

# What's New

CAMWORKS Solids 2011 / SOLIDWORKS 2011

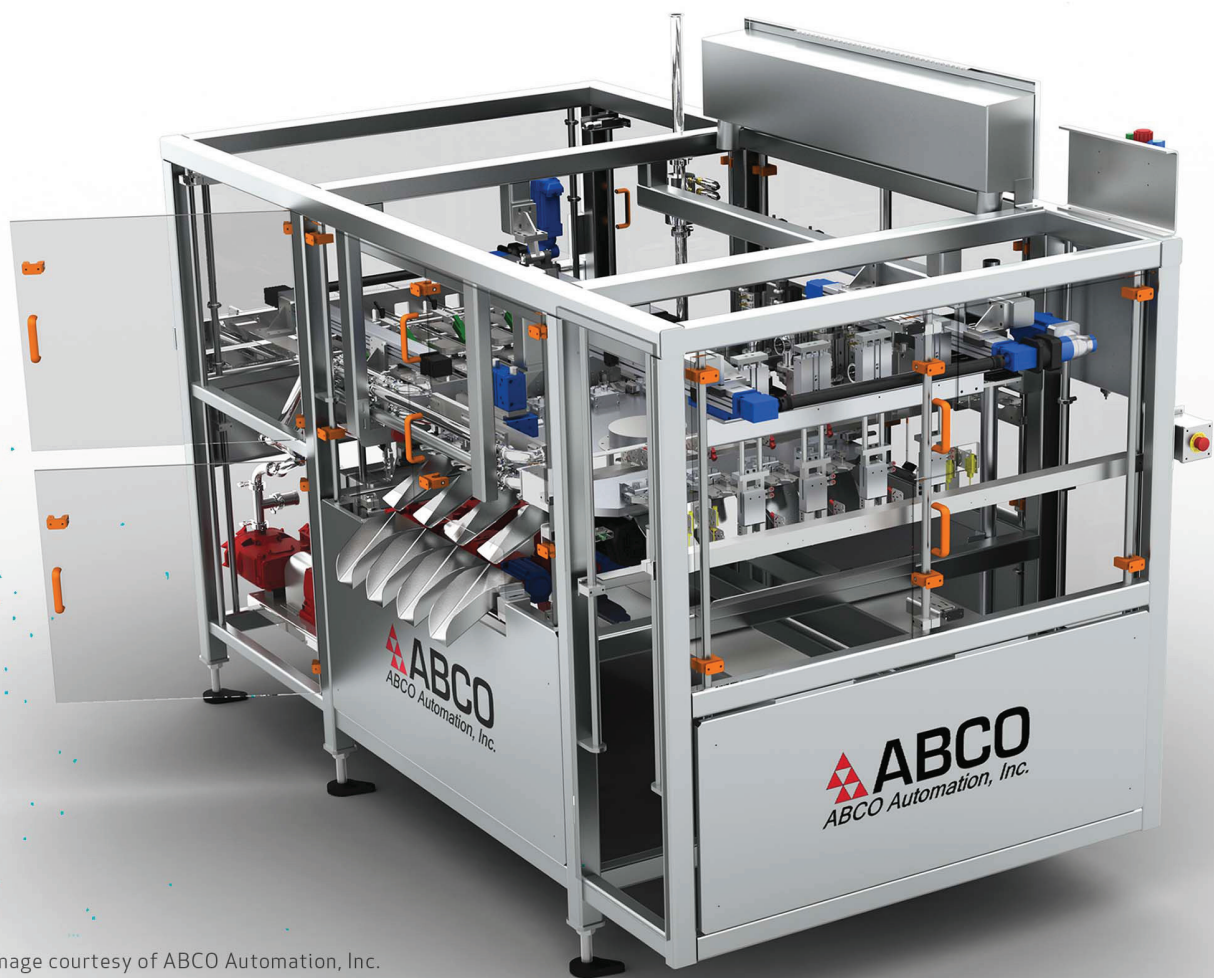


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LET'S GO  
DESIGN

**3D**  
SolidWorks

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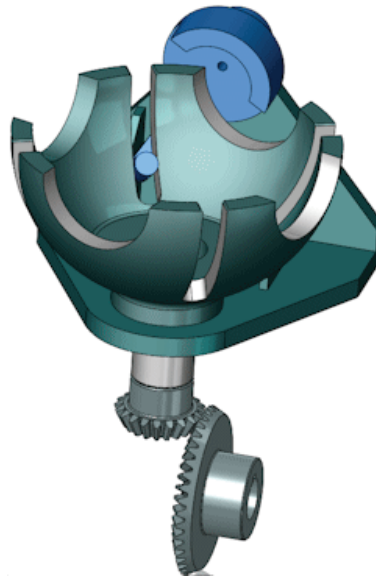


# What's New: Highlights of SolidWorks 2011


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SolidWorks® 2011 includes many enhancements and improvements, most in direct response to customer requests. This release focuses on the following themes:

- Design faster and more efficiently
- Improved collaboration and visualization
- Enhanced support for manufacturing



## Top Enhancements

The top enhancements for SolidWorks 2011 provide improvements to existing products and innovative new functionality. Throughout this guide, look for the  symbol in these areas:

### Assemblies

**Fillets and Chamfers** on page 23

**Weld Bead Display** on page 165

**Defeature for Assemblies** on page 25

### Drawings and Detailing

**Alignment Options for Dimension Palette** on page 43

**Auto Arrange Dimensions** on page 49

**Dual Unit Support** on page 56

**Weld Support in Drawings** on page 169

### Enterprise PDM

**Enterprise PDM Menus** on page 65

### Model Display

**DisplayManager** on page 86

**PhotoView 360** on page 90

### Parts and Features

**Defeature for Parts** on page 101


**Sharing Equations Among Models** on page 101

**Suppression States of Features and Components** on page 103

<b>Simulation</b>	<b>Surface Extrudes From a 2D or 3D Face</b> on page 109
	<b>New 2D Simplification Study (Professional)</b> on page 130
<b>Weldments</b>	<b>Weld Beads</b> on page 164
	<b>Weld Bead Display</b> on page 165
	<b>Fillets and Chamfers</b> on page 166

## For More Information

Use the following resources to learn about SolidWorks:

<b>What's New in PDF and HTML</b>	This guide is available in PDF and HTML formats. Click:
	<ul style="list-style-type: none"> <li>• <b>Help &gt; What's New &gt; PDF</b></li> <li>• <b>Help &gt; What's New &gt; HTML</b></li> </ul>
<b>Interactive What's New</b>	<p>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.</p> <p>To enable Interactive What's New, click <b>Help &gt; What's New &gt; Interactive</b>.</p>
<b>What's New Examples</b>	<p>What's New Examples are updated at every major release to provide examples of how to use most top enhancements in the release.</p> <p>To open What 's New Examples click <b>Help &gt; What's New &gt; What's New Examples</b>.</p>
<b>Online Help</b>	Contains complete coverage of our products, including details about the user interface, samples, and examples.
<b>Release Notes</b>	Provides information about late changes to our products.

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# User Interface


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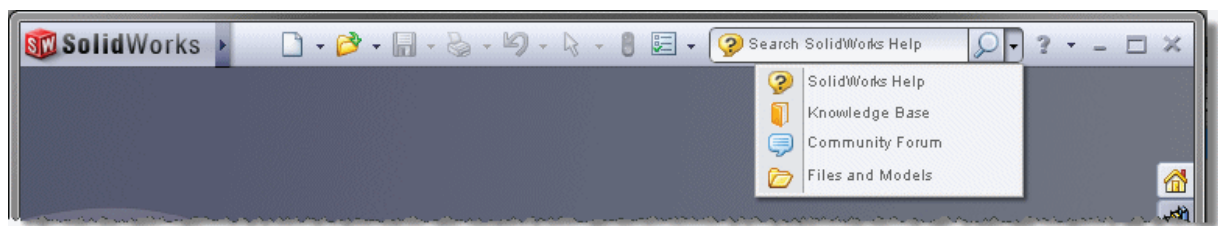
This chapter includes the following topics:

- **SolidWorks Search**
- **Save As and File Properties Buttons on Standard Toolbar**
- **PropertyManager Icons**
- **Error Reporting**

## SolidWorks Search



New search modes have been added to SolidWorks® Search. In addition to searching for files and models, you can search SolidWorks Help, the Knowledge Base, or the community forums.

On the Menu Bar, in the SolidWorks **Search** box , select the location to search and type the text to search for.



See *SolidWorks Help: Search*.

## Save As and File Properties Buttons on Standard Toolbar

The Standard toolbar now has two new buttons, Save As  and File Properties . Use **Tools > Customize > Commands** to add the buttons to a toolbar or the CommandManager.

See *SolidWorks Help: Customize Tool Buttons*.

## PropertyManager Icons

Icons for labels and buttons in PropertyManagers and dialog boxes are updated for greater consistency with toolbar and FeatureManager design tree icons.

## Error Reporting

The SolidWorks Error Report dialog box now allows SolidWorks® to gather more information about why the software stopped responding.

The dialog box lets you do the following:

- Send a failure report to SolidWorks.
- Include a short description of your actions when the application stopped responding.
- Indicate whether this is a new problem or is similar to a previous problem. This information helps SolidWorks Engineering identify stability trends.
- Preview the contents of the report to send to SolidWorks.
- View the SolidWorks data privacy policy. The reports are confidential, subject to this policy.

To send reports to SolidWorks for every failure, select **Enable performance feedback** in **Tools > Options > General**.

# SolidWorks Fundamentals

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This chapter includes the following topics:

- **Application Programming Interface**
- **Pack and Go**
- **Documentation**

## Application Programming Interface

Major enhancements are new interfaces, methods, properties, and delegates.

You can now:

- Display .NET controls in the SolidWorks user interface
- Access Pack and Go
- Get data objects containing curve, spline, and surface parameter properties
- Manage line styles in drawings
- Get the contents of folders in the FeatureManager design tree
- Create flyout groups in the CommandManager
- Convert draft quality drawing views to high quality
- Access a ray-trace rendering engine, such as PhotoView 360, and its options
- Determine whether the current sketch is a boundary box
- Get the direction of a bend line
- Get the boundary-box sketch display data of a flat-pattern drawing view
- Add and delete material from specific display states in the active configuration of a model
- Add a standard SolidWorks or custom button to the Task Pane
- Get the name of the active PropertyManager page
- Get custom and stock render references for a model
- Get whether a command or PropertyManager page is active
- Add a line to a C# or VB.NET macro and to the SolidWorks journal file
- Send notification:
  - Before a command or PropertyManager page executes or opens
  - When a BOM or general table is inserted in an assembly, drawing, or part document
  - After an entity is selected in a part, assembly, or drawing document
- Use Design Checker to:
  - Build checks from existing SolidWorks documents, templates, and drawing standards
  - Check against an existing file
  - Get summary results and save its report in Microsoft Word format

Click **Help > API Help > SolidWorks API Help > SolidWorks APIs > Release Notes**.



## Pack and Go

New options on the Pack and Go dialog box let you save system-provided and custom decals, appearances, and scenes associated with the model.

The options are:

- **Include default decals, appearances and scenes**
- **Include custom decals, appearances and scenes**




As an alternative to Pack and Go, go to **Tools > Options > Document Properties > Model Display** and select **Store Appearance, Decal and Scene data in model file**.

## Documentation

### New Tutorials

SolidWorks® 2011 includes the following tutorials: Assembly Visualization, Event-based Motion Studies, Mouse Gestures, and SustainabilityXpress.

To access these tutorials:

1. Click **Help**  > **SolidWorks Tutorials**.
2. Click one of the following:
  - **All SolidWorks Tutorials (Set 1)**
  - **All SolidWorks Tutorials (Set 2)**
3. Select a new tutorial from the list.

### Toolbox Administration Overview

SolidWorks Help contains new Toolbox administration information.

See *SolidWorks Help: Toolbox Administration Overview*.

# 3D ContentCentral

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This chapter includes the following topics:

- [Defeature Tool](#)
- [Configuration Publisher](#)
- [3D ContentCentral for Suppliers](#)

## Defeature Tool

Use the SolidWorks® **Defeature** tool to remove details from a part or assembly and save the results to a new file in which the details are replaced by dumb solids (that is, solids without feature definition or history). You can then share the new file without revealing all the design details of the model. For more information, see [Defeature for Parts](#) on page 101.

## Configuration Publisher

### Multiple Parent/Child Control

The Configuration Publisher offers more flexibility when defining data parents for controls.

You can define parents explicitly, and multiple parents are allowed. You no longer automatically inherit grandparents (that is, the parents of controls which you have specified as parents). See [Parent/Child Relationships](#) on page 37.

### Model Tests

You can test the health of models before uploading them to 3D ContentCentral®.

A sample of possible configurations is tested. Results are reported in a log file that lists the configurations that were tested and whether they rebuilt successfully. See [Model Tests](#) on page 37.

## 3D ContentCentral for Suppliers

Supplier Services introduces new self-service publishing capabilities for industrial component suppliers.

### Part Numbers

Suppliers can use the Configuration Publisher to generate part numbers for a configurable model, and users can search for models by part number on 3D ContentCentral. See [Part Numbers](#) on page 36.

## **Model Health**

Suppliers can test configurable models for Model Health in SolidWorks before uploading them to 3D ContentCentral. See [Model Tests](#) on page 37.

## **Defeature Tool**

Suppliers can enable the rules they have defined in SolidWorks to remove details from configurable models prior to download. See [Defeature for Parts](#) on page 101.

This chapter includes the following topics:

- **Converting Files to SolidWorks 2011**
- **Installation Improvements**
- **SolidWorks Rx**

## Converting Files to SolidWorks 2011

Opening a SolidWorks® document from an earlier release can take extra time. After you open and save a file, subsequent opening time returns to normal.

You can use the SolidWorks Task Scheduler (SolidWorks Professional) to convert multiple files from an earlier version to the SolidWorks 2011 format. Click **Windows Start > All Programs > SolidWorks 2011 > SolidWorks Tools > SolidWorks Task Scheduler**.

In the Task Scheduler:

- Click **Convert Files** and specify the files or folders to convert.
- For files in a SolidWorks Workgroup PDM vault, use **Convert Workgroup PDM Files**.

For files in a SolidWorks Enterprise PDM vault, use the utility provided with Enterprise PDM.



After you convert files to SolidWorks 2011, you cannot open them in older versions of SolidWorks.

## Installation Improvements

SolidWorks Installation Manager and the installation process have significant improvements.

### License Activation

SolidWorks License Activation is enhanced.

- You can activate and transfer licenses for multiple serial numbers in a single step.
- You can transfer a license from a computer even if SolidWorks is no longer installed on that computer. You can download and run SolidWorks Activation Wizard from the SolidWorks customer portal to transfer the license.

### SolidNetWork License Manager

SolidNetWork License Manager is enhanced.

- The display of borrowed licenses is improved.

- The list of product licenses available to borrow now shows only what has been purchased and is available on the SolidNetWork License Manager.
- Borrowing a SolidWorks package license now automatically borrows the pre-requisite SolidWorks Standard license.
- You can update SolidNetWork License Manager to a new version using SolidWorks Installation Manager, rather than removing the previous version and then installing the new one.

## Installation Manager

SolidWorks Installation Manager is enhanced.

- You can specify which languages to include in the SolidWorks product installation, rather than installing all languages. English is always installed, even if you do not specify it during installation.



The **Languages** specification in the **Product Summary** applies only to the SolidWorks product installation. It does not apply to other product installations such as SolidWorks eDrawings®, SolidWorks Workgroup PDM, and SolidWorks Explorer, which install all languages.

- The Workgroup PDM and SolidNetWork License Manager installations are more integrated into the SolidWorks Installation Manager workflow.
- You can create 32-bit and 64-bit administrative images on both 32-bit and 64-bit operating systems. For example, you can:
  - Create a 32-bit image on a 64-bit operating system
  - Create a 64-bit image on a 32-bit operating system
  - Create 32-bit and 64-bit images on the same operating system

## Administrative Image Option Editor

The interface and workflow for SolidWorks Administrative Image Option Editor are enhanced.

- The Option Editor opens automatically after SolidWorks Installation Manager creates the administrative image.
- You can manage both 32-bit and 64-bit administrative images using the same Option Editor on either a 32-bit or a 64-bit operating system.
- You can perform all administrative image configuration steps using SolidWorks Administrative Image Option Editor (some of which were handled previously in SolidWorks Installation Manager).
- In SolidWorks Administrative Image Option Editor, you can specify whether automatic updating is enabled for one or more machines or groups. You also can specify when the update occurs for specific machines or group of machines. This allows you to manage automatic updates (for example, avoiding the overload of a particular server because all the updates take place simultaneously).

## Background Downloader

You can use SolidWorks Background Downloader to download installation files in the background. You are notified when downloading is complete and the files are ready for installation. You can choose to install now, delay the installation, or delete the download.





## SolidWorks Rx

### Diagnostics


The improved page design and content on the Diagnostics tab helps you locate and download the correct graphics driver at

<http://www.solidworks.com/sw/support/videocardtesting.html>.

