices. When you reach the endpoint of the second arc, press **ALT** + double-click the pointer at the endpoint.

   Pressing **ALT** applies automatic tangent relations at the last control vertex point.
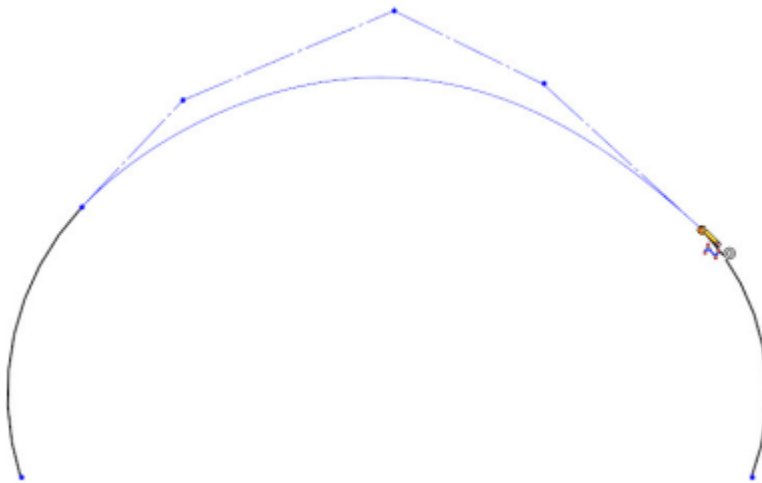
   The bridge curve between the two sketch entities is complete.
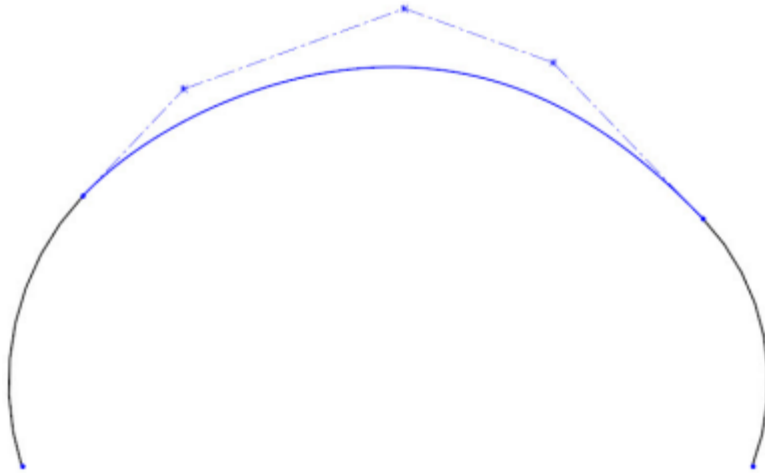
Inserting a Control Vertex

The **Insert Control Vertex** command adds one or more control vertices to style splines. Each control vertex you add increases the curve's degree.

Control vertices are similar to spline points. They help you manage the shape of the curve. You can add dimensions between control vertices and add relations.

**To insert a control vertex:**

1. In an open sketch, right click anywhere on the control polygon and select **Insert Control Vertex** .

    The pointer changes to .

2. Hover the pointer over the control polygon segment where you want to place the control vertex and click to insert the point.
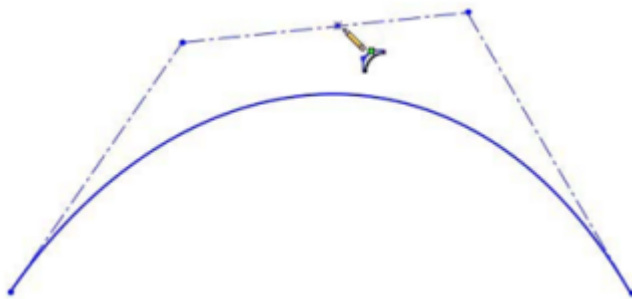
    The control polygon splits the segment at the new control vertex and the curvature adjusts accordingly.
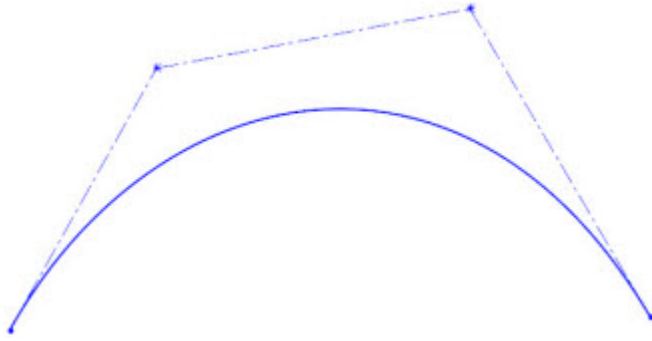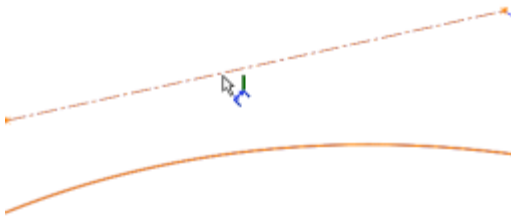
Constraining a Style Spline