Status shown in the Diagnostics tab for different system and graphics card combinations on your machine:

Machine		Status	Fields Displayed
<b>System</b>	Certified		All system and graphics card related fields
<b>Graphics Card</b>	Certified only for previous version of SolidWorks		
<b>System</b>	Not certified		Graphics card related information
<b>Graphics Card</b>	Certified only for previous version of SolidWorks		
<b>System</b>	Certified		All system and graphics card related fields
<b>Graphics Card</b>	Out-of-date		
<b>System</b>	Certified		All system and graphics card related fields
<b>Graphics Card</b>	Current		



If you have no internet connection, the status shows the error icon .

### Usability and Workflow Improvements

The Problem Capture tab helps you describe the problem, record the video, and package the SolidWorks files.

1. On the Problem Capture tab, click **Describe Problem**.
2. In the Problem Capture Details dialog box, describe the problem and click **OK**  
This dialog box has been shortened to improve usability.
3. Click **Record Video**.  
A new SolidWorks session starts. This button replaces the Re-create the problem dialog box.
4. In the Capturing Problem dialog box, click **Start Recording**.

The floating dialog box stays on top of your SolidWorks session.

5. Record the problem and click **Finish Recording**.
6. When prompted to shut down the SolidWorks session, click **OK**.
7. Click **Add Files** to add parts, drawings, or assembly files, then click **Package files Now**.  
A zipped folder is created. You can rename the folder, save it on your hard disk, and send it to the Technical Support team.

## Hardware Benchmarks

The Addins tab now has a link to [www.solidworks.com/benchmarks](http://www.solidworks.com/benchmarks) for you to find details about running SolidWorks and computer-related benchmarks. You can determine whether your system is qualified to run SolidWorks effectively.

To run the SolidWorks performance benchmark, on the Addins tab, click **Start the SolidWorks Performance Benchmark**.

## Graphics Card Health Checker

Enhancements to the Graphics Card Drivers page help you identify the correct graphics driver. See <http://www.solidworks.com/sw/support/videocardtesting.html>.

Improvements:

- A one-click application that profiles your system and graphics card, and populates the information on the web page for you to identify the correct graphics card.



You need .NET 2.0 or later installed on your machine to view this button on the web page.

- An updated results chart with color-coded keys that improves readability.



# 5

## Assemblies

---

This chapter includes the following topics:

- [Assembly Features](#)
- [Assembly Visualization](#)
- [Defeature for Assemblies](#)
- [Equations](#)
- [Interference Detection](#)
- [Mates](#)
- [Rebuild Report](#)
- [SpeedPak](#)

### Assembly Features

Enhancements include the ability to add fillets, chamfers, and weld beads to assemblies.

#### **Fillets and Chamfers** ★

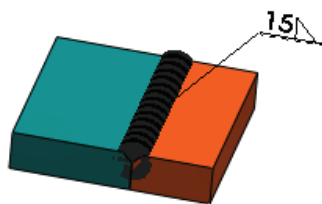
In assemblies, you can create fillets and chamfers, which are useful for weld preparation. As with other assembly features, you can propagate these features to the parts they affect.




For a step-by-step example of adding chamfers in an assembly, see [Fillets and Chamfers](#) on page 166.

#### **Weld Beads** ★

You can add simplified weld beads to assemblies. Simplified weld beads provide a lightweight, simple representation of weld beads.



 In previous versions of SolidWorks®, you added weld beads as components of the assembly. This method is no longer supported. However, you can still edit existing weld bead components.

For more information about simplified weld beads, see [Weld Beads](#) on page 164.

For a step-by-step example of adding a weld bead to an assembly, see [Weld Beads](#) on page 165.

## Assembly Visualization

Enhancements support complex sorting scenarios and provide additional display and save options.


### Sorting and Column Options

You can build more complex sorting scenarios by doing the following:

- Adding multiple columns
- Adding multiple parameters to the sorting hierarchy
- Rearranging column position to change the order in which the parameters are sorted

Right-click a column header and select the following:

- **Add Column.** Adds a column to the Assembly Visualization list. You can then change which property is listed in the new column.
- **Delete Current.** Deletes the column you right-clicked. The minimum number of columns is three. The **File Name** and **Quantity** columns cannot be deleted.
- **Add to Sort Hierarchy.** Enables you to sort by the values in that column. A sorting widget ▼ appears under the column title. To remove a column from the sort hierarchy, right-click it and click **Remove from Sort Hierarchy**. When multiple columns have sorting widgets, the hierarchy of the sort is from left to right. Drag a column header to move a column left or right to move it up or down in the sorting hierarchy.

 You can adjust the width of a column to fit the column contents by double-clicking the column separator.

### Save and Display Options

With enhanced save and display capabilities, you can:

- Export the Assembly Visualization list in top-level only, parts only, or indented format when saving the list to a separate file. Right-click any column header and click **Save as**. Then in the Save As dialog box, select **Top-level only**, **Parts only**, or **Indented**.
- Save the Assembly Visualization view as a display state. Click the arrow ▶ to the right of the column headers and click **Add Display State**.

- Apply the same color to components that have identical values (for example, when components are sorted by a discrete property such as **Vendor** or **Status**). Right-click a color slider and click **Group Identical**. The software adds sliders as needed to apply a distinct color for each discrete value.
- Set the length of the value bars to be calculated as a percentage of the value for the highest-value component or as a percentage of the value for the entire assembly. Click the arrow ► to the right of the column headers, click **Value Bars**, and then click **Component Driven** or **Assembly Driven**.

## Defeature for Assemblies ★


With the **Defeature** tool, you can remove details from a part or assembly and save the results to a new file in which the details are replaced by dumb solids (that is, solids without feature definition or history). You can then share the new file without revealing all the design details of the model.



Before



After

Click **Defeature**  (Tools toolbar) or **Tools > Defeature** to access the Defeature PropertyManager, which provides tools for manual and automatic selection of details to keep and remove.

The PropertyManager has multiple pages.

### Defeature - Step 1: Components

You can specify components to remove from the model.

In this example, you use the **Defeature** tool to remove all internal components and features, and some external features, from an oil pump assembly. You retain the mounting holes.


1. Open `install_dir\samples\whatsnew\assemblies\oil_pump.sldasm`.



2. Click **Tools > Defeature**.

The Defeature PropertyManager opens to **Step1: Components**.



First, examine a section view of the assembly.

3. In the PropertyManager, under **Section View**, click **Plane1** . You can see internal details and components.



Change **Offset Distance**  to view different sections of the assembly.



4. Click **Plane1**  to clear the section view.
5. Under **Remove**, select **Internal components**.
6. Click **Next** .

## Defeature - Step 2: Motion


If you want to allow motion in an assembly, you can remove details from groups of components and allow motion between the groups.

For this example, skip this step.

Click **Next** .

## Defeature - Step 3: To Keep

You can specify features that you want to retain, such as mounting holes, that might otherwise be removed by the **Defeature** tool.

1. For **Features to Keep** :
  - a) In the graphics area, select the three mounting holes on the cover.



- b) Rotate the assembly and select the three mounting holes on the housing.



2. Click **Next** .

The screen splits and a preview of the model appears. Features on the housing are removed to close up the assembly. The mounting holes you selected are retained.



When you rotate the assembly, the preview also rotates.




Assembly



Preview

### Defeature - Step 4: To Remove

You can specify to remove features that were not automatically removed.

1. In the flyout FeatureManager design tree, expand **Cover** and select **Rib Cut**. In the PropertyManager, **Rib Cut** and its dependent features appear in **Items to Remove** . In the graphics area, the cut feature is highlighted.




2. Click **Next** .


The preview shows that the cut feature is removed.




## Defeature - Feature Removal Complete

You can save the less-detailed model in a separate part file or publish it to 3D ContentCentral. The settings you select in the PropertyManager are saved in the original model.



1. Under **Results**, select **Save the model as a separate file**.
2. Click .
3. In the Save As dialog box:
  - a) For **File name**, type `oil pump - details removed`.
  - b) For **Save as type**, select **Part (\*.sldprt)**.
  - c) Click **Save**.

The assembly is saved as a part. The part has only one feature, **Imported1** , and has no references to the original assembly.



In the assembly, **Defeature**  appears near the top of the FeatureManager design tree.



You can right-click **Defeature** , click **Edit Feature** , change the settings, and save another version of the assembly, with different details removed.

4. In the part window, click **Section View**  (Heads-Up toolbar) or **View > Display > Section View**.  
The part has no internal details.



## Equations

Equation functionality is enhanced.

- You can use equations to control the suppression state of assembly components.
- You can share equations and global variables among models.
- You can configure global variables.

See [Equations](#) on page 101.

## Interference Detection

**Interference Detection** has been enhanced.

### Ignore Hidden Bodies

You can ignore interferences with hidden bodies.

For example, suppose that in a multibody part, you create a body for construction purposes and then hide it. In an assembly, you can choose to not show interferences between the hidden body and other components.

Click **Interference Detection**  (Assembly toolbar) or **Tools > Interference Detection**. In the PropertyManager, under **Options**, select **Ignore hidden bodies**.

### Quality and Performance

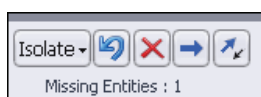
The quality and performance of **Interference Detection** have been improved. Because of the changes, the software might find interferences that were previously missed, resulting in changes in the number of interferences reported.

## Mates

### Replace Mate Entities

Replace Mate Entities is enhanced.

- The Missing Entities popup toolbar is now available while you replace any mate entities, not just while you replace a component that includes mate entities.





- While replacing multiple mate entities, you can postpone the solving of the mates. Select **Defer update** so that all mates are solved together when you exit the PropertyManager, instead of solving individually each time you select a replacement entity.

Select a mate, a component, or a **Mates** folder and click **Replace Mate Entities**  (Assembly toolbar).

## Mates in Motion

To support more accurate motion simulations, certain mates are now solved relative to other components rather than relative to the assembly origin.

The reference components for gear mates, rack and pinion mates, and screw mates are detected automatically. For linear/linear coupler mates, you specify the appropriate reference components in the Mate PropertyManager.



The reference component information is used in SolidWorks Motion simulations.

## Rebuild Report

AssemblyXpert now provides a report of the rebuild time for the total assembly.



You must rebuild the assembly during the current session of SolidWorks before generating the report.


1. Click **AssemblyXpert**  (Assembly toolbar) or **Tools > AssemblyXpert**.
2. In the AssemblyXpert dialog box, in the information-only row about the rebuild report, click **Show these Parts** .

## SpeedPak

New commands make it easier to switch between SpeedPak and parent configurations of subassemblies. Also, sketches now appear in SpeedPak configurations.

### Switching All Subassemblies to SpeedPak

Do one of the following:

- When opening an assembly, click **Open**  (Standard toolbar) or **File > Open**. In the Open dialog box, select **Use SpeedPak**.
- In an assembly that is already open, in the FeatureManager design tree, right-click the top-level assembly and click **Use SpeedPak**.

If a subassembly's active configuration has a SpeedPak configuration, then the SpeedPak configuration is used.

### Switching Selected Subassemblies to SpeedPak

In the FeatureManager design tree, right-click one or more subassemblies and click **Use SpeedPak**.

If the selected subassembly's active configuration has a SpeedPak configuration, then the SpeedPak configuration is used.

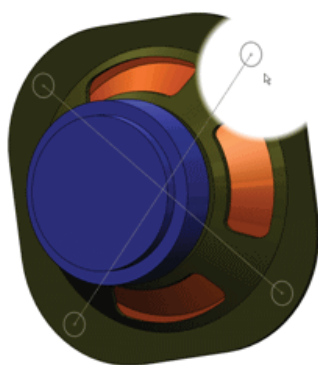
## Switching from SpeedPak to Parent Configurations

In the FeatureManager design tree, right-click the top-level assembly or one or more selected subassemblies and click **Set SpeedPak to Parent**.

The subassemblies switch from the SpeedPak configuration to the SpeedPak's parent configuration.

## Sketches in SpeedPak Configurations

Unabsorbed sketches are included in SpeedPak configurations. You can suppress unneeded sketches to avoid having them affect performance of the SpeedPak configuration. As in any assembly, you suppress and unsuppress sketches by right-clicking them in the FeatureManager design tree and selecting **Suppress** ↓ or **Unsuppress** ↑.



# Configurations

---

This chapter includes the following topics:

- **Configuration Publisher**
- **Modify Configurations**
- **Parameters**
- **SpeedPak**

## Configuration Publisher

### 3D ContentCentral

The Configuration Publisher has new capabilities for 3D ContentCentral suppliers.

#### Part Numbers

A part number can be displayed with your model on 3D ContentCentral. Your customers can search for the model by part number.

To include a part number with the model:

1. In the design table of the model, create a column with the header `$PARTNUMBER`.
2. Under `$PARTNUMBER`:
  - For single-row design tables, use Microsoft Excel functions such as `CONCATENATE` to make the value of the cell reflect the model's part number based on values in other columns.
  - For multiple-row design tables, enter a part number for each configuration by typing or using Microsoft Excel functions.
3. In the Configuration Publisher, add controls, rules, and values.  
On the 3DCC Preview tab, **Supplier part number** displays the part number of the selected configuration. As you change selections, the part number updates to reflect the current selections.
4. Click **Upload to 3D ContentCentral** and click **Yes** to save the model.  
The software generates a list of part numbers and stores it within the model in a format that is accessible to the 3D ContentCentral software.
  - For single-configuration models, a part number is generated for each possible configuration, based on the rules and values you defined in the Configuration Publisher.
  - For multiple-configuration models, the list contains the part numbers you entered for each configuration in the design table.

On 3D ContentCentral:

- On the Configure & Download page, the part number appears at the bottom of the **Configure** area and updates as you change selections.
- You can search for the model by part number or partial part number.

### Model Tests

You can test the health of models before uploading them to 3D ContentCentral.

A sample of possible configurations is tested. Results are reported in a log file that lists the configurations that were tested and whether they rebuilt successfully.

To test a model before uploading it:

1. In the Configuration Publisher, on the 3DCC Preview tab, click **Test**.
2. In the Configuration Tester dialog box, specify the sample size and where to store the log file, and click **Begin Test**.

The dialog box reports the percentage of the samples that succeeded. You can view the log file for more details.


### Parent/Child Relationships

The Configuration Publisher offers more flexibility when defining data parents for controls.

You can define parents explicitly, and multiple parents are allowed. You no longer automatically inherit grandparents (that is, the parents of controls which you have specified as parents).




You must have at least two controls already defined. Then you can add multiple data parents to the third or later controls.

In the Configuration Publisher, under **Data**, one **Parent** field is initially visible. To add others, click **Add Parent** .

## Modify Configurations

The Modify Configurations dialog box has been enhanced.

- The **All Parameters** option has been moved from the table view drop-down list to a separate button .
- Linked dimensions are grouped in a column labeled **Linked Dimension** instead of appearing in columns for individual features. Each linked dimension appears in the list only once, even if it is used in several features.

Right-click an item and select **Configure dimension**, **Configure feature**, **Configure component**, or **Configure Material**.

## Parameters

You can configure parameters in the following areas.

- Scale features (See [Scale](#) on page 109.)
- Global variables (See [Global Variables](#) on page 106.)
- Cosmetic threads (See [Cosmetic Threads](#) on page 53.)

## SpeedPak

New commands make it easier to switch between SpeedPak and parent configurations of subassemblies. Also, sketches now appear in SpeedPak configurations.

For more information, see [SpeedPak](#) on page 30.

## Design Checker Task in Enterprise PDM

You can use the new Design Checker task inside the Administration Tool of Enterprise PDM to validate selected SolidWorks documents in the vault using standards files (.swstd) created in Design Checker.

See [Design Checker Validation](#) on page 71.

This chapter includes the following topics:

- **Injection Molding**
- **Running an Injection Molding Check**

## Injection Molding

DFMXpress supports injection molded parts. You set the **Minimum wall thickness** and **Maximum wall thickness** parameters that DFMXpress uses to validate manufacturability.

## Running an Injection Molding Check

1. With a part open, click **Tools > DFMXpress**.
2. Click **Settings**.
3. Under **Manufacturing Process**, select **Injection Molding**.
4. Under **Rule Parameters**, set **Minimum wall thickness** and **Maximum wall thickness**.
5. Click **Run**.  
The **Manufacturing Process: Injection Molding** report displays a list of rules passed and rules failed.



# Drawings and Detailing

---

This chapter includes the following topics:

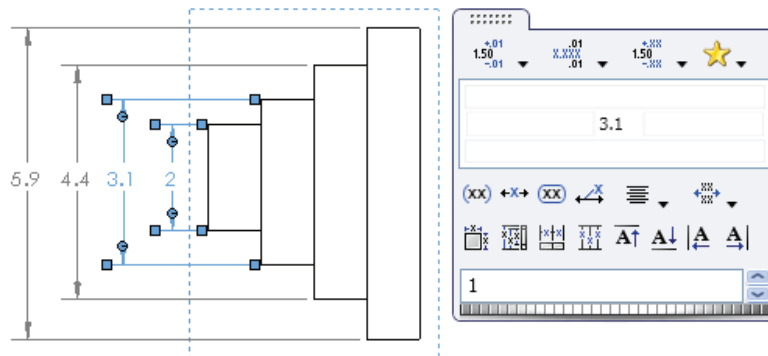
- **Alignment Options for Dimension Palette**
- **Auto Arrange Dimensions**
- **Bounding Box for Sheet Metal**
- **Cut List Properties in Drawings**
- **Center Marks in Assembly Drawings**
- **Cosmetic Threads**
- **Display Scale in Orthogonal Views**
- **Drawing Sheet Format**
- **Dual Dimensions for Chamfers**
- **Hide Bodies in Drawing View**
- **Hole Tables**
- **Merge and Unmerge Cells in Tables**
- **Notes**
- **Show Dimension Units**
- **Show Model Colors in Drawings**
- **GB Drafting Standard**
- **ANSI Drafting Standard**
- **3D Drawing Views**

## Alignment Options for Dimension Palette ★

Alignment tools are available on the dimension palette when you select more than one dimension.