You can constrain a style spline by dimensioning line segments in the control polygon.
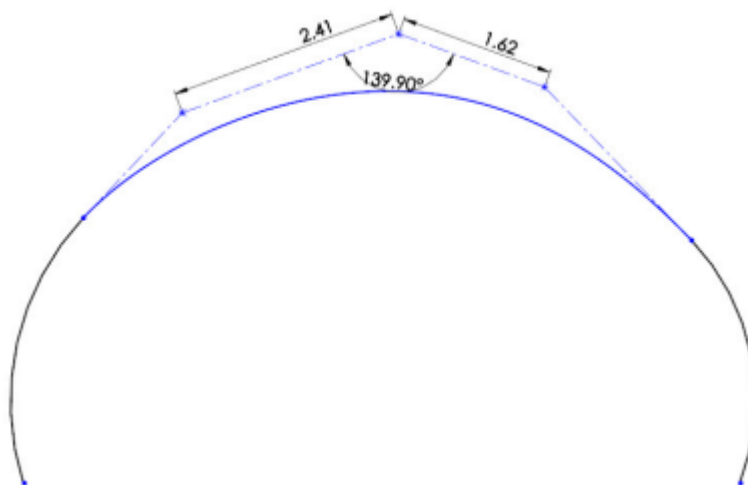
To constrain line segments in a control polygon:

1. In an open sketch, click **Smart Dimension** (Dimensions/Relations toolbar) or **Tools** > **Dimensions** > **Smart**.
2. Click a line segment in the control polygon.

3. Move the pointer to show the dimension preview.

4. Set the value in the **Modify** dialog box and click ✔ .

You can continue to place more dimensions, such as angular dimensions.

Using Local Edit

Local edit allows you to drag and manipulate the shape of a style spline without affecting adjacent style splines. This setting is helpful when you are working with multiple style splines that are connected but not fully constrained.

To modify a style spline using Local edit:

1. In an open sketch, select the style spline you wish to edit locally only.
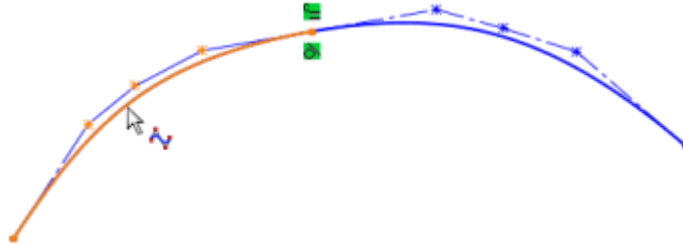
> **Local edit** only works with two or more style splines.

The Style Spline PropertyManager displays.

2. In the Style Spline PropertyManager, under **Options**, select **Local edit**.
3. In the graphics area, select and drag a control vertex.

    You can manipulate the shape of the control polygon without affecting the shape of the other style spline that it is attached to.

4. Click ✔ .

Using Stick Fonts

You can use stick fonts when applying sketch text to parts or assemblies for laser engraving, water jet, and for Computer Numerical Control (CNC) machining.

Stick fonts are also called single line, stroke, or open loop fonts. Use the OLFSimpleSansOC Regular font for all stick fonts.

To use stick fonts:

1. Click **Tools** > **Sketch Entities** > **Text**.
2. In the PropertyManager, under **Text**, clear **Use document font**.
3. Click **Font**.

4. In the Choose Font dialog box, under **Font**, select the **OLF SimpleSansOC** font.

5. Select any other options, and click **OK**.
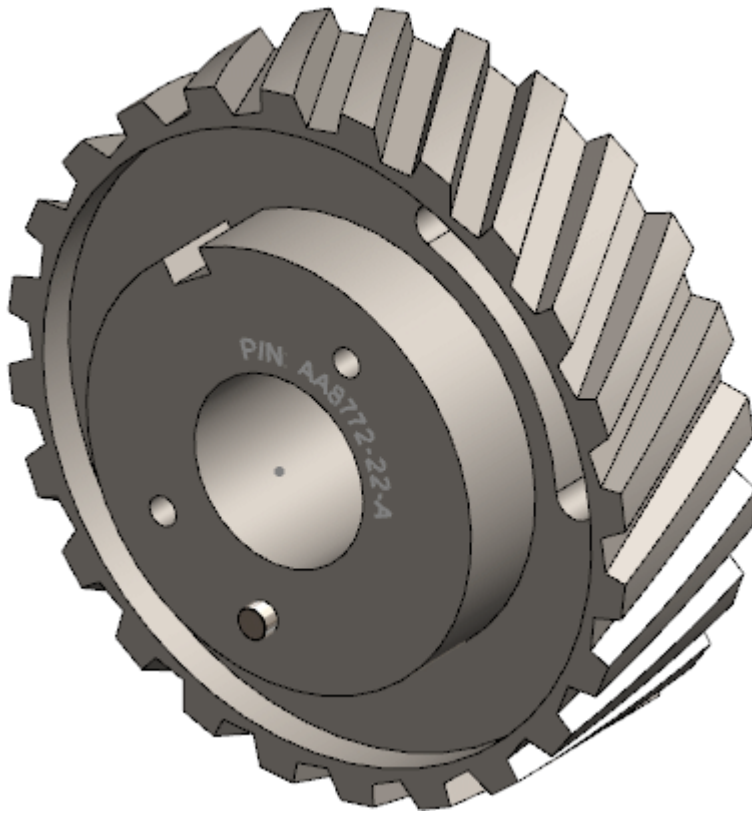
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