To display the dimension palette, select dimensions and move the pointer over the

**Dimension Palette** rollover button .



The following alignment tools are on the dimension palette.



These tools are also available on the Align toolbar.

### Auto Arrange Dimensions

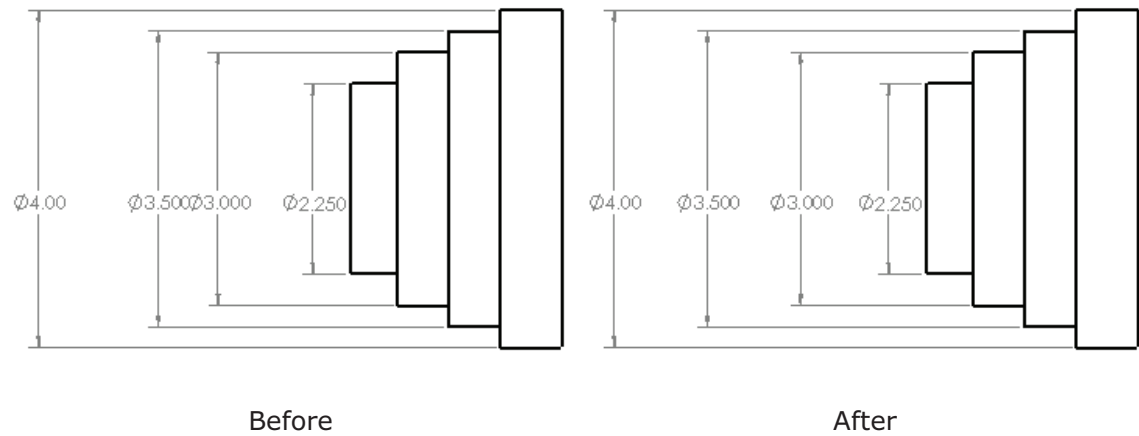


Automatically arranges selected dimensions. See [Auto Arrange Dimensions](#) on page 49.

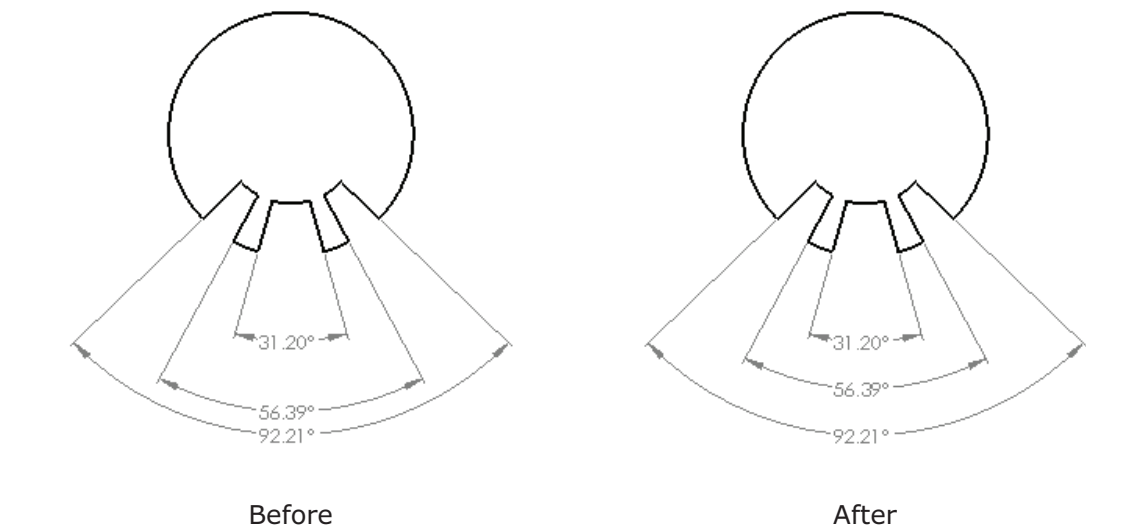
## Space Evenly Linear/Radial

Equally spaces all dimensions, linearly or radially, between the closest and furthest selected dimensions from the part.

Linear

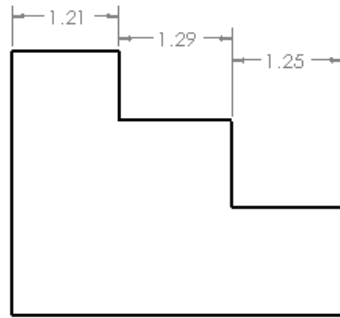


Radial

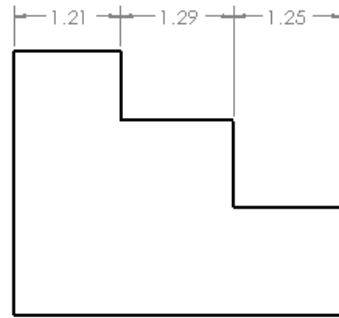


### Align Collinear

Aligns dimensions horizontally, vertically, or radially.



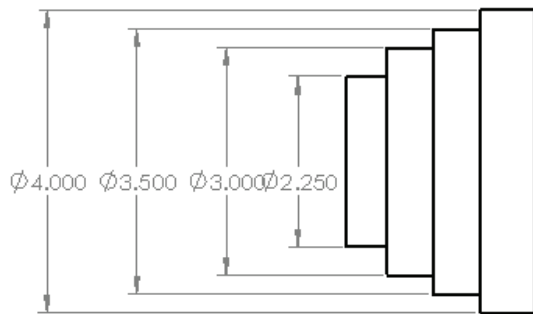
Before



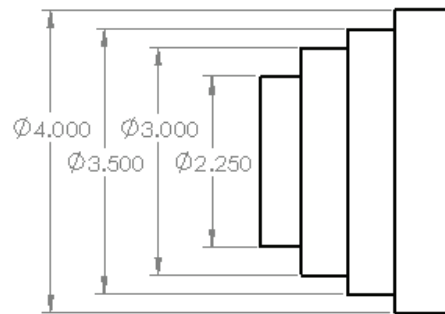
After

### Align Stagger

Stagger linear dimensions.



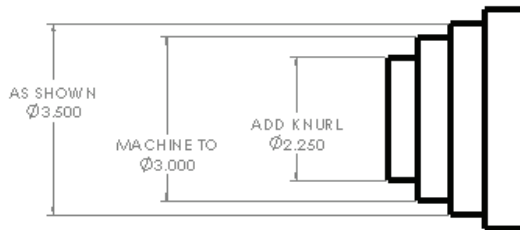
Before



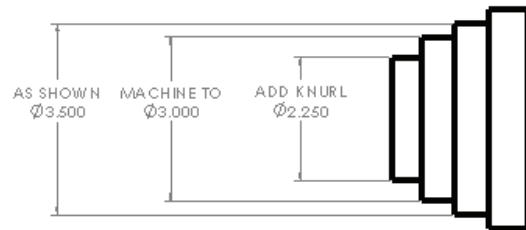
After

### Top Justify Dimension Text

Top justifies linear dimension text.



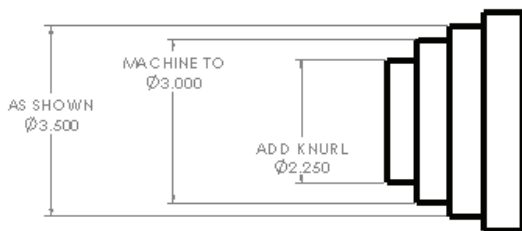
Before



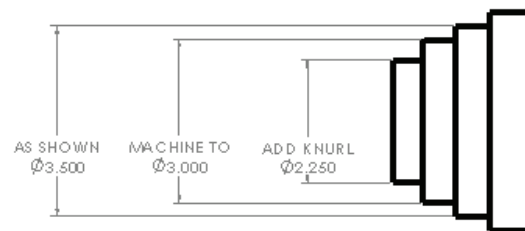
After

### Bottom Justify Dimension Text

Bottom justifies linear dimension text.



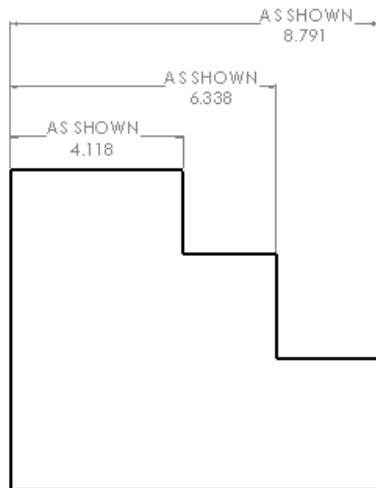
Before



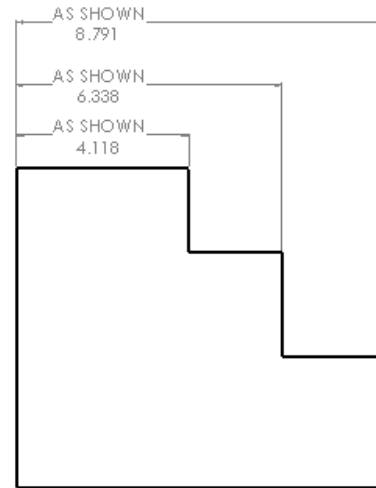
After

## Left Justify Dimension Text

Left justifies linear dimension text.



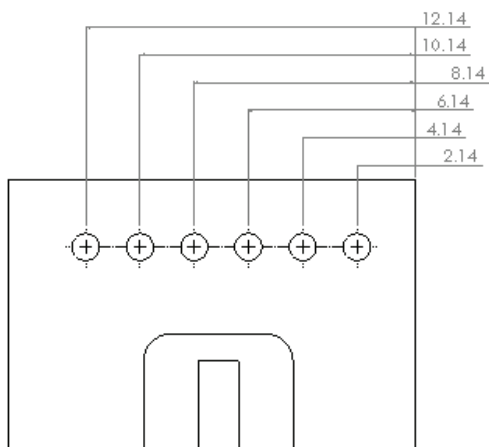
Before



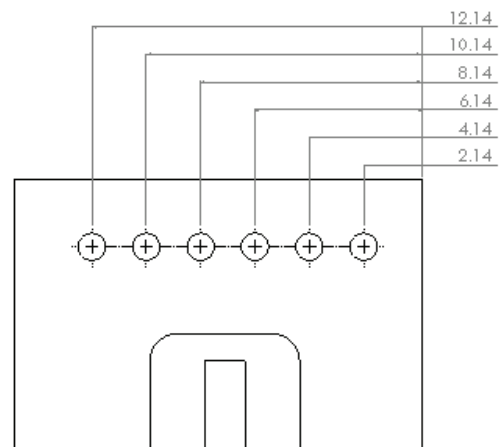
After

## Right Justify Dimension Text

Right justifies linear dimension text.



Before



After

You can adjust the spacing between dimensions using the thumbwheel in the dimension palette or by manually dragging the dimensions.



- The thumbwheel appears in the dimension palette only when spacing options are available.
- The **Dimension Spacing Value** is the scaling factor for the spacing. For example, 2 increases the spacing by a factor of 2, or doubles it.

## Auto Arrange Dimensions ★

The **Auto Arrange Dimensions** tool positions dimensions quickly and easily.

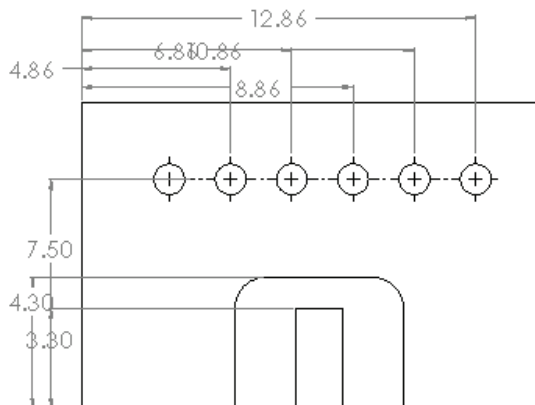
When you use **Auto Arrange Dimensions** , the selected dimensions are placed as follows:



- Spaced from smallest to largest
- Aligned and centered, if possible
- Spaced with the offset distances defined in Document Properties - Dimensions
- Adjusted to avoid overlapping
- Staggered, if necessary

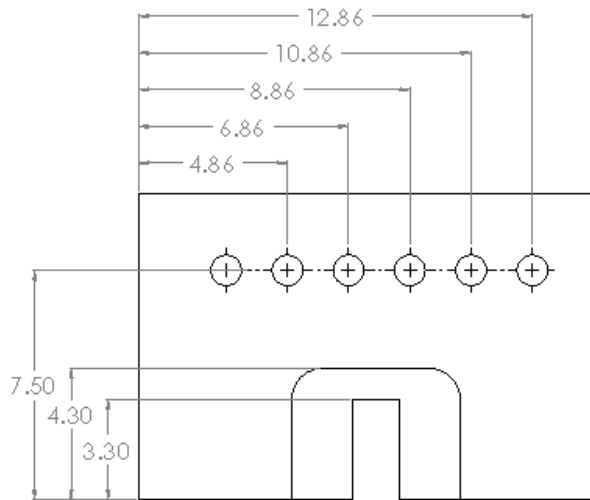
### Using Auto Arrange Dimensions

To use **Auto Arrange Dimensions**:

1. Open `install_dir\samples\whatsnew\drawings\Auto_arrange.SLDDRW`.



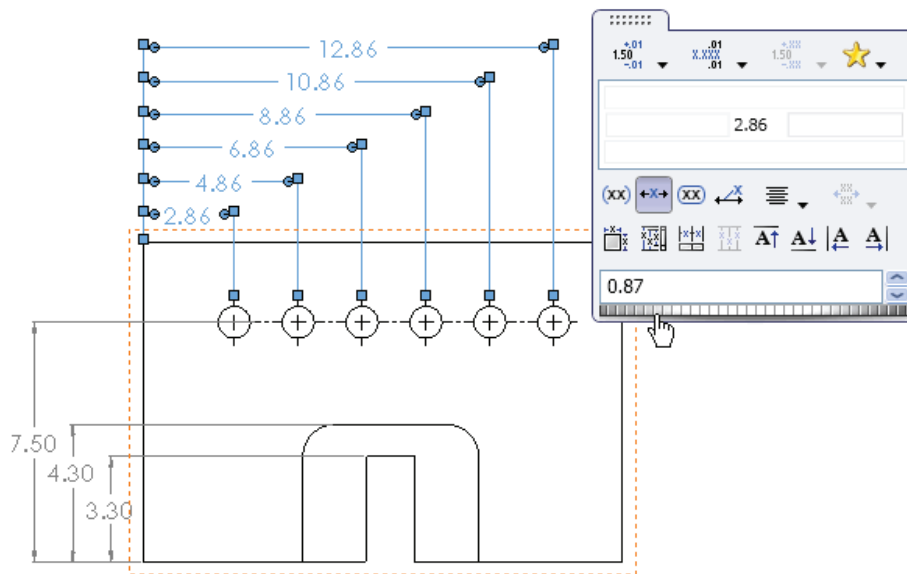
2. Box select all of the dimensions.
3. Move the pointer over the **Dimension Palette** rollover button  to display the dimension palette.
4. Click **Auto Arrange Dimensions** .
5. Click in the graphics area off of the dimension palette.  
The dimensions are automatically arranged.



## Adjusting Spacing

To adjust the horizontal spacing in the dimensions at the top of the drawing:

1. Select the horizontal dimensions at the top of the drawing and display the dimension palette.

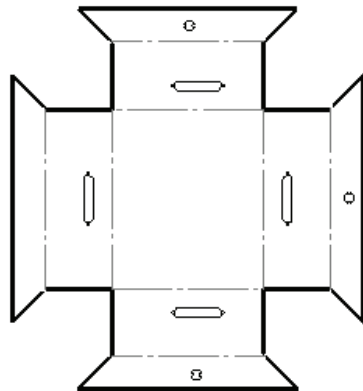


2. Move the thumbwheel to the right and left to adjust the **Dimension Spacing Value**.

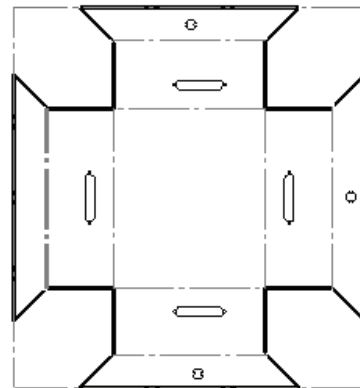
## Bounding Box for Sheet Metal

You can display a bounding box that fits a flat pattern for sheet metal parts and bodies, with or without grain direction.





No Bounding Box



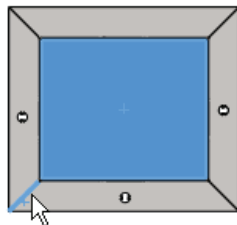
Bounding Box

To display the bounding box:

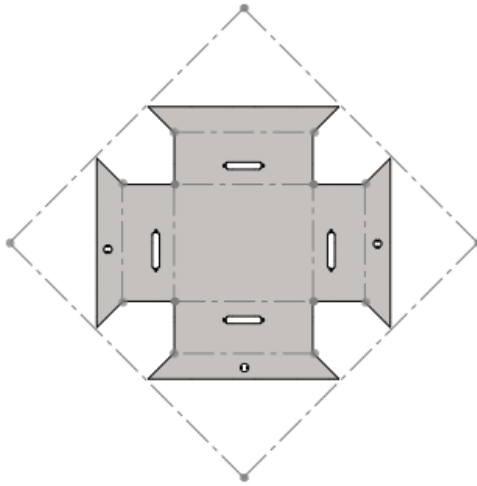
1. Right-click a drawing view and click **Properties**.
2. In the Drawing View Properties dialog box, select **Display bounding box**.

To set grain direction:

1. In the part, edit the Flat Pattern feature.
2. Click in **Grain Direction**.
3. Select an edge or sketch line to define the grain direction.



4. Click .



## Cut List Properties in Drawings

You can insert the cut list properties as an annotation when you insert a flat pattern view of a sheet metal part in a drawing.


To insert the cut list properties:

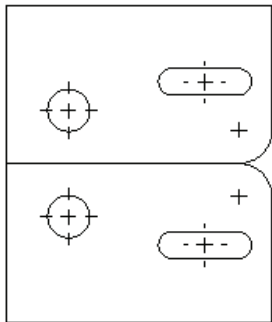
1. In the graphics area, right-click the flat pattern view and click **Annotations > Cut List Properties**.
2. Click to place the list in the graphics area.

## Center Marks in Assembly Drawings

You can insert center marks automatically in assembly drawings.


To insert center marks on holes, fillets, or slots automatically in assembly drawing views:

1. In an assembly drawing document, click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the Document Properties tab, click **Detailing**.
3. Under **Auto insert on view creation**, select from:
  - **Center marks - holes - assembly**
  - **Center marks - fillets - assembly**
  - **Center marks - slots - assembly**



## Cosmetic Threads

You can configure callouts for cosmetic threads.

 Cosmetic thread callouts are editable, and therefore configurable, only when **Standard** is set to **None** in the Cosmetic Thread PropertyManager.

Use one of the following methods:

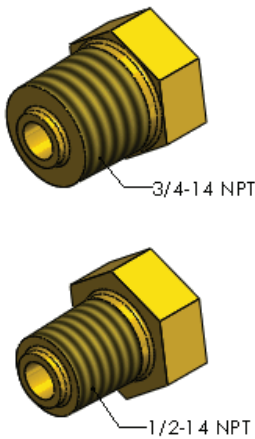
**PropertyManager** Under **Thread Callout**, click **Configurations** and specify the configurations to which the callout applies.

**Design Table** The column header for controlling the callout of a cosmetic thread uses this syntax:

`$THREAD_CALLOUT@cosmetic_thread_feature_name`

Example:


	A	B
1	Design Table for: 560726	
2		\$THREAD_CALLOUT@Cosmetic Thread1
3	P001	3/4-14 NPT
4	P002	1/2-14 NPT

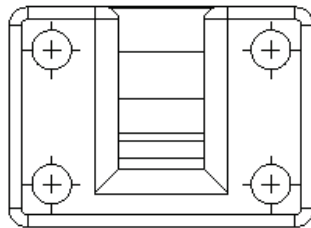


## Display Scale in Orthogonal Views

You can display the scale of an orthogonal view if the scale is different than the default drawing scale. Previously, you could only display the scale of a detail, section, or auxiliary view in a drawing.

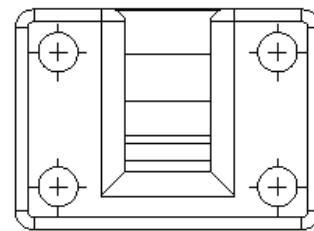
To display the scale of an orthogonal view:

1. In a drawing, click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the Document Properties tab, expand **View Labels** and click **Orthographic**.
3. Select **Show label if view scale differs from sheet scale**.
4. Click **OK**.



SCALE 1 : 1.5

Scale Displayed



Scale Not Displayed

## Drawing Sheet Format

The default location for storing drawing sheet formats is now at:

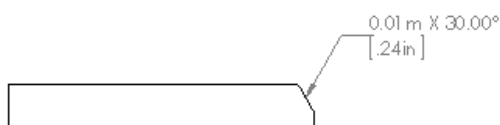
<b>Windows XP</b>	C:\Documents and Settings\All Users\Application Data\SolidWorks\version\lang\language\sheetformat
<b>Windows 7</b>	C:\ProgramData\SolidWorks\version\lang\language\sheetformat
<b>Windows Vista</b>	C:\ProgramData\SolidWorks\version\lang\language\sheetformat

To change the default location, click **Tools > Options > System Options > File Locations**. In **Show folder for**, select **Sheet Formats**.

## Dual Dimensions for Chamfers

You can display dual units in chamfer dimensions. For example, you can display the chamfer dimensions in inches and centimeters.

To display dual dimensions for all chamfers in a drawing, click **Tools > Options > Document Properties > Dimensions > Chamfer**. Under **Dual dimensions**, select **Dual dimensions display**. To display units, select **Show units for dual display**.



To display dual dimensions for a particular chamfer, select the chamfer dimension and then in the Dimension PropertyManager, select **Dual Dimension**.

## Hide Bodies in Drawing View

In addition to hiding components in a drawing view, you can now hide a body in a drawing view.

### Hiding a Body

To hide a body:

In the drawing view, right-click the body and click **Show/Hide**, then **Hide Body**.

### Showing a Body

To show a body:

1. In the FeatureManager design tree or graphics area, right-click the drawing view and click **Properties**.
2. In the Drawing View Properties dialog box on the Hide/Show Bodies tab, select the body to view and press **Delete**.
3. Click **OK**.

## Hole Tables

### Tags

You can renumber the tags when you add or delete holes from a hole table.

#### Renumbering All Tags

To renumber all tags:


Right-click the hole table in the graphics area or in the FeatureManager design tree and select **Renumber All Tags**.

#### Renumbering a Series

To renumber a series:


Right-click any row in the series and select **Renumber Series**.

In this example, hole A3 is removed and the corresponding A series is renumbered.



TAG	X LOC	Y LOC	SIZE
A1	25.23	67.53	Ø10 THRU
A2	25.23	94.68	Ø10 THRU
A4	57.89	94.68	Ø10 THRU
B1	89.73	84.68	Ø20 THRU
B2	132.39	84.68	Ø20 THRU
C1	110.35	37.11	Ø25 THRU

Before renumbering series



TAG	X LOC	Y LOC	SIZE
A1	25.23	67.53	Ø10 THRU
A2	25.23	94.68	Ø10 THRU
A3	57.89	94.68	Ø10 THRU
B1	89.73	84.68	Ø20 THRU
B2	132.39	84.68	Ø20 THRU
C1	110.35	37.11	Ø25 THRU

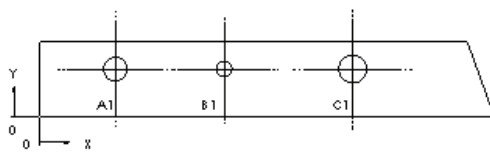
After renumbering series

## Dual Unit Support

You can display dual units in hole tables. For example, you can display the size of a hole in millimeters and inches.

To display dual dimensions in hole tables, click **Tools > Options > Document Properties > Tables > Hole**. Under **Dual dimensions**, select **Dual dimensions display**. To display units, select **Show units for dual display**.

You can also right-click a hole table and click **Show dual dimensions**. When dual dimensions are displayed, you can right-click the hole table and click **Show units for dual dimensions**.



TAG	X LOC	Y LOC	SIZE
A1	[0.161in] 4.09mm	[0.100in] 2.54mm	Ø[0.051in] 1.284mm
B1	[0.393in] 9.99mm	[0.100in] 2.54mm	Ø[0.033in] 0.836mm
C1	[0.668in] 16.96mm	[0.100in] 2.54mm	Ø[0.059in] 1.503mm

## Merge and Unmerge Cells in Tables

You can merge and unmerge cells in a table using tools from the Table pop-up toolbar.

To merge cells:


1. Select the cells.
2. Click **Merge Cells**  (Table pop-up toolbar).

To unmerge cells:

1. Select the cell.
2. Click **Unmerge a Cell**  (Table pop-up toolbar).

## Notes


### Fit Text in Notes

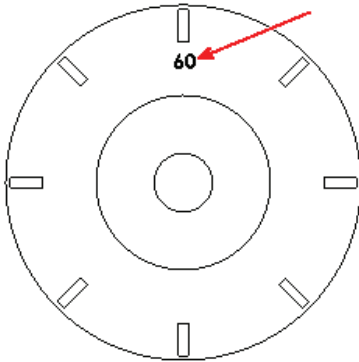
The **Fit Text**  tool is available in the PropertyManager and pop-up toolbar when you create notes. Previously, it was available only when you edited notes.


### Pattern Notes

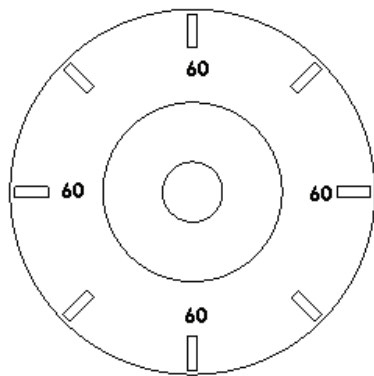
You can display notes in linear or circular patterns in drawings.

First create a note, then create the linear or circular pattern of the note.

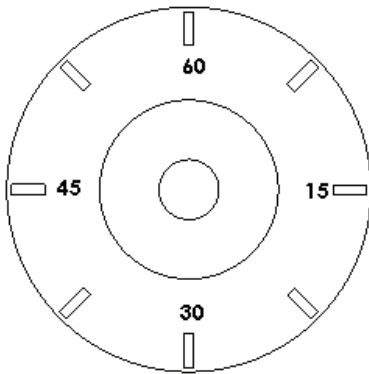
1. In a drawing, click **Note**  (Annotation toolbar) or **Insert > Annotations > Note**.
2. In the PropertyManager, set the options for the note and place it in the drawing.



3. To create a circular note pattern, click **Circular Note Pattern**  (Annotations toolbar), or click **Insert > Annotations > Circular Note Pattern**.
4. In the PropertyManager, set the options.




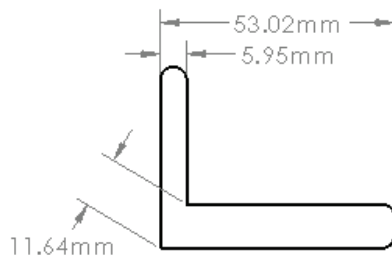
5. To edit the notes, double-click each note and enter the text.



## Show Dimension Units

You can show dimension units in drawings.

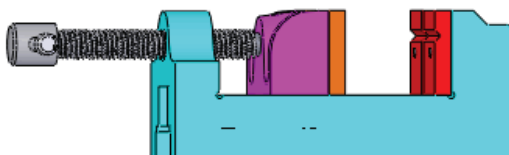
1. In a drawing, click **Options**  (Standard toolbar) or **Tools > Options**.
2. On the Document Properties tab, click **Dimensions** and select **Show units of dimensions**.
3. Click **OK**.



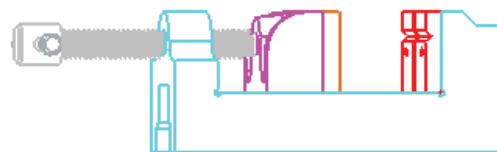
## Show Model Colors in Drawings

You can view the model colors of a part or assembly in a drawing in HLR/HLV. Any assigned layer overrides this setting.

To show model colors, click **Tools > Options > Document Properties > Detailing**. Select **Use model color for HLR/HLV in drawings**.



Assembly with colors



Drawing with colors




## GB Drafting Standard

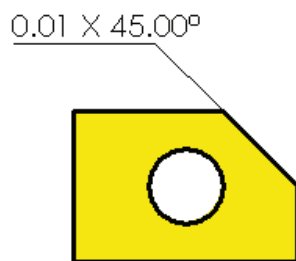
Additional requirements of the GB drafting standard are supported.

In **Tools > Options > Document Properties > Dimensions**:

- Arrow length increased from 3.3mm to 4.08mm.
- Arrows changed from hollow to solid.
- Extension line:
  - **Gap** changed from 1mm to 0mm.
  - **Beyond dimension line** changed from 1mm to 2mm.

In **Tools > Options > Document Properties > Dimensions > Chamfer**, a new **Text**

**position** option, **Horizontal Text, along the model line extension**,  puts the chamfer dimension in line with the model edge.






## ANSI Drafting Standard

The SolidWorks® software supports some of the requirements of ASME Y14.5-2009.

### Datum Feature Symbols


#### Leaders

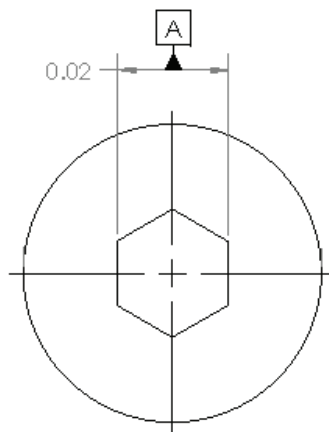
You can use leaders for datum feature symbols that you apply to planar surfaces.

Click **Datum Feature**  (Annotation toolbar) or **Insert > Annotations > Datum Feature Symbol**. In the PropertyManager, under **Leader**, click **Leader**  or **No Leader** .

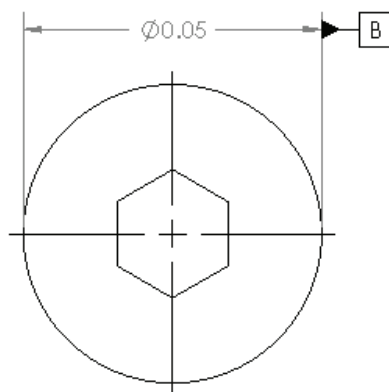
#### Attachment to Linear Dimensions

You can attach datum feature symbols to linear dimensions.

Click **Datum Feature**  (Annotation toolbar) or **Insert > Annotations > Datum Feature Symbol**. In the PropertyManager, set the options, then select a linear dimension to which you want to attach the symbol. The datum is placed in the center of the dimension.




If you attach the symbol to a dimension that is centered between its extension lines, the symbol is placed as shown.

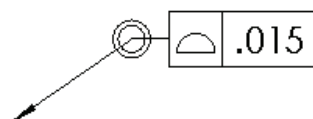


## Geometric Tolerance Symbols

### Leaders

**All Over Leader** is available as a new leader type.

Click **Geometric Tolerance**  (Annotation toolbar) or **Insert** > **Annotations** > **Geometric Tolerance**. In the PropertyManager, under **Leader**, click **All Over Leader**



# 12

## eDrawings

---

Available in SolidWorks® Professional and SolidWorks Premium.

This chapter includes the following topics:

- **Display Enhancements**
- **File Synchronization**
- **Triad Manipulation**
- **Filtering by Component Name**
- **Native 64-Bit Support**

### Display Enhancements

The performance of display and printing in SolidWorks® eDrawings® is significantly improved for some documents.

eDrawings now displays DWG files more accurately. Gradient hatches are supported, and the handling of overlapping entities more closely matches the display order used in AutoCAD.

### File Synchronization

eDrawings can now detect when the graphics data for a component may not be in sync with a later version modified and saved within a SolidWorks assembly.

When you open a potentially outdated component, a watermark displays a warning and suggests that you rebuild the part in SolidWorks.

To control the display of the watermark during the session, click **View > Show Warning Watermark**.

### Triad Manipulation



You can manipulate components of an assembly with greater control and precision.

You can use the triad tool to constrain component movement to a coordinate axis or plane and to rotate components about a coordinate axis. You can also enter values to specify position, translation, or rotation.

Click **Tools > Move Component**  and select **Use triad or enter values**.

### Filtering by Component Name

You can filter the assembly tree to show component names that contain text you specify. This feature was previously available only in eDrawings for Mac®.

In the Component tree , type your search text in **Enter text to filter list** .

## Native 64-Bit Support

SolidWorks eDrawings software can now run as a native 64-bit application on a 64-bit Microsoft® Windows® operating system, which permits larger documents to be opened.

# 15

## Import/Export

---

This chapter includes the following topics:

- **Exporting .IFC Files**
- **DXF DWG Import Wizard**
- **Exporting Sheet Metal Parts to DXF or DWG Files**

### Exporting .IFC Files

You can export SolidWorks® models to the Industry Foundation Classes .ifc files format.

To export SolidWorks models as .ifc files:

1. Click **File** > **Save As**.
2. For **Save as type**, select **IFC 2x3**.
3. Navigate to the correct folder and enter a file name.
4. Click **Options**, select the OmniClass™ and units, and click **OK**.
5. Click **Save**.

For more information, see *SolidWorks Help: .IFC Files*.

### DXF DWG Import Wizard

#### Importing Layers from .DWG or .DXF Files

When importing a .dwg or .dxf file as a 2D sketch for a part, you can create a new sketch for each layer in the file.

1. Open a .dwg file with layers.
2. In the DXF/DWG Import wizard, select **Import to a new part as** and **2D sketch**.
3. Click **Next**.
4. Select **Import each layer to a new sketch**.
5. Select other options and click **Next** or **Finish**.

#### Defining the Sketch Origin and Orientation on .DWG or .DXF Import

When importing a .dwg or .dxf file as a 2D sketch for a part, you can define the model origin and orientation.

1. Open a .dwg file.
2. In the DXF/DWG Import wizard, select **Import to a new part as** and **2D sketch**.
3. Click **Next**.
4. Select part document options and click **Next**.

5. Click **Define Sketch Origin** and click a point in the sketch preview to define the origin.
6. Adjust the origin values and click **Apply**.
7. To change the model orientation about the origin, select **Rotate about the origin** and enter the angle of rotation.
8. Select other options and click **Finish**.

### Filtering Sketch Entities on .DWG or .DXF Import

When importing a .dwg or .dxf file as a 2D sketch for a part, you can filter out unnecessary entities.

1. Open a .dwg file.
2. In the DXF/DWG Import wizard, select **Import to a new part as** and **2D sketch**.
3. Click **Next**.
4. Select part document options and click **Next**.
5. In the preview, select entities to remove and click **Remove Entities**.



To undo this action, click **Undo Remove Entities** .

6. Select other options and click **Finish**.

### Repairing Sketches After .DWG or .DXF Import

When importing a .dwg or .dxf file as a 2D sketch for a part, you can launch the SolidWorks Repair Sketch tool from the DXF/DWG Import Wizard to fix gap or overlap errors after import.


1. Open a .dwg file.
2. In the DXF/DWG Import wizard, select **Import to a new part as** and **2D sketch**.
3. Click **Next**.
4. Select part document options and click **Next**.
5. Select **Run Repair Sketch**.
6. Select other options and click **Finish**.

## Exporting Sheet Metal Parts to DXF or DWG Files

### Exporting a Bounding Box

When exporting a sheet metal part as a .dxf or .dwg file, you can export the bounding box and assign the bounding box to a specific layer.

To assign the bounding box sketch to a layer:


1. Click **File > Save As**.
2. For file type, select **.dxf** or **.dwg**.
3. Click **Options**.
4. In the DXF/ DWG Output PropertyManager, under **Entities to Export**, select **Bounding box**.
5. Select other options and click .
6. In the SolidWorks to DXF/DWG Mapping dialog box:

- Assign layers to entities.
- Map other properties.
- Click **OK**.

## Exporting Bend Line Directions

You can map bend line directions to specific layers when you export sheet metal models as .dxf or .dwg files. For example, in sheet metal parts with up and down bend directions, you can map the different bend line directions to separate layers when you export the part.

To export and map bend line directions for a sheet metal part:

1. Click **File > Save As**.
2. For file type, select **.dxf** or **.dwg**.
3. Click **Options**.
4. Under **Custom Map SolidWorks to DXF/DWG**, select **Enable**.
5. Set other export options and click **OK**.
6. In the DXF/ DWG Output PropertyManager, under **Entities to Export**, select **Bend lines**.
7. Select other options and click .
8. In the SolidWorks to DXF/DWG Mapping dialog box:
  - Assign layers to entities.
  - Map other properties.
  - Click **OK**.

# Large Scale Design

---

Large Scale Design helps you use the tools that the SolidWorks® software offers for creating equipment, facilities, and plants.


This chapter includes the following topics:

- **Walk-through**
- **Exporting .IFC Files**
- **Grid System**

## Walk-through

You can use the **Walk-through** function to explore or create a video of the 3D geometry of plants or other systems. You “see” the geometry of the system as you maneuver through it. You can save what you see and play it back for further study.

To activate the function, click **View > Lights and Cameras > Add Walk Through**.

Define motion constraints and other parameters, such as the camera height, in the PropertyManager, and click **Capture Motion** to open the control panel. Define your starting point, click **Record** , and use the controls to maneuver through the system.

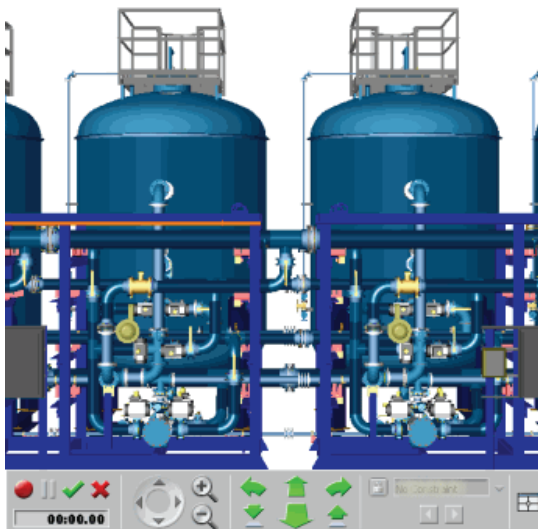
You can also use the arrow keys and the mouse to maneuver through the system. You may eventually find that a combination of mouse and keyboard controls allows you to maneuver through the system most efficiently.

To use the mouse, left-click to manipulate the view.

The keyboard controls are:



<b>Arrows</b>	Move forward or back, turn left or right.
<b>Shift + arrows</b>	Move up, down, left, or right.
<b>Control + arrows</b>	Turn up or down.
<b>Alt + arrows</b>	Look up, down, left, or right. Press the <b>Alt</b> and <b>arrow</b> keys simultaneously.
<b>Home</b>	Reset "view" direction.
<b>Z, Shift + Z</b>	Zoom out or in.
<b>+, -</b>	Increase or decrease speed.
<b>1 - 9</b>	Set speed.
<b>Scroll Lock</b>	Lock to constraint.
<b>Page Up</b>	Next constraint.
<b>Page Down</b>	Previous constraint.
<b>R</b>	Record.
<b>Space</b>	Pause.
<b>Esc</b>	Cancel.
<b>M</b>	Toggle map view.



## Exporting .IFC Files

You can export SolidWorks® models to the Industry Foundation Classes .ifc files format.

To export SolidWorks models as .ifc files:

1. Click **File** > **Save As**.
2. For **Save as type**, select **IFC 2x3**.
3. Navigate to the correct folder and enter a file name.

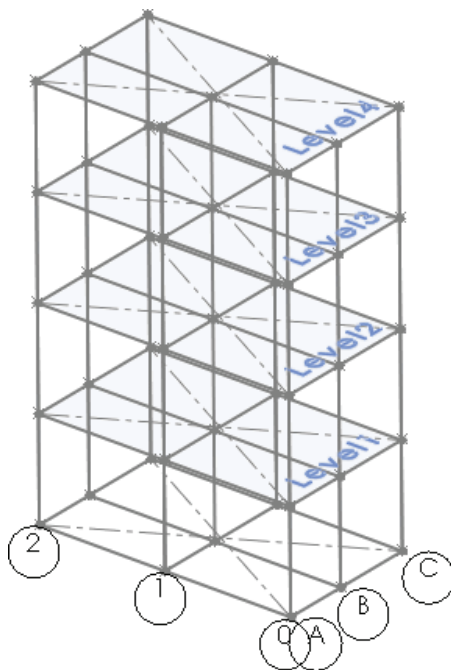
4. Click **Options**, select the OmniClass™ and units, and click **OK**.
5. Click **Save**.

For more information, see *SolidWorks Help: .IFC Files*.

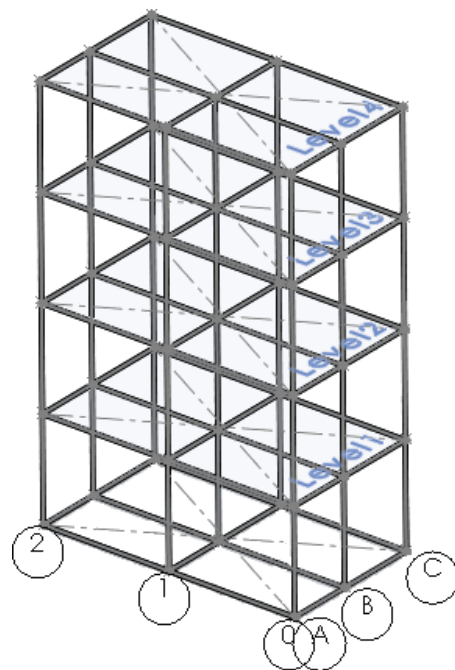
## Grid System

You can use the **Grid System** tool to lay out a grid system for large structures. The grid system is useful when creating welded structures. The grid system is also helpful when you work with multiple users who use different third-party applications to make grids. Multiple users can use the grid system and work from the same baseline.


You can create a grid to help specify the location of key elements in structures. When you use the **Grid System** tool, you create a sketch to represent the grid. You can then specify the number of floors for the structure and the distance between each floor. The sketch is replicated for every floor in the structure. Balloons are attached to the grid items to help with orientation.



Grid system sketch





Grid system with weldments applied

Click **Grid System**  (Features toolbar) or **Insert** > **Reference Geometry** > **Grid System**.

See *SolidWorks® Help: Creating a Grid System*.

The graphics area can appear cluttered when the grid system creates a lot of geometry or if you create multiple grid systems. To reduce clutter, you can hide the geometry. Display options are available to help you visualize the grid structure and to ensure correct grid inferencing.

To access the display options, in the FeatureManager design tree, right-click **Grid System**  and click **View Grid Components**. In the View Grid Components dialog box, select an item and click a display option, such as **Normal To** .


# Model Display

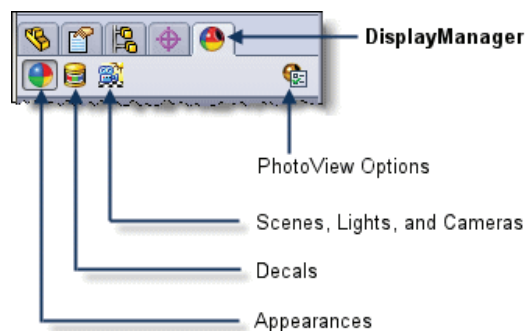
This chapter includes the following topics:

- **DisplayManager**
- **Appearances**
- **Lights**
- **Scenes**
- **Decals**
- **PhotoView 360**
- **Working with Appearances and Rendering a Model**

## DisplayManager ★

The SolidWorks® DisplayManager is the central location for managing appearances, decals, scenes, cameras, lights, and Walk Through. Use the DisplayManager to view, edit, and delete display items applied to the current model.


On the Manager Pane, click the DisplayManager tab .



See *SolidWorks Help: DisplayManager*.

## Appearances

All appearance functionality, including controls previously available only in PhotoView or PhotoWorks™, is now available in SolidWorks Standard. The DisplayManager lists the appearances applied to the currently active model. You can also save custom appearances in SolidWorks Standard.

On the DisplayManager tab, click **View Appearances** .



See *SolidWorks Help: Appearances*.

## Lights

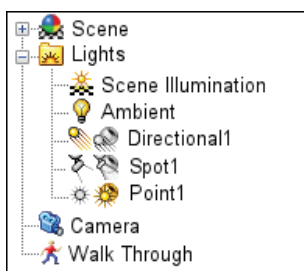
The DisplayManager is the central location for managing all aspects of lighting, including lighting controls that are available only when PhotoView is added in. The DisplayManager lists the lights applied to the currently active model. Controls for shadows and fog now provide a stronger integration with PhotoView. Light intensity is controlled with wattages.

Lighting controls are separate for SolidWorks and PhotoView 360.

**SolidWorks** By default, point, spot, and directional lights are on in SolidWorks. Scene lighting is not possible in RealView, so you often need to illuminate models manually.

**PhotoView** By default, lighting is off in PhotoView. With lights off, you can use the realistic lighting provided by scenes, which is usually sufficient for rendering. You typically need additional lighting in PhotoView to illuminate occluded spaces in the model.

The icons for specific lights in this image from the Scene, Lights, and Cameras pane of the DisplayManager have the following meanings:



<b>Directional1</b>	On in SolidWorks
	Off in PhotoView
<b>Spot1</b>	Off in SolidWorks
	Off in PhotoView
<b>Point1</b>	Off in SolidWorks
	On in PhotoView

On the DisplayManager tab, click **View Scenes, Lights, and Cameras** .

See *SolidWorks Help: Lights*.

## Scenes

Scene functionality is enhanced to allow full control of the scene that is visible behind the model. The DisplayManager lists the background and environment applied to the currently active model. The new Edit Scene PropertyManager, available from the **View Scene** pane in the DisplayManager, lets you size the floor, control the background or environment, and save custom scenes.

On the DisplayManager tab, click **View Scenes, Lights, and Cameras** .

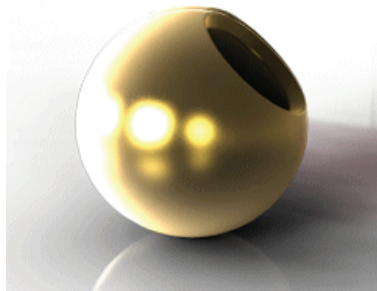
Scenes are simplified and now consist of the following:

- A spherical environment based on a preset scene or image you select is mapped around the model.
- A 2D background that can be a single color, a gradient of color, or an image you select. Although partially obscured by the background, elements of the environment are reflected in the model. You can also turn off the background and show the spherical environment instead.
- A 2D floor on which you can see shadows and reflections. You can change the distance of the model from the floor.

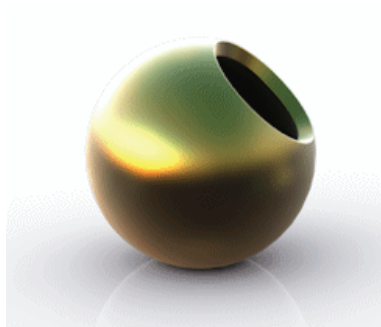
The **Insert Picture** functionality is removed. Background images in legacy models are discarded and are not displayed. You can use the Edit Scene PropertyManager to add a background image to the scene.

To save a custom scene, click **Save Scene** on the Advanced tab of the Edit Scene PropertyManager.

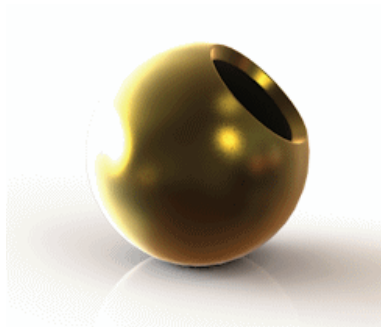
Office Space scene



Rooftop scene



Warm Kitchen scene

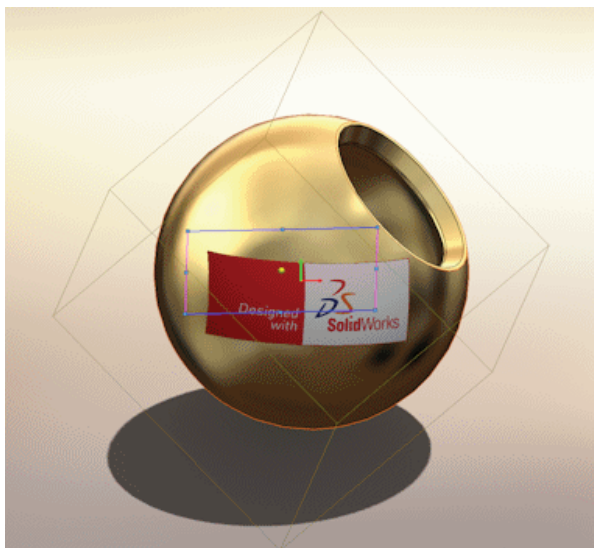


See *SolidWorks Help: Scenes*.

## Decals

Decals are now part of SolidWorks Standard. Use the DisplayManager to view and manage decals applied to the current model.

On the DisplayManager tab, click **View Decals** .



See *SolidWorks Help: Decals*.

# 18

## Mold Design

---

This chapter includes the following topics:

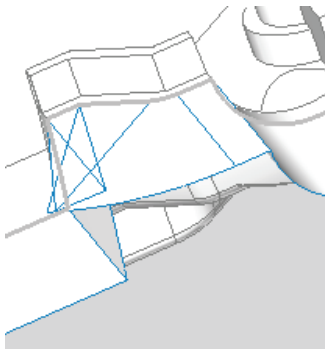
- **Manual Mode for Creating Parting Surfaces**

### Manual Mode for Creating Parting Surfaces

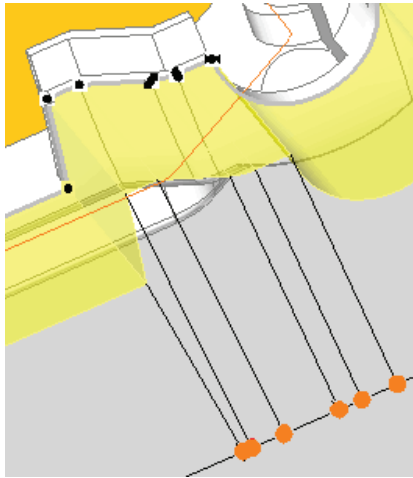
You can create parting surfaces using **Manual Mode**, which lets you modify surface direction. **Manual Mode** lets you override the defined parting surface direction and manually create a portion of the surface.

Select **Manual Mode** in the Parting Surface PropertyManager to display handles that you can manipulate to adjust the parting surface. To further modify the surface, right-click an inner vertex and select **Start fill surface region** or **End fill surface region**.

The first image below shows an irregular parting surface. In the second image the surface has been fixed.







# Parts and Features

---


This chapter includes the following topics:

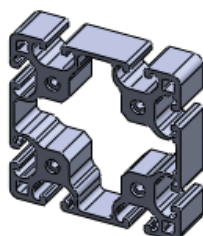
- [Parts](#)
- [Features](#)
- [Surfaces](#)
- [FeatureWorks](#)

## Parts

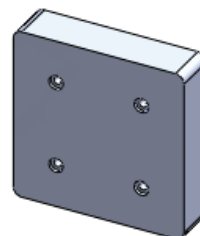
### Defeature for Parts

With the **Defeature** tool, you can remove details from a part or assembly and save the results to a new file in which the details are replaced by dumb solids (that is, solids without feature definition or history). You can then share the new file without revealing all the design details of the model.

Click **Defeature**  (Tools toolbar) or **Tools > Defeature** to access the Defeature PropertyManager, which provides tools for manual and automatic selection of details to keep and remove.



Before



After

For a step-by-step example, see [Defeature for Assemblies](#) on page 25.

## Equations

### Sharing Equations Among Models

You can share equations and global variables among models.

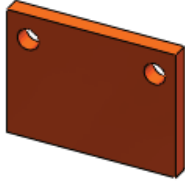
You export selected equations and variables from a model to an external text (.txt) file. Alternatively, you can build the text file manually using an application such as Notepad. Then you import information from the text file into other models. You can choose to link

models to the text file, so that changes you make in the text file update the equations and variables in the models.

### Exporting Equations



In this example, you export equations from a part to a text file.

1. Open `install_dir\samples\whatsnew\parts\frontplate_01.sldprt`.



2. Click **Tools > Equations**.  
The Equations dialog box lists five equations.
3. Click **Export**.  
All the equations are listed in the Equations Export dialog box. Under **Active**, all the equations are selected for export. **Link to file** is selected as well.
4. Click **Save**.
5. In the Save As dialog box, for **File name**, type `my_equations`.
6. Click **Save**.  
The equations are saved in a text file. The text file is available to import into other parts and assemblies. Because **Link to file** was selected, changes you make in the text file are propagated to the model.

In the Equations dialog box:

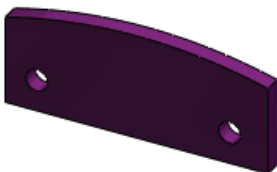
- Under **Active**, the icon  indicates equations that are linked to the external file.
- At the bottom of the dialog box, **Linked File**  displays the path to the external file.

7. Click **OK**.
8. Save the part. If prompted to rebuild, click **Yes**.

### Importing Equations

Now import the equations from the text file to another part.




1. Open `install_dir\samples\whatsnew\parts\backplate_01.sldprt`.

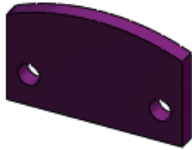


2. Click **Tools > Equations**.
3. In the Equations dialog box, click **Import**.
4. In the Open dialog box:
  - a) Select `my_equations.txt`.
  - b) Select **Link to file**.
  - c) Click **Open**.

Five equations are imported into the model.

In the Equations dialog box:

- Under **Active**, the icon  indicates equations that are linked to the external file.
  - At the bottom of the dialog box, **Linked File**  displays the path to the external file.
5. Click **OK**.
  6. Click **Rebuild**  (Standard toolbar).  
The equations are applied to the model.



7. Save the part.

### Suppression States of Features and Components

You can use equations to control the suppression state of part features and assembly components.

In the Add Equation dialog box, you use the Visual Basic **IIf** function to specify when to suppress or unsuppress a feature or component.

 The syntax of the Visual Basic **IIf** function is:

```
iif(expression, truepart, falsepart)
```

where:

- *expression* is the expression you want to evaluate
- *truepart* is the value to use if *expression* is true
- *falsepart* is the value to use if *expression* is false

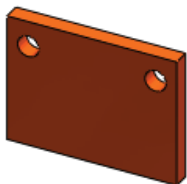
### Defining a Feature Suppression Equation


In this example, you suppress a hole in a plate when the length of the plate is less than 40 mm.

The hole you want to suppress is part of a linear pattern, so you define an equation to suppress or unsuppress the linear pattern feature depending on the part length.

To suppress the feature:

1. Open `frontplate_01.sldprt` from the previous example.



2. In the FeatureManager design tree, right-click **Equations**  and click **Add Equation**.  
The Equations and Add Equations dialog boxes open.
3. In the FeatureManager design tree, click the linear pattern feature **LPattern1**.

"LPattern1" appears in the Add Equations dialog box.




If **Instant 3D** is active, you need to click-pause-click **LPattern1**. The first click selects the linear pattern. The second click adds it to the Add Equations dialog box.

4. In the dialog box, complete the equation:

```
"LPattern1" = iif ("overall length"<40, "suppressed", "unsuppressed" )
```



You can type the entire equation or use the following tips to enter various pieces:

- To insert global variable "overall length", expand **Equations**  in the FeatureManager design tree and click "overall length"=100.
- To insert "suppressed" and "unsuppressed", click the **suppress** and **unsuppress** buttons in the dialog box.

5. Click **OK**.

The new equation is added to the Equations dialog box.


6. Click **OK**.

#### Testing the Feature Suppression Equation


Now decrease the length of the part to 35 mm, which triggers the suppression of the linear pattern feature.

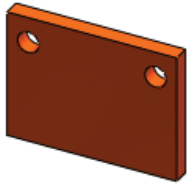
The length of the part (length@outline) is controlled by the global variable "overall length", which is defined in the linked external file.

To trigger the suppression state equation:

1. Open my\_equations.txt.
2. For "overall length", change 100 to 35.
3. Save the text file.
4. Click **Rebuild**  (Standard toolbar).  
The external equations file is checked for changes and the length of the part is updated from 100 to 35. Because the length is now less than 40 mm, **LPattern1** is suppressed, which eliminates the second hole.




5. Edit my\_equations.txt again, and change "overall length" back to 100.
6. Save and close the text file.
7. Click **Rebuild**  (Standard toolbar).  
The external equations file is checked for changes and the length of the part is updated from 35 to 100. Because the length is now greater than 40 mm, **LPattern1** is unsuppressed.



8. Save the part.

#### Exporting the Feature Suppression Equation

Now add the new equation to the external text file.

1. In the FeatureManager design tree, right-click **Equations**  and click **Edit Equation**.
2. In the dialog box, click **Export**.  
The new equation appears at the bottom of the list.
3. Click **Save**.
4. In the Save As dialog box, select `my_equations.txt` and click **Save**. When asked to replace the existing file, click **Yes**.
5. Click **OK**.
6. Open `my_equations.txt`.  
The feature suppression equation appears in the text file.

#### Defining a Component Suppression Equation

Now use an equation to suppress a component in an assembly.

1. Open `install_dir\samples\whatsnew\parts\plate_assembly_111.sldasm`. If prompted to rebuild, click **Yes**.



First, import `my_equations.txt` so that you can use the global variable "overall length" in the component suppression equation.

2. Click **Tools > Equations**.
3. In the Equations dialog box, click **Import**.
4. In the Open dialog box:
  - a) Select `my_equations.txt`.
  - b) Select **Link to file**.
  - c) Click **Open**.


Because the assembly does not contain all the same dimensions as the parts, warnings about invalid equations appear.

5. Click **OK** to dismiss each warning.

Two equations are imported into the model, including global variable "overall length".


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

Now add an equation to suppress the second instance of the pin when overall length is less than 40 mm.

7. In the FeatureManager design tree, right-click **Equations**  and click **Add Equation**.
8. In the FeatureManager design tree, click **pin<2>**.  
"pin<2>" appears in the Add Equation dialog box.
9. In the dialog box, complete the equation:  
"pin<2>" = iif ( "overall length"<40, "suppressed" , "unsuppressed" )




You can type the entire equation or use the following tips to enter various pieces:

- To insert global variable "overall length", expand **Equations**  in the FeatureManager design tree and click "overall length"=100.
- To insert "suppressed" and "unsuppressed", click the **suppress** and **unsuppress** buttons in the dialog box.

10. Click **OK**.  
The new equation is added to the Equations dialog box.
11. Click **OK**.
12. Save the assembly. In the Save Modified Documents dialog box, click **Save All** . If prompted to rebuild, click **Yes**.

#### Testing the Component Suppression Equation

Now decrease the overall length to 35 mm, which triggers the suppression of the linear pattern feature in the two parts and the suppression of the second instance of the pin in the assembly.

1. In my\_equations.txt, for "overall length", change 100 to 35.
2. Save the text file.
3. In the assembly, click **Rebuild**  (Standard toolbar).  
The length of the plates changes to 35 mm and the linear pattern is suppressed. In the assembly, the second instance of the pin is suppressed.



## Global Variables

You can configure global variables.

In a design table, the column header for controlling the value of a global variable uses this syntax:

`$VALUE@global_variable_name@equations`

In the table body cells, type the value for the global variable. If you leave a cell blank, it inherits the value from the configuration that was active when you opened the design table.

Example:

	A	B	C
1	Design Table for: pump		
2			
3	small	250	
4	large	600	

## Features

### Helix

The PropertyManager and callouts are enhanced. Methods of defining a **Variable pitch** helix have been expanded.

#### PropertyManager and Callouts

The PropertyManager and callouts display more information.

- Callouts for **Constant pitch** cases are now displayed in the graphics area, similar to those that already appear for **Variable pitch** cases.
- The callouts show all parameters, including those that are for information only (shown in gray).
- In the PropertyManager, for **Variable pitch**, the table now shows all parameters, including those that are inactive or for information only (shown in gray).

	H	Rev	P	Dia
1	0mm	0	5mm	20mm
2	50mm	10	5mm	20mm
3	150m	20	15mm	20mm
4	200m	25	5mm	20mm
5				20mm

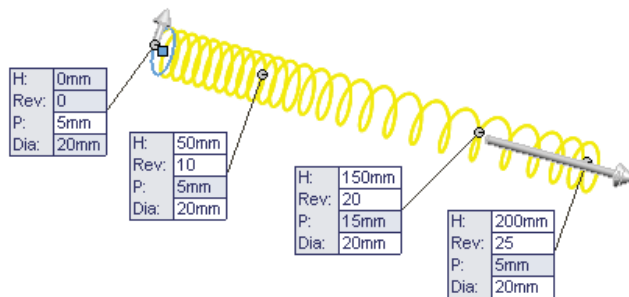
### Variable Pitch Helix

You can define a **Variable pitch** helix by specifying **Height and Revolution**.



Previously, only **Pitch and Revolution** and **Height and Pitch** were available for **Variable pitch**, and you could use **Height and Revolution** only when defining a **Constant pitch** helix.

In a part, select a sketch that contains a circle. Click **Insert > Curve > Helix/Spiral**. In the PropertyManager, under **Defined By**, select **Height and Revolution**. Under **Parameters**, select **Variable pitch**. Enter values for **H** and **Rev** in the table.



## Revolve




More end conditions are available when creating a revolve feature.

New end conditions include:

- **Up To Vertex**
- **Up To Surface**
- **Offset From Surface**

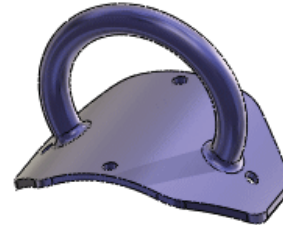
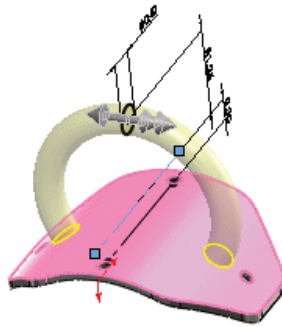
You can specify separate end conditions for each direction (clockwise and counter-clockwise from the sketch plane).

The new end conditions are available when you click the following:

- **Revolved Boss/Base**  (Features toolbar) or **Insert > Boss/Base > Revolve**
- **Revolved Cut**  (Features toolbar) or **Insert > Cut > Revolve**
- **Revolved Surface**  (Surfaces toolbar) or **Insert > Surface > Revolve**

In the PropertyManager, under **Direction1**, in **Revolve Type**, select an end condition. Select **Direction2** to specify an end condition for the second direction.

Example: For this revolved boss, the end condition is **Up To Surface** for **Direction1** and **Direction2**:



## Scale

You can configure the X, Y, and Z scale factors.

Use one of the following methods:

**PropertyManager** Under **Configurations**, specify the configurations to which the scale factors apply.

**Design Table** The column headers for controlling the X, Y, and Z factors of a scale feature use this syntax:

`$X_AXIS@scale_feature_name`

`$Y_AXIS@scale_feature_name`

`$Z_AXIS@scale_feature_name`

Example:

	A	B	C	D
1	Design Table for: bracket			
2		<code>\$X_AXIS@Scale1</code>	<code>\$Y_AXIS@Scale1</code>	<code>\$Z_AXIS@Scale1</code>
3	plastic	1.5	1.7	1.3
4	metal	1.07	1.07	1.12

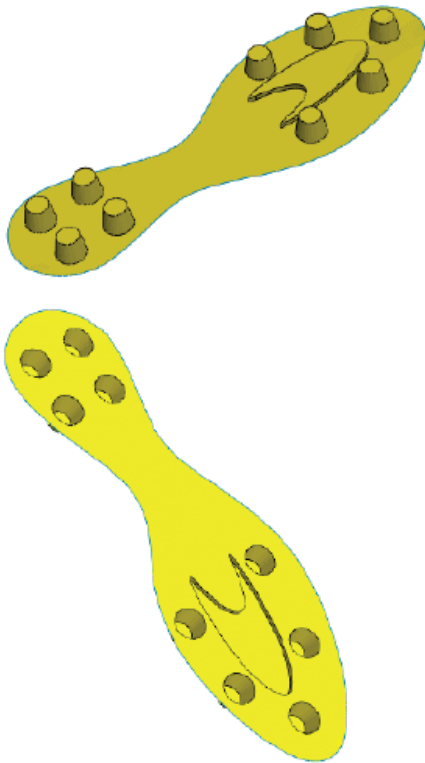


For uniform scaling, only `$X_AXIS@scale_feature_name` needs to be defined.


## Surfaces

### Surface Extrudes From a 2D or 3D Face

You can create extruded surfaces from models that include 2D or 3D faces and knit the extruded surfaces to surrounding features.





Click **Insert** > **Surface** > **Extrude**.

1. Select a face:
  - To extrude from a 3D face, select a 3D face.
  - To extrude from a 2D face, press **Alt** + select the planar face.
2. Select the end condition.
3. For 3D faces, select a plane, edge, 2D face, or sketch line to define the direction of extrusion .



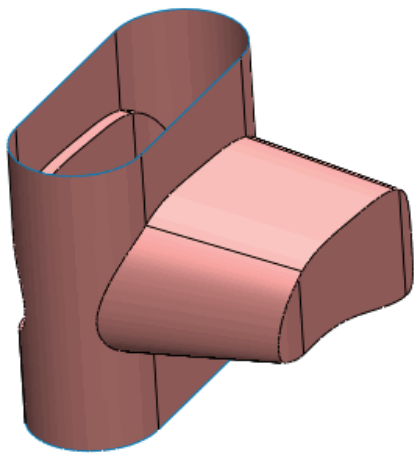
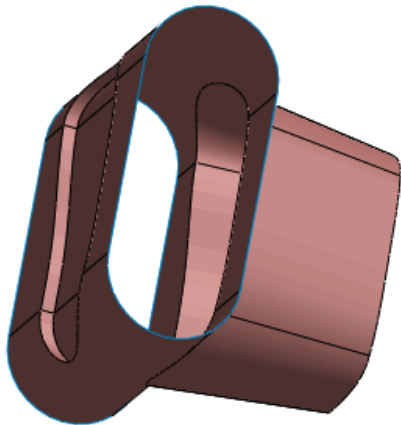
Select a plane to define an extrude direction normal to the plane.

4. To remove the faces defining the extrude from the model after extruding, click **Delete original faces**.
5. To create a single body from the extrude when faces are deleted, select **Knit result** .
6. Set other options and click .

### Surface Extrudes from Faces

You can create extruded surfaces from models that include 2D or 3D faces and knit the extruded surfaces to surrounding features.

In this example, you examine a part that has two surface extrudes. Both extrudes are created from three contiguous faces: one planar face and two 3D faces.

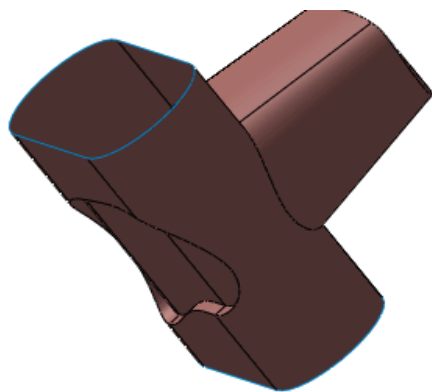


### Opening the Model

First, you open the model and examine some components.

1. Open

`install_dir\samples\whatsnew\surfaces\multiface-surf-extrude_example.SLDPRT.`



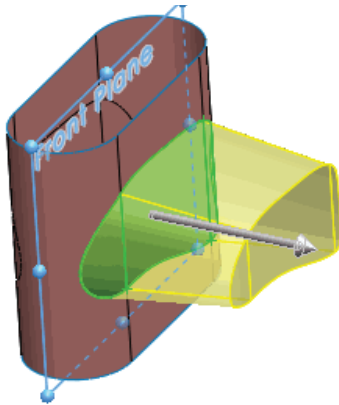
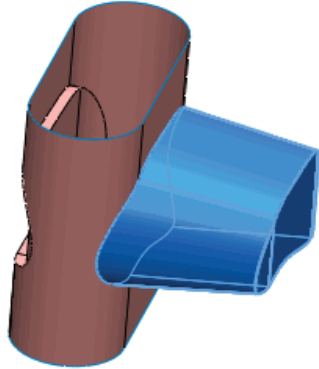
2. Notice the surface extrude features in the FeatureManager design tree.

The two extrusions into and out of the central portion of the model were created by selecting three faces, deleting the original faces, and capping one end of the extrusion.

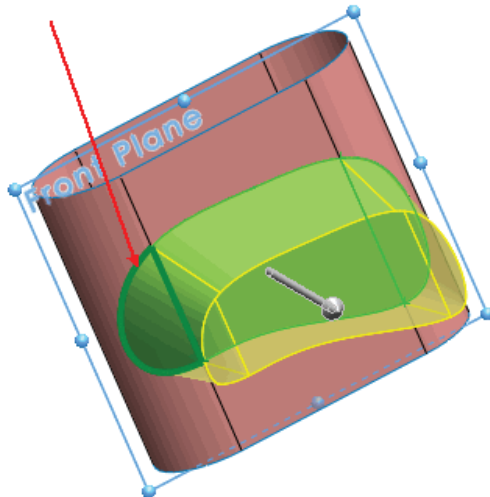
### Viewing the Surface Extrude Options

Next, you examine the options used to specify the surface extrude for the model.

1. Click **Surface-Extrude2** and click **Edit Feature** .

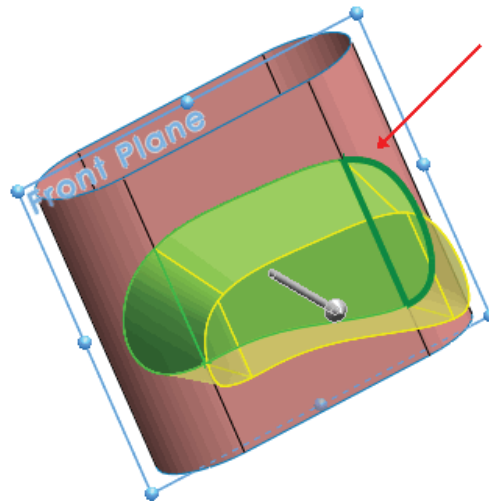


2. To view one of the 3D faces used to create the surface extrude, select **Face<1>** under **Faces to Extrude** in the PropertyManager.

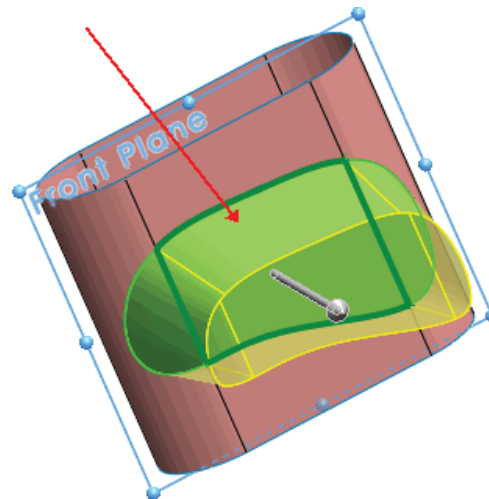


3. To view the other faces used to create the surface extrude, select each of them under **Faces to Extrude** in the PropertyManager.

A 3D face defines each end portion of the extrude.






A planar face defines the middle portion of the extrude.



4. To display the plane defining the direction of extrusion, click **Front Plane** in the PropertyManager.

Notice the extrude direction is normal to the front plane.

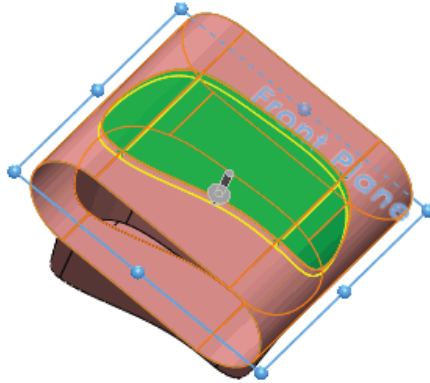
5. Note the selected options:

- **Cap end**. Seals the **Direction 1** end of the extrude with a translated copy of the selected 3D face.
- **Delete original faces** . Deletes the faces selected under **Faces to Extrude** . The 3D faces are removed from the model.
- **Knit result** . Knits the resulting extrude to the surface boundary where the original face is deleted.

6. Close the PropertyManager.



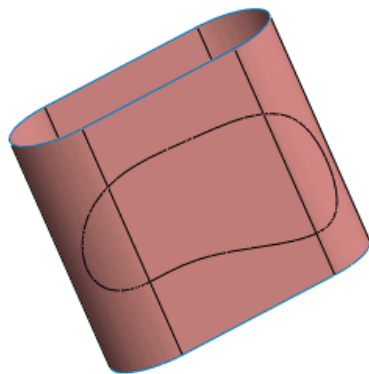
To examine the other surface extrude created from 3D faces, rotate the model and repeat Steps 1 - 6 for **Surface-Extrude3**.



#### Viewing the Original Surface Before the Extrude

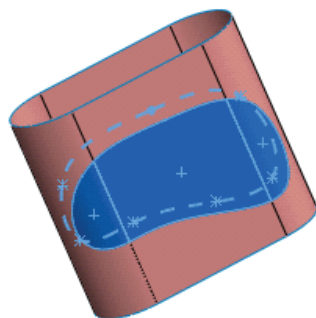
Next, you suppress a surface extrude to view the surface before the extrude.

1. Click **Surface-Extrude2** and select **Suppress** .



The three faces were created by splitting the surfaces with a Split Line feature.

2. Press **Ctrl** + select the three faces in the middle to view the original surface used to create this model.



3. Close the model without saving it.

## Capping an Extruded Surface

You can cap an extruded surface at one end or at both ends.

Click **Insert** > **Surface** > **Extrude**. To seal an end of the extrude, under **Direction 1** in the PropertyManager, select **Cap end**.



You can seal the other end of the extrude by selecting **Cap end** under **Direction 2**. When you cap both ends of an extrude to define an enclosed volume, a solid is created automatically.

Set other options and click .

## FeatureWorks

Available in SolidWorks® Professional and SolidWorks Premium.

### Bosses and Cuts Recognition

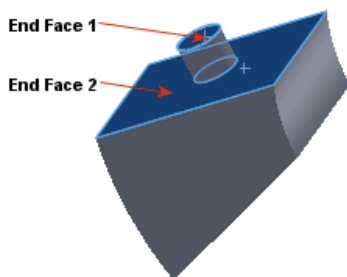
Feature recognition is enhanced and new feature types are recognized.

#### Interactive Recognition of Boss and Cut Sweeps

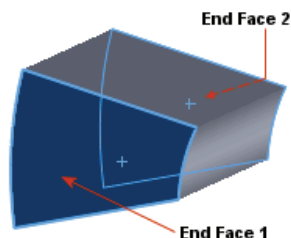
FeatureWorks® recognizes boss sweep and cut sweep features during interactive recognition when two faces are selected.

##### Boss Sweep

1. Open `install_dir\samples\whatsnew\FeatureWorks\FeatureWorks-BossSweep.x_t`.
2. First recognize the upper boss sweep. During interactive recognition of boss sweep features, select **End Face1** and **End Face 2/Support Face**.



3. Then, recognize the lower boss sweep. Select **End Face1** and **End Face 2/Support Face**.



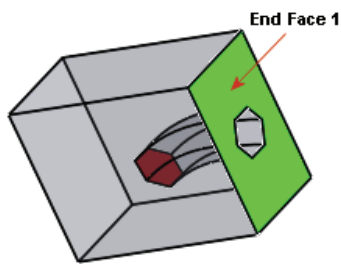




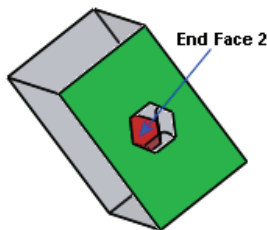
In the **FeatureWorks** PropertyManager under **Interactive Features**, the **Boss-Sweep** feature type replaces **Base-Sweep**. Boss-Sweep recognizes both base and boss sweeps.

### Cut Sweep

1. Open `install_dir\samples\whatsnew\FeatureWorks\FeatureWorks-CutSweep.x_t`.
2. Use interactive recognition and under **Feature type**, select **Cut-Sweep**.
3. Select **End face** and then select **End Face 1**.



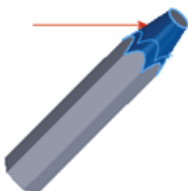
4. Select **End Face 2**. If necessary, rotate the model.



### Interactive Recognition of Cut Revolve Features

FeatureWorks recognizes cut revolve features of the type known as "pencil sharpening cuts" during interactive recognition.

1. Open `install_dir\samples\whatsnew\FeatureWorks\FeatureWorks-Pencil.x_t`.
2. During interactive recognition of cut revolve features, select the face of the cut revolved feature.





Do not select the top planar face. Verify that **Chain revolved faces** is cleared.

### Interactive and Automatic Recognition of Bosses and Cuts From Non-Planar Faces

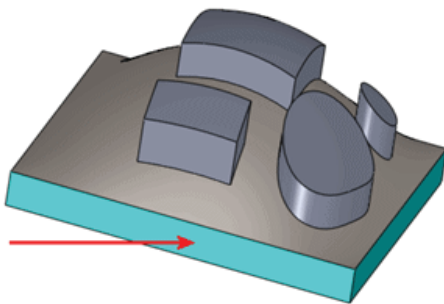
FeatureWorks recognizes extrude features (bosses and cuts) that are created from non-planar faces in both interactive and automatic recognition modes.

1. Open

`install_dir\samples\whatsnew\FeatureWorks\FeatureWorks-NonPlanar_1.x_t.`

2. Use automatic recognition and under **Automatic Features**, select **Extrudes**.

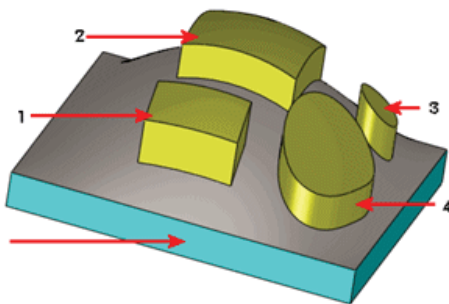
Previously, FeatureWorks could recognize only the highlighted boss extrude.



FeatureWorks now recognizes the original boss extrude as well as the four boss extrudes created from the non-planar face.



The extrudes must be exact offsets from the supporting face.



1. Open

`install_dir\samples\whatsnew\FeatureWorks\FeatureWorks-NonPlanar_2.x_t.`

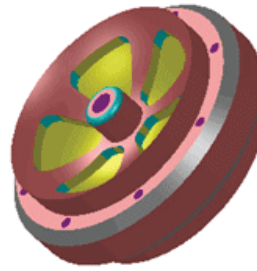
2. Use automatic recognition and under **Automatic Features**, click **Check all filters**



FeatureWorks recognizes one base revolve, ten cut extrudes, nine holes, two chamfers, and 42 fillets. FeatureWorks finds two circular patterns. FeatureWorks maps all of the features automatically.



Original Part

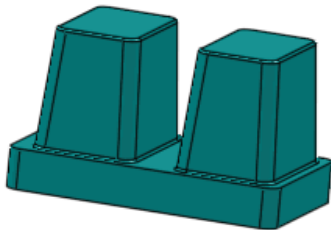


Recognized Features

### Automatic Recognition of Draft Features

FeatureWorks recognizes draft features during automatic recognition. Drafts were previously recognized only in interactive mode.

1. Open `install_dir\samples\whatsnew\FeatureWorks\FeatureWorks-Draft.x_t`.
2. In automatic recognition mode, select **Standard features**. Under **Automatic Features**, select **Drafts** and **Fillets/Chamfers** only.



FeatureWorks maps all of the features automatically.


### Combining Like Features During Automatic Feature Recognition

Automatic feature recognition can now combine fillets, chamfers, or holes of the same geometry into a single feature. For example, seven fillets with the same radius are recognized as one feature. This reduces the number of features in the FeatureManager design tree and lets you group features together for better readability and smaller file size.

1. Open `install_dir\samples\whatsnew\FeatureWorks\FeatureWorks-CombineFeatures.x_t`.
2. In automatic recognition mode, select **Standard features**. Under **Automatic Features**, select **Fillets/Chamfers**.



Previously, FeatureWorks recognized 36 chamfers in this example. Now, one chamfer feature appears in the FeatureManager design tree.

To turn off the combine features functionality, click **FeatureWorks Options**  (Features toolbar) or **Insert > FeatureWorks > Options**. On the Advanced Controls tab, under **Automatic Recognition**, select or clear **Combine Fillets**, **Combine Chamfers**, or **Combine Holes**.

This chapter includes the following topics:

- **Bend Calculation Tables**
- **Convert to Sheet Metal**
- **Flat Patterns**
- **K-Factor in Configurations**
- **Mapping Bend Directions When Exporting to DXF/DWG Files**
- **Mirroring Edge Flanges and Miter Flanges**
- **Sheet Metal Properties**
- **Patterns of Edge Flanges and Tab Features**

## Bend Calculation Tables

You can calculate the developed length of sheet metal parts using bend calculation tables.

In previous releases, you calculated the developed length with K-Factor, bend table, gauge table, bend allowance, and bend deduction methods. With bend calculation tables, you can define different angular ranges, assign equations to those ranges, and calculate the developed length of the part.

Click any sheet metal tool where you can select a calculation method for the developed length. In the PropertyManager, under **Bend Allowance**, in **Bend Allowance Type**, select **Bend Calculation** and set the options.

See *SolidWorks® Help: Bend Calculation Tables*.

## Convert to Sheet Metal

The **Convert to Sheet Metal** tool is enhanced.

### Gauge Tables

You can use gauge tables with the **Convert to Sheet Metal** tool. The sheet metal parameters (material thickness, bend radius, and bend calculation method) use the values stored in the gauge table unless you override them.

Click **Convert to Sheet Metal**  (Sheet Metal toolbar) or **Insert > Sheet Metal > Convert To Sheet Metal**. In the PropertyManager, under **Sheet Metal Gauges**, select **Use gauge table** and select a gauge table. You can specify the file locations for gauge tables in **Tools > Options > System Options > File Locations**. In **Show folders for**, select **Sheet Metal Gauge Table**.

## Rip Types

You can define parameters for rip edges and rip sketches created with the **Convert to Sheet Metal** tool. Available rip types are open butt, overlap, and underlap. You can also control individual rip settings by clicking a rip callout in the graphics area. Additionally, you can set the **Default overlap ratio for all rips** and show or hide callouts in the graphics area.

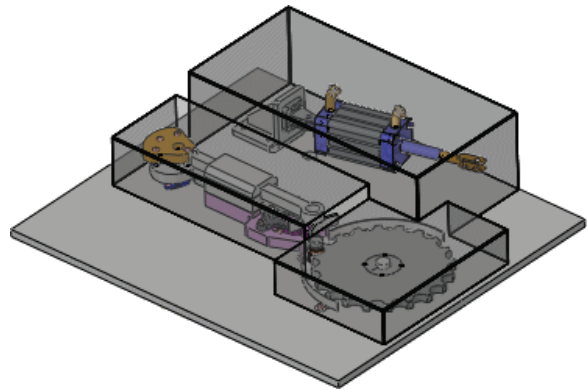
Click **Convert to Sheet Metal**  (Sheet Metal toolbar) or **Insert > Sheet Metal > Convert To Sheet Metal**. In the PropertyManager, under **Corner Defaults**, select a rip type and set the options.

See *SolidWorks Help: Converting a Solid Part to a Sheet Metal Part*.

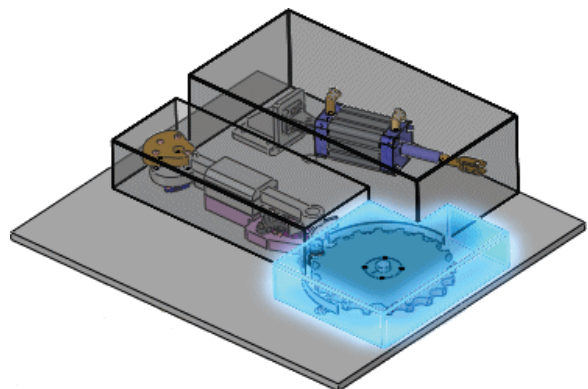
## Keep or Consume Solid Bodies

When using the **Convert to Sheet Metal** tool, you can keep the solid body to use with multiple **Convert to Sheet Metal** features or specify that the entire body be consumed by the tool.

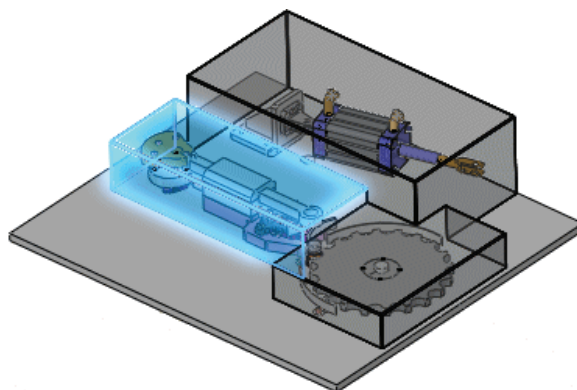
Machine with a sheet metal enclosure.



Use **Convert to Sheet Metal** to create the first sheet metal body. **Keep body** is selected. You want to keep the solid body to derive another sheet metal body from the same solid body.



Use **Convert to Sheet Metal** to create the second sheet metal body. **Keep body** is cleared so the solid body is consumed.



Click **Convert to Sheet Metal**  (Sheet Metal toolbar) or **Insert > Sheet Metal > Convert To Sheet Metal**. In the PropertyManager, under **Sheet Metal Parameters**, select or clear **Keep body**.

## Bend Allowance

You can select the bend allowance method for the developed length calculation of the part. In previous releases, you completed the **Convert to Sheet Metal** command and then changed the bend allowance in a separate step.

Click **Convert to Sheet Metal**  (Sheet Metal toolbar) or **Insert > Sheet Metal > Convert To Sheet Metal**. In the PropertyManager, under **Custom Bend Allowance**, select a bend allowance.

## Callouts

You can show or hide callouts for **Bend Edges**, **Rip Edges found**, and **Rip Sketches**.

Click **Convert to Sheet Metal**  (Sheet Metal toolbar) or **Insert > Sheet Metal > Convert To Sheet Metal**. In the PropertyManager, select or clear **Show callouts**.

## Flat Patterns


Improvements to flattening sheet metal parts make flattening succeed for complex shapes which previously failed. These improvements also provide better flattened geometry for certain corner treatments, lofted bends, and in some cases where cuts intersect bend regions.

You can update existing flat patterns created prior to SolidWorks 2011 to use the improved method.

In the FeatureManager design tree, right-click **Flat-Pattern** and click **Edit Feature**. In the Flat-Pattern PropertyManager, under **Parameters**, select **Recreate flat-pattern**.

## K-Factor in Configurations

When you use K-Factor as the bend allowance type in sheet metal parts, you can specify different values for the K-Factor in different configurations.

In the FeatureManager design tree, right-click **Sheet-Metal1**  and click **Configure feature**. In the dialog box, in **Sheet-Metal1**, select the variable that corresponds to K-Factor. Set the K-Factor values for each configuration.

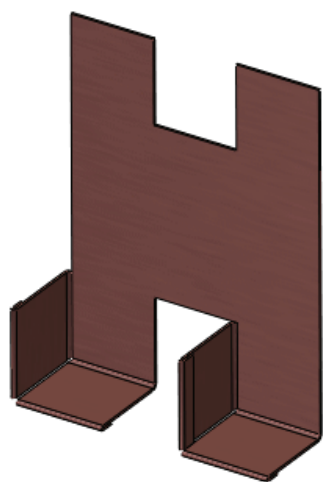
## Mapping Bend Directions When Exporting to DXF/DWG Files

You can map bend line directions to specific layers when you export sheet metal models as **.dxf** or **.dwg** files. For example, in sheet metal parts with up and down bend directions, you can map the different bend line directions to separate layers when you export the part.

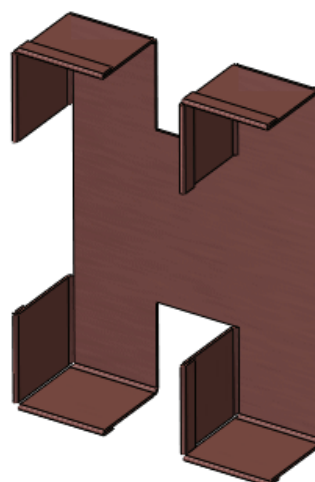
See [Exporting Bend Line Directions](#) on page 81.

## Mirroring Edge Flanges and Miter Flanges

You can mirror edge flanges that consist of several flanges attached to different edges. You can also mirror modified miter flanges where the flanges are shortened so the geometry does not overlap.




Part with multiple edge flanges



Mirrored edge flanges

## Sheet Metal Properties

New properties specific to sheet metal parts are calculated and displayed in the Cut-List Properties dialog box.

Some of the properties that are calculated are based on the bounding box, the smallest rectangle in which the flat pattern can fit. You can set a grain direction to determine the smallest rectangle that aligns with the grain direction to fit the flat pattern. The bounding box is represented by a sketch when you flatten the sheet metal part and is located in the FeatureManager design tree under **Flat-Pattern** .



When exporting a sheet metal part as a **.dxf** or **.dwg** file, you can export the bounding box and assign the bounding box to a specific layer.

See [Exporting Sheet Metal Parts to DXF or DWG Files](#) on page 80.

The following properties are calculated in sheet metal parts:



<b>Bounding Box Length</b>	Longest side of the bounding box
<b>Bounding Box Width</b>	Shortest side of the bounding box
<b>Bounding Box Area</b>	Bounding box length x Bounding box width
<b>Bounding Box Area-Blank</b>	Area of the flat pattern excluding the through cut-outs
<b>Cutting Length-Outer</b>	Outer perimeter of the flat pattern (blank), which is used for calculating the machine's cutting time
<b>Cutting Length-Inner</b>	Sum of the perimeters of internal loops or cut-outs
<b>Cut Outs</b>	Closed cut outs (through holes) on the flat pattern, which are used for calculating the machine's idle time
<b>Bends</b>	Number of bends in the part

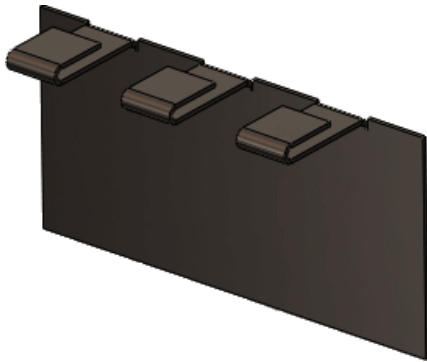
To view these properties, in the FeatureManager design tree, expand **Cut list** . Right-click a **Cut-List-Item**  and click **Properties**. The properties are updated whenever you update the cut list or flatten the part.

See *SolidWorks Help: Sheet Metal Properties* and *Setting the Grain Direction for Bounding Boxes*.

## Patterns of Edge Flanges and Tab Features

You can pattern edge flanges and tab features that have additional sheet metal features attached.



The patterns are supported for the same sheet metal body only. Multibody is not supported. Pattern tools include linear, circular, curve driven, sketch driven, and table driven patterns.



You cannot pattern tab features with attached jogs or sketched bends.

See *SolidWorks® Help: Creating a Grid System*.

The graphics area can appear cluttered when the grid system creates a lot of geometry or if you create multiple grid systems. To reduce clutter, you can hide the geometry. Display options are available to help you visualize the grid structure and to ensure correct grid inferencing.

To access the display options, in the FeatureManager design tree, right-click **Grid System**  and click **View Grid Components**. In the View Grid Components dialog box, select an item and click a display option, such as **Normal To** .

This chapter includes the following topics:

- **New Supported Regions**
- **Sustainability Link for Custom Material**

## New Supported Regions

SolidWorks® Sustainability and SustainabilityXpress support these additional regions: **South America, Australia, and India**. You can specify the regions as **Manufacturing** or **Transportation and Use** regions.

## Sustainability Link for Custom Material

You can perform a sustainability analysis using custom materials. This is done by linking your custom material to a material with similar characteristics in the default SolidWorks Materials database. The analysis is then performed using the Sustainability characteristics of the SolidWorks material.

To link your custom material to a similar material:

1. In the Material dialog box, in a custom materials library, select your custom material.
2. In the Properties tab, in **Material properties**, click **Select**.
3. In the Match Sustainability Information dialog box, select the material that is most similar to your custom material.



Only materials with a link to the Sustainability database are listed.

4. Click **OK**.
5. Click **Apply** and **Close**.

This chapter includes the following topics:




- **Cut Lists**
- **Weld Beads**
- **Weld Support in Drawings**

## Cut Lists

### Cut List Icons

New icons appear in the FeatureManager design tree under **Cut list**.

The icons are based on the type of body in the model:

- Sheet Metal 
- Weldment or structural members 
- Other bodies 

### Reordering and Excluding Cut List Items

You can reorder **Cut-List-Item** folders. Because the order of the **Cut-List-Item** folders drives the cut list entries, you can apply a customized order to the cut list. The reordering propagates to the cut lists in the part and drawing. You can also exclude **Cut-List-Item** folders from cut lists.

To reorder the **Cut-List-Item** folders:

- Drag and drop folders in the FeatureManager design tree.
- Use the Cut-List Properties dialog box to drag and drop folders on the Cut List Summary or Cut List Table tabs.

To exclude **Cut-List-Item** folders from cut lists:

- Right-click a folder in the FeatureManager design tree and click **Exclude from cut list**.
- Use the Cut-List Properties dialog box to select **Exclude from cut list** on the **Cut List Summary** and **Cut List Table** tabs.

See *SolidWorks Help: Excluding Folders from Cut Lists*.




You should finish modeling the part before modifying the cut list items. You may lose changes to the cut list if you change the geometry after modifying the cut list.

## Weld Beads

You can add simplified weld beads to weldment parts and assemblies, and multibody parts.

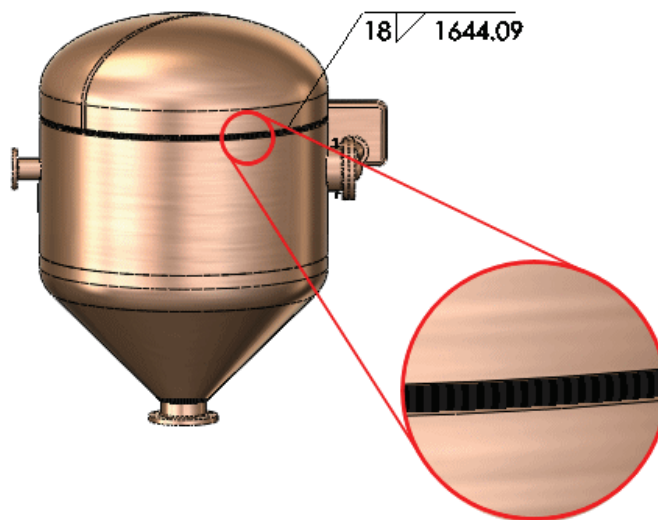
Benefits of simplified weld beads:

- Uniform implementation in parts and assemblies
- Compatibility with all types of geometry, including bodies with gaps
- Lightweight, simplified weld bead display
- Inclusion of weld bead properties in drawings using weld tables
- Smart Weld selection tool for face selection of weld bead paths
- Association of weld bead symbols with the weld beads
- Handles that assist in defining weld paths (lengths)
- Inclusion in the **Weld Folder**  in the FeatureManager design tree

Additionally, you can set properties for **Weld Sub Folders**, including:

- Weld material
- Weld process
- Weld mass per unit length
- Welding cost per unit mass
- Welding time per unit length
- Number of weld passes

Click **Weld Bead**  (Weldments toolbar) or **Insert > Weldments > Weld Bead**. In an assembly, click **Insert > Assembly Feature > Weld Bead**.



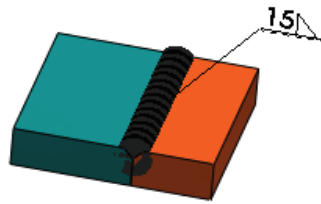
## Weld Bead Display

Weld beads are displayed as graphical representations in models. The weld beads are lightweight and do not affect performance.

## Weld Beads in Assemblies

### Weld Beads

You can add simplified weld beads to assemblies.



In previous versions of SolidWorks®, you added weld beads as components of the assembly. This method is no longer supported. However, you can still edit existing weld bead components.

### Fillets and Chamfers ★

In assemblies, you can create fillets and chamfers, which are useful for weld preparation. As with other assembly features, you can propagate these features to the parts they affect.



### Example: Adding Chamfers and Weld Beads to an Assembly




In this example, you add chamfers to two plates to prepare them for welding. Then you add a simplified weld bead.

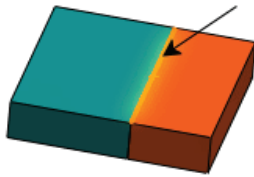
#### Adding a Chamfer to an Assembly Component

First, add a chamfer to the plate on the right.

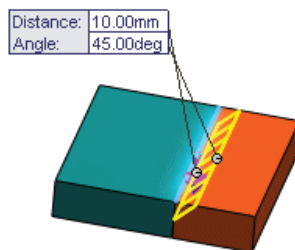
1. Open `install_dir\samples\whatsnew\assemblies\plate_assembly.sldasm`.



2. Click **Assembly Features**  (Assembly tab on the CommandManager) and click **Chamfer** , or click **Insert** > **Assembly Feature** > **Chamfer**.
3. In the PropertyManager, under **Chamfer Parameters**, for **Edges and Faces or Vertex** , do the following to select the edge of plate01:
  - a) Right-click the top edge between the two plates and click **Select Other**.



- b) In the dialog box, select **Edge@[plate01<1>]**.  
A preview of the chamfer appears.



4. Under **Chamfer Parameters**:

- a) Select **Angle distance**.

- b) Set **Distance**  to 5.

- c) Set **Angle**  to 45.



Under **Feature Scope**, you can select **Propagate feature to parts** if you want to add the chamfer to the plate's part file.

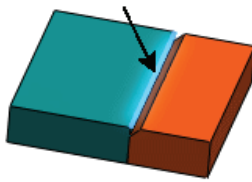
5. Click .  
The chamfer appears.





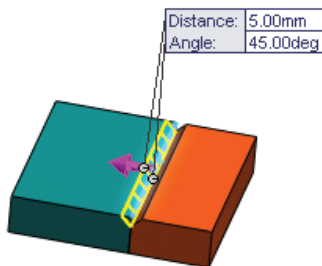
### Adding a Chamfer to a Second Component


Now add a chamfer to the other plate.

1. Select the edge of the plate.



2. Click **Assembly Features**  (Assembly tab on the CommandManager) and click **Chamfer** .  
A preview of the chamfer appears.






3. In the PropertyManager, click .  
The chamfer appears in the second plate.

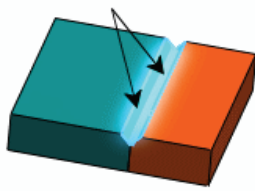


### Adding a Weld Bead

Now add a weld bead to the assembly.

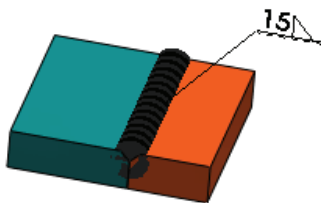
1. Do one of the following:
  - Click **Weld Bead**  (Weldments toolbar).
  - Click **Assembly Features**  (Assembly tab on the CommandManager) and click **Weld Bead** .
  - Click **Insert** > **Assembly Feature** > **Weld Bead**.
2. In the PropertyManager, under **Settings**:
  - a) For **Weld selection**, select the faces of the two chamfers in the graphics area.



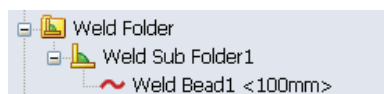


b) Set **Bead size**  to 15.

3. Click .
- The weld bead appears in the graphics area.



In the FeatureManager design tree, **Weld Folder**  appears. Weld bead features that have identical properties are stored together in subfolders. **Weld Bead1**  is the weld you just created.



## Weld Support in Drawings

### Weld Symbols

- In drawings, you can insert bi-directional weld symbols that are attached to weld paths. Click **Insert** > **Model Items** and under **Annotations**, click **Weld Symbols**.
- You can insert weld symbols, caterpillars, and end treatments for weld beads in specific views. Click **Insert** > **Model Items** and under **Source/Destination**, click **Selected feature** for **Source**. Under **Annotations**, click **Weld Symbols**, **Caterpillar**, or **End Treatment**. Move the pointer to highlight a weld path and click to place the annotation. You can also select weld bead features from the FeatureManager design tree while in the **Model Items** command.

### Weld Tables



You can insert weld tables in a drawing that summarize the weld bead data.

By default, weld tables include:

- Weld size
- Symbol
- Weld length
- Weld material

- Quantity

You can add other weld bead custom properties to the table such as weld cost and weld time, and save it as a template for future use.

In a drawing view, click **Weld Table**  (Table toolbar) or **Insert > Tables > Weld Table**. In the PropertyManager, set the options and click .

ITEM NO.	WELD SIZE	SYMBOL	WELD LENGTH	WELD MATERIAL	QTY.
1	3		50	CARBON STEEL	2
2	3		40	CARBON STEEL	2
3	4		20	CARBON STEEL	2
4	4		12.5	CARBON STEEL	2
5	2		15	CARBON STEEL	